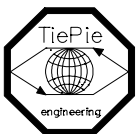


User manual

HANDYSCOPE

a multifunctional
PC measuring instrument



TiePie engineering

ATTENTION!

Measuring directly on the **LINE VOLTAGE** can be very dangerous. The **OUTSIDE** of the **BNC CONNECTORS** at the **HANDYSCOPE** 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 **HANDYSCOPE** is connected to a positive voltage, a short-circuit current will be flowing. Because of this short-circuit current both the **HANDYSCOPE** and the computer can be damaged.

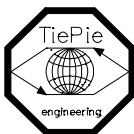
Despite the care taken for the compilation of this user manual, **TiePie engineering** can not be held responsible for any damages resulting from errors that may appear in this book.

All rights reserved. No part of this book may be reproduced, stored in a database or retrieval system, or published, in any form or in any way, electronically, mechanically, by print, photoprint, microfilm or any other means without prior written permission from **TiePie engineering**.

User manual

HANDYSCOPE

a multifunctional
PC measuring instrument



TiePie engineering

Contents

Safety	5
Declaration of conformity	7
Introduction	9
Sampling	9
- Aliasing	10
Digitizing	11
The oscilloscope	12
The spectrum analyzer	13
The voltmeter	14
The transient recorder	15
The probes	16
Hardware installation	17
Software	19
Structure of the program	20
Starting the program	20
Controlling the program	22
- Controlling by mouse	22
- Controlling the main menu	22
- Controlling popup menus	23
Help screens	23
Program settings	24
- Saving instrument settings	24
- Loading instruments settings	24
- Setting the file type for measurement data	24
- Setting the location of files	25
- Setting the default HANDYSCOPE port	25
- Setting the printer port	25
- Setting the printer type	25
- Default text on the printer output	26
- Setting the screen colors	26

Storing measurement data on disk	26
Loading saved measurement data	27
Printing measured data	27
Choosing an instrument	28
Terminating the program	28
Oscilloscope (device live/hold)	29
Aliasing	29
Displaying channels	30
Setting up a channel	31
- Setting the sensitivity of a channel	31
- Setting the vertical position of a channel	31
- Enlarging or reducing the vertical axis	33
- Changing the units of the vertical axis	33
Setting up the time axis	35
- Setting the timebase	35
- Magnification of the horizontal axis	35
Triggering	36
- Selecting the trigger source	37
- Setting the trigger hysteresis	37
- Setting the trigger slope	37
- Adjusting the trigger level	38
- Setting the trigger time-out	39
- Stopping the display	39
Magnifying a part of the screen	40
Storage oscilloscope (device: HOLD)	41
Differences with the oscilloscope	41
Measuring with the cross hair	41
Spectrum analyzer (device: SPECTRUM)	43
Description of FFT	43
- Aliasing	44
- Windowing	44
Similarities with the oscilloscope	49
Displaying channels	49
Setting up a channel	49
- Setting the sensitivity	49
- Adapting the vertical axis	50
Setting up the frequency axis	50

- Magnifying the frequency axis	50
Selecting the window	51
Averaging of spectra	51
Starting a spectrum measurement	52
Measuring a harmonic distortion	52
Calculate the spectrum from the HOLD signal	52
Voltmeter (device: VOLT)	53
Setting up a channel	53
- Setting the input range	53
- Changing the units of measure	54
- Changing the units per volt	54
- Relative measurements	54
- Comparison measurements	55
- Setting the impedance	55
- Setting the gate time	55
- Processing the measured data	55
- Displaying the measured value	57
Setting the frequency range	58
Zeroing the voltmeter	59
Perform measurements	60
Storing measurement values on disk or paper	60
- Setting the time between two measurements	60
- Start an acquisition	61
Changing the display characters	61
Transient recorder (device: TRANS)	63
Setting the measuring type	63
- Setting the frequency range	64
Setting the sampling time	64
Setting the number of samples	65
Clear the screen	65
Start a measurement	65
Go to a specific sample	66
Hardware	67
Connections	67
- Pin definition HANDYSCOPE	67
Hardware diagram	68
Description of the hardware	70

- Setting the input sensitivity	70
- Reading data	71
Description of the parallel printer port	72
Programming the HANDYSCOPE	75
Programming the input sensitivity	75
Programming the A/D convertor	76
Programming example	77
Specifications	79
Hardware	79
- A/D convertor	79
- General	79
Software	80
- Spectrum analyzer	80
- Oscilloscope	81
- True RMS voltmeter	82
- Transient recorder	82

Safety

Before you start working with the HANDYSCOPE, 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 HANDYSCOPE 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 HANDYSCOPE is connected to a positive voltage, a short-circuit current will be flowing. Because of this short-circuit current both the HANDYSCOPE and the computer can be damaged.
- Do not connect the HANDYSCOPE to other ports at your PC than the Centronics parallel printer port. Connecting the HANDY-SCOPE to any other port may damage your HANDY-SCOPE.

Declaration of confirmity

 **TiePie engineering**
Battenserreed 2
9023 AR Jorwerd
Nederland

EG-verklaring van overeenstemming

Wij verklaren geheel onder eigen verantwoordelijkheid, dat het produkt

HANDYSCOPE


waarop deze verklaring betrekking heeft, in overeenstemming is met de geharmoniseerde Europese normen

EN 55011, EN 55022, EN 50081-1 en EN 50082-1

Volgens de bepalingen van de EMC-richtlijn 89/336/EEG, gewijzigd door de richtlijn 92/31/EEG en 93/68/EEG

Jorwerd, 2-1-1996 ir. A.P.W.M. Poelsma



 **TiePie engineering**
Battenserreed 2
9023 AR Jorwerd
The Netherlands

EC declaration of Conformity

We declare, on our own responsibility, that the product


HANDYSCOPE

for which this declaration is valid, is in compliance with

EN 55011, EN 55022, EN 50081-1 and EN 50082-1

according to the conditions of the EMC standard 89/336/EEG, and the amendments 92/31/EEC and 93/68/EEC

Jorwerd, 2-1-1996 ir. A.P.W.M. Poelsma



 **TiePie engineering**
Battenserreed 2
9023 AR Jorwerd
The Netherlands

EG - Konformitätserklärung

Wir erklären, in Eigenverantwortlichkeit, hiermit, daß das Produkt

HANDYSCOPE

für das diese Erklärung gültig ist, mit

EN 55011, EN 55022, EN 50081-1 und EN 50082-1,

gemäß den Anforderungen des EMC-standards 89/336/EEC, und den Zusatzbestimmungen 92/31/EEC und 93/68/EEC übereinstimmt.

Jorwerd, 2-1-1996 ir. A.P.W.M. Poelsma



 **TiePie engineering**
Battenserreed 2
9023 AR Jorwerd
The Netherlands

Déclaration de conformité C.E.

Nous déclarons, sous notre responsabilité, que le produit

HANDYSCOPE

pour lequel cette déclaration est valide, est conforme aux:

EN 55011, EN 55022, EN 50081-1 et EN 50082-1

selon les conditions du standard CEM N° 89/336/EEC, et les amendements 92/31/EEC et 93/68/EEC

Jorwerd, 2-1-1996 ir. A.P.W.M. Poelsma





TiePie engineering
 Battenserreed 2
 9023 AR Jorwerd
 The Netherlands

Dichiarazione di Conformita' CE

Dichiariamo sotto la nostra esclusiva responsabilità che il prodotto:

HANDYSCOPE

per il quale vale la presente dichiarazione, è conforme alle norme

EN 55011, EN55022, EN50081-1 e EN50082-1

conforme alle condizioni della normativa EMC 89/336/EEC, e successive modifiche 92/31/EEC e 93/68/EEC.

Jorwerd, 2-1-1996

ir. A.P.W.M. Poelsma



TiePie engineering
 Battenserreed 2
 9023 AR Jorwerd
 The Netherlands

EC-declaración de conformidad

Nosotros declaramos, bajo nuestra propia responsabilidad, que el producto

HANDYSCOPE

para el cual esta declaración es válida, está en relación con

EN55011, EN 55022, EN 50081-1 y EN 50082-1

Según las condiciones del EMC estándar 89/336/EEC, y los movimientos 92/31/EEC y 93/68/EEC

Jorwerd, 2-1-1996

ir. A.P.W.M. Poelsma



TiePie engineering
 Battenserreed 2
 9023 AR Jorwerd
 The Netherlands

EF-Overensstemmelseserklæring

Undertegnede erklærer herved, at følgende apparat overholder beskyttelseskravene i Rådets direktiv 89/336/EØF om elektromagnetisk kompatibilitet (EMC).

Identification af apparat:

Kategori: Måleinstrument

Model/type: **HANDYSCOPE**

Standarder der er anvendt som grundlag for erklæring, eller henvisning til den prøvningsrapport, der er udstedt af et betydningsfuldt laboratorium:

EN 55011, EN 55022, EN 50081-1 og EN 50082-1

CE-mærket angiver kun overensstemmelse med EMC-direktiv 89/336/EØF

Jorwerd, 2-1-1996

ir. A.P.W.M. Poelsma



TiePie engineering
 Battenserreed 2
 9023 AR Jorwerd
 The Netherlands

EC Hyväksyntäilmoitus

Velvollisuutenamme on ilmoittaa, että tuotteenme

HANDYSCOPE

jota tämä selvitys koskee, on hyväksytty

EN55011, EN55022, EN50081-1 ja EN 50082-1

EMC standardien 89/336/EEG, ja lisästandardien 92/31/EEC ja 93/68/EEC mukaisesti

Jorwerd, 2-1-1996

ir. A.P.W.M. Poelsma



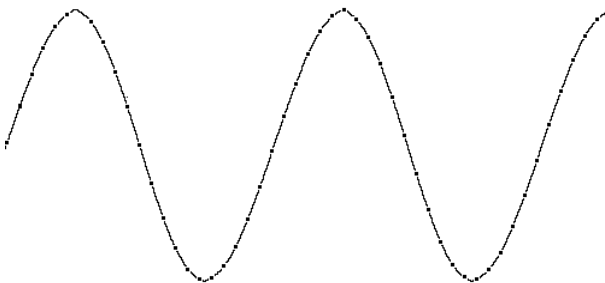
Note before using the HANDYSCOPE, first read the chapter about Safety

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.

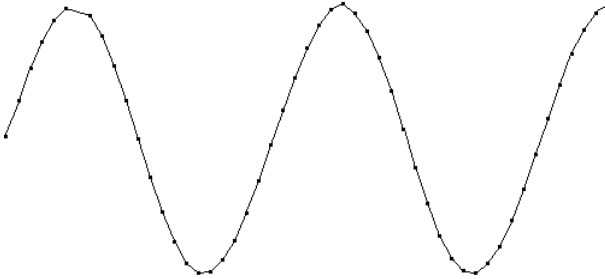
The HANDYSCOPE is a two channel, 12 bits, 100ksamples/sec measuring instrument. With the accompanying software the HANDYSCOPE can be used as an oscilloscope, a storage oscilloscope, a spectrum analyzer, a true RMS voltmeter or a transient recorder. All instruments measure by sampling the input signals, digitizing the values, process them, save them and display them.

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.



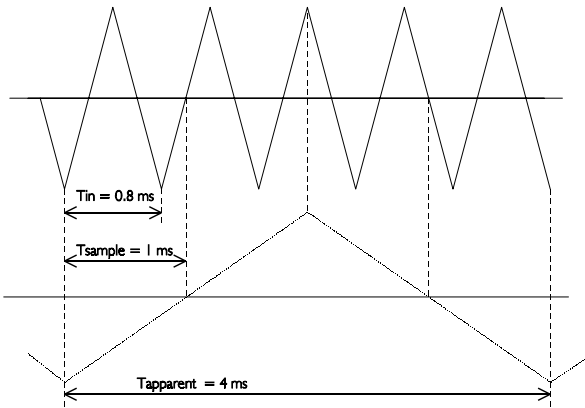
In the latter illustration a sine wave signal is sampled with 50 samples. By connecting the adjacent samples, the original signal can be reconstructed. See also the next illustration.



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

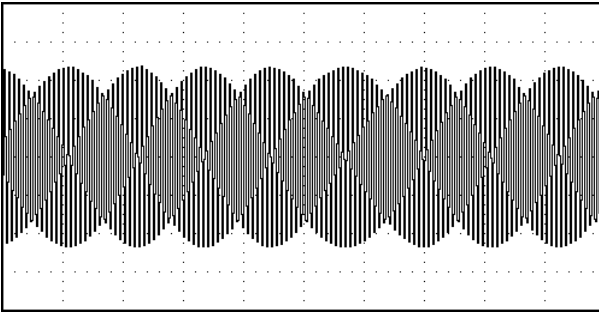
If the sampling frequency is lower than 2 times the frequency of the input signal, 'aliasing' will occur. The following illustration shows how aliasing occurs.



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 ($1.25 \text{ kHz} - 1 \text{ kHz}$).

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 digitised. 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 realised 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 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. Except for the time domain, functions and dependencies of quantities can also be displayed and examined in the X-Y mode.

The HANDYSCOPE oscilloscope has two separated inputs (channels), for which the sensitivity of each channel can be set.

The HANDYSCOPE is equipped with two types of oscilloscopes, LIVE and HOLD. The oscilloscope LIVE measures 480 samples, 1 page. The oscilloscope HOLD measures 4800 samples, 10 pages.

The sensitivity range for the channels goes from 125 mV/div to 5 V/div. The timebase goes from 0.5 msec/div to 2 sec/div.

The display of the oscilloscope is built of 480 x 360 (wxh) pixels, which corresponds with 12x8 (wxh) divisions. Left and right of the display the units of the 2 channels are displayed. Under the display, the time axis is displayed.

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 behaviour of an electrical system.

Not all electrical systems can be characterised in the time domain. Circuits like filters, amplifiers, oscillators, mixers, modulators and detectors can be characterised best by their frequency behaviour. That frequency behaviour 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 in their frequency components, where the voltage of each frequency 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).

The HANDYSCOPE is equipped with a two channel spectrum analyzer. The two channels can be set independently. The spectrum analyzer takes 1024 samples, of which a spectrum of 512 spectral components is calculated by means of FFT.

The sensitivity of the channels can be set from 125 mV/div to 5 V/div. The frequency range of the spectrum analyzer goes from 0.025 Hz to 36 kHz.

The display of the spectrum analyzer is built of 480 x 360 (wxh) pixels, which corresponds with 12x8 (wxh) divisions. Left and right of the display the units of the 2 channels are displayed. Under the display, the frequency axis is displayed.

The voltmeter

If from the input signals only the size is important and not time information or frequency components, the voltmeter is a suitable instrument to measure with. The two channel voltmeter measures the size of the input signal and displays the values using a large 7 segment display. The two independent channels each have their own display.

The voltmeter can measure the input signal in several ways:

TRUE RMS	the true RMS value is measured
PEAK PEAK	the peak peak value is measured
MEAN	the mean value is measured
MAX	the maximum value is measured
MIN	the minimum value is measured
dBm	the value is given in decibel
POWER	the power of the input signal is determined
CREST	the crest factor is calculated
FREQ.	the frequency of the input signal is measured

The input of the voltmeter is autoranging or manually settable between 500 mV full scale and 20 V full scale.

Besides this, the voltmeter has the possibility to measure at certain times and send the data to a printer or write it on disk.

The transient recorder

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

The two channel transient recorder measures the input signals at adjustable times. The time between two measurements is adjustable from 0.01 second to 300 seconds. The number of samples is also adjustable from 1 to 30000. The maximum measuring time is $300 \text{ sec} \times 30000 \text{ samples} = 9000000 \text{ seconds}$ (104.16 days).

The sensitivity of the two channels is adjustable between 125 mV/div and 5 V/div.

The display of the transient recorder is built of 480×360 (wxh) pixels, which corresponds with 12×8 (wxh) divisions. Left and right of the display the units of the 2 channels are displayed. Under the display, the time axis is displayed. In addition, the value, time and date of the samples are given.

The transient recorder measures in one of the following ways:

MOMENTAL	the momentary value of the input signal is measured
TRUE RMS	the true RMS value of the input signal is measured
MEAN	the mean value of the input signal is measured
MAX	the maximum value of the input signal is measured
MIN	the minimum value of the input signal is measured

The probes

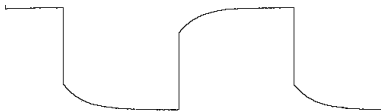
The HANDYSCOPE is shipped with two probes. These are $1\times/10\times$ selectable passive probes. This means that the input signal is passed through directly or 10 times attenuated.

The $\times 10$ 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.

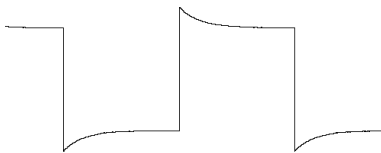
Therefore the probe is equipped with a setscrew, with which the parallel capacity of the attenuation network can be altered. To adjust the probe, switch the probe to the $\times 10$ and attach the probe to a 1 kHz square wave signal. Then adjust the probe for a square front corner on the square wave displayed. See also the following illustration.



correct



undercompensated



overcompensated

Chapter 2

Hardware installation

The HANDYSCOPE is an external measuring instrument which can be connected to a PC.

The HANDYSCOPE is connected to the parallel printer port of the PC using the attached cable. The HANDYSCOPE does not need an external power supply, but is powered by the computer.



Note Switch, to avoid damage at the HANDYSCOPE or the computer, before connecting the cable the computer off.

If your computer is equipped with more than one parallel printer port, you need to setup the software. Refer to the next chapter to do that.

Installation of the software

The software of the HANDYSCOPE is shipped on one diskette.

First you have to make a backup of the original diskette. Refer to your DOS manual for making a backup of a diskette. When you have made the backup, store the original diskette in a safe place and use the backup.

The software is shipped ready to use, without installation program. You can run the software direct from the diskette or from a hard disk.



Hint A hard disk is much faster than a diskette and has usually more free space available to store data.

To install the software on a hard disk, first make a directory for the software. Then copy all files from the diskette to the directory. The files are:

HS.EXE	the program
HS_ENG.HLP	the English help file
HS_ENG.INF	information for the help file
HS.FNT	font file for printing output
HS.DEF	file with program settings

An example, where the diskette is in drive A: and the software has to be installed in a directory called HS on the C: drive.

```
C :  
MD HS  
CD HS  
COPY A:*.>*
```

The software is now installed on the harddisk. After starting the program, the DEFAULT HANDYSCOPE PORT has to be set. This is the printer port the HANDYSCOPE is connected to. Also the DEFAULT DIRECTORIES can be set. These indicate where the program has to store and

look for help files, instrument setting files and measurement data files. You can set the DEFAULT HANDYSCOPE PORT and the DEFAULT DIRECTORIES with the menu **SETTINGS**, in the lower right corner of the screen.

Structure of the program

The program contains five integrated measuring instruments:

- 1 An oscilloscope, device: live.
- 2 A storage oscilloscope, device: hold.
- 3 A spectrum analyzer, device: spectrum.
- 4 A true RMS voltmeter, device: volt.
- 5 A transient recorder, device: trans.

Only one measuring instrument can be active at the same time.

From each measuring instrument another measuring instrument can be activated. Standard features of the above five measuring instruments are options to make print outs and to leave the program to return to DOS.

Starting the program

The program can easily be started according to the below-mentioned procedure:

```
HS par1 par2
```

If no parameters are given, the device LIVE is selected, with the default settings.

The parameters par1 and par2 have the following meaning:

par1 Indicates the device that will be activated after starting the program. Valid values are:

- LIVE : Measuring instrument oscilloscope.
- HOLD : Measuring instrument storage oscilloscope.
- SPECTRUM : Measuring instrument spectrum analyzer.
- VOLT : Measuring instrument true RMS voltmeter.
- TRANS : Measuring instrument transient recorder.

par2 Indicates the filename of the instrument setting (see for writing or reading instrument settings: SETTINGS menu).

Example:

```
HS trans b:HS.set
```

means the measuring instrument transient recorder is activated with the file settings b:HS.set. Incorrect instrument or filenames will be ignored.

The parameters par1 and par2 may be exchanged sequentially, but have to be separated by a space.



Note The HANDYSCOPE software is designed to run under DOS and not under WINDOWS. Running the software under Windows will give incorrect results.

The timebase of the HANDYSCOPE is generated by the computer. If the software is running under Windows, the software can (and will) be interrupted by Windows and other running tasks. This results in 'jumps' in the signal on the screen, since some information of the signal is missed during the interrupt.

To avoid these problems, run the software under DOS. Running from a DOS prompt within Windows will NOT solve the problems because Windows is still active. Quit Windows by pressing <Alt-F4>.

Windows 95 users: Shut down Windows 95 and restart the computer in MS-DOS mode.

Controlling the program

After starting the program, first an intro screen with program information is displayed. This screen indicates what kind of display adapter the program has detected and whether a *Microsoft*® compatible mouse is available.

After the intro screen has disappeared, the normal working screen appears. This screen is divided in two parts: a part for displaying the measurement data and a part for the main menu, for controlling the program. The program can be controlled by the keyboard as well as by a mouse.

Controlling by mouse

When using the mouse, the left mouse button is used for <Enter> and the right mouse button is used for <Esc>.

Controlling the main menu

Each instrument has its own main menu, in the lower part of the screen. The menu is built of three rows of 6 columns.

One of the buttons of the menu is always displayed inverted. That is the cursor, indicating the selected choice. For controlling the main menu, the following keys are available:



places the cursor one row up



places the cursor one row down



places the cursor one column to the left



places the cursor one column to the right



activates the selected choice





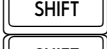

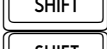

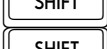

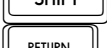





displays a list of instruments

Depending on the choice, after activating, a popup menu appears or the text in the menu changes, as well as the instrument setting.

Controlling popup menus

Various settings in the software are made by means of popup menus. For controlling the popup menus the following keys are valid:

	Move the cursor up	
	Move the cursor down	
	Move the cursor to the top of the menu	
	Move the cursor to the bottom of the menu	
		Move the popup menu to the right
		Move the popup menu to the left
		Move the popup menu up
		Move the popup menu down
	Accept the selected choice	
	Leave the popup menu without changes	
Space	Leave the popup menu without changes	
Character	Place the cursor at the choice beginning with that character	

When a popup menu has been moved by means of the $\langle \rightarrow \leftarrow \uparrow \downarrow \rangle$ keys in combination with the $\langle \text{Shift} \rangle$ key, the new position of the menu will be saved in the file HS.DEF.

Help screens

The program contains integrated help screens. The screens can be invoked at several places in the program, by pressing function key $\langle \text{F1} \rangle$. The invoked help screen gives information about controlling the active program part or information about the active program part.

Program settings

Several parts of the program are configurable. To do that, activate the choice **SETTINGS** from the main menu. A popup menu will appear.

The contents are partly dependent on the type of graphics card in your PC. If an EGA or VGA card is available, two choices for setting the screen colors and the monitor type are present. If no EGA or VGA card is detected, those two choices will not be present.

Settings concerning the program itself (and not the measuring instruments of the program) are saved immediately in the file HS.DEF, which is read at the start of the program.

Saving instrument settings

When the **SAVE** choice is activated all settings of the 5 measuring instruments will be stored in a file to be indicated. The default extension of this file is "SET". At starting the program this file can also be given as parameter.

Loading instruments settings

When the **RESTORE** choice is activated a setting stored by **SAVE** can be restored.

Setting the file type for measurement data

With the choice **SET DATAFILE TYPE** the file type with which the data will be stored on disk can be set. When the **BYTE** option is selected the data file exists of records of 4 bytes (byte 1 and 2 are the value of CH1 and byte 3 and 4 are the value of CH2).

When the **ASCII** option is selected the data file exists of an ascii file of 4 columns, separated by a comma. Column 1 is the sample number, column 2 the sampling time, column 3 the voltage value of CH1 and column 4 is the voltage value of CH2.

Setting the location of files

With the choice **DEFAULT DIRECTORY** the default directories for the HANDYSCOPE, the print files and the data files can be set. The default directory for the HANDYSCOPE is the directory containing the files HS.EXE, HS_ENG.HLP, HS_ENG.INF and HS.FNT.

Setting the default HANDYSCOPE port

With the choice **DEFAULT HANDYSCOPE PORT** the printer port to which the HANDYSCOPE is connected can be indicated.

Setting the printer port

With the choice **DEFAULT PRINTER PORT** the parallel printer ports 1, 2 or 3 to be connected to the printer can be selected.



Note Printer port 1 has the I/O address \$3BC, printer port 2 has the I/O address \$378 and printer port 3 has the I/O address \$278.

When only one printer port is available, a conflict arises when a print command is given. In that case the program asks you to disconnect the HANDYSCOPE from the computer and to connect the printer to the computer. Then the printout is generated. When printing is finished, you are requested to disconnect the printer and to connect the HANDYSCOPE to the computer again. The HANDYSCOPE is then ready to measure again.



Hint to avoid connecting and disconnecting the HANDYSCOPE and the printer often, place a second printer port in your PC

Setting the printer type

With the **SET PRINTER TYPE** choice you can choose between an **EPSON** compatible matrix printer or an **HP-LASER** compatible printer. This setting is used for printing out data.

Default text on the printer output

With **SET USER TEXT** you can fill in three text lines. These text lines are placed at the top on each print out (with the PRINT option). For example, you can fill in your name and address information in these text lines.

The text lines are saved in the HS.DEF file. At the start of the program the text lines are read automatically.

Setting the screen colors

When an EGA/VGA color monitor is available, the colors of the background, the foreground, CH1, CH2 and the cross hair can be set. While changing, the new color is displayed immediately. The color brightness can be changed using the <↔> keys or the mouse.

Also a selection can be made between a color or a black and white display.

Storing measurement data on disk

By activating the **WRITE DISK** choice a file name is requested. A file name without extension has to be entered. The data are stored in filename.DAT and the corresponding instrument setting in filename.GEG.

In case the file name already exists, the program asks to overwrite the existing file. If there is insufficient disk space an error message appears. A file with the extension "DAT" and a file with the extension "GEG" are made during a WRITE DISK. The data format is determined by SET FILE TYPE.



Note The voltmeter has a completely different way of storing measurement data. This will be described in the chapter Voltmeter.

The transient recorder always stores data on disk, so the choice WRITE DISK is not available.

Loading saved measurement data

By activating the choice **READ DISK** a popup menu appears, giving an overview of the data files contained in the indicated default data directory. With the <←↑↓→> keys a file can be selected. When a selection has been made, the data and set files will be read from disk. When the data is not from the measuring instrument only the data file is read, or an error message appears.

The following file operations can be carried out with the function keys:



View the file information (size, date and time)



View the file (only text files)



Delete the file



Rename the file



Copy the file



Enter a new directory mask and/or path and/or disk

When <F4>, <F5> or <F6> is used, both the .DAT file and the .GEG file are acted on.

Printing measured data

There are two ways to print the measurement data.

The first way is pressing <**Shift-PrtSc**>. A quarter page size screen dump is made, containing only the information that is found on the display.

The second way is activated by the menu choice **PRINT**. A popup menu appears, displaying 2 choices.

The **DISK** choice requests for a file name to create the print file. This print file can be sent to the printer using the DOS command: **COPY/B filename PRN**, where *filename* stands for the file name you entered in the input box.

With the **PRINTER** option the screen can be printed out on an EPSON, HP-LASER or a compatible printer. What printer type has been installed is indicated at the bottom of the popup menu. This printout is full page, containing extra information like date and time of the measurement, the comment lines and the USER TEXT.

Comments can be added to the printout using the **COMMENT** choice. When activated, a dialog box appears, in which three lines of text can be entered. These comments entered here are printed together with a printout. The comments are stored on disk together with the data. So each data file has its own comments. The key <F1> gives an overview of valid function keys in the editor.

Choosing an instrument

By activating the choice **DEVICE**, menu a popup menu appears, displaying 6 choices.

When the **QUIT** choice is activated, the program will be terminated. The other options activate the indicated measuring instrument.

The measuring instruments (the devices) to be selected are:

LIVE	=	oscilloscope
HOLD	=	storage oscilloscope
SPECTRUM	=	spectrum analyzer
VOLT	=	voltmeter
TRANS	=	transient recorder

This menu can also be activated by means of the <Esc> key or the right mouse button (if no other menu is active).

Terminating the program

The popup menu **DEVICE** can be invoked from all program sections by means of the <Esc> key or the right mouse button. The **QUIT** option enables you to leave the program and takes you back to the operating system (DOS). The popup menu **DEVICE** can also be invoked with the <Esc> key or the right mouse button, if no other menu is active.

Chapter 4

Oscilloscope (device: LIVE)

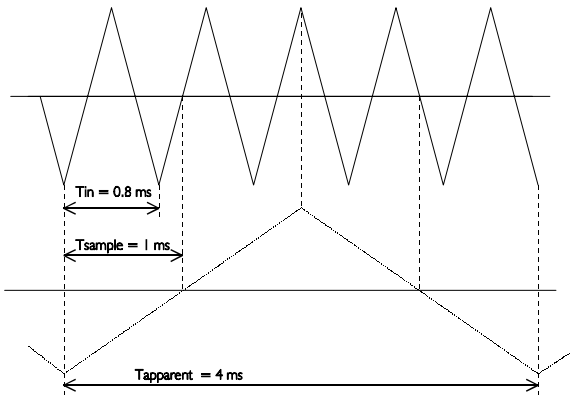
An oscilloscope is a measuring instrument which displays the changes of a voltage in the time domain.

The HANDYSCOPE is equipped with an oscilloscope with 2 inputs (channels), of which the sensitivity of each channel is adjusted separately. 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 480 samples per channel. Between two adjacent samples on the screen a line is drawn. The speed at which the samples are taken, is adjustable.

Aliasing

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

Aliasing occurs when too low and therefore a wrong sampling speed is chosen. In the next illustration is shown 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 HANDYSCOPE has several modes to display the two channels. The mode can be set by activating the choice **MODE** from the main menu.

By activating MODE a popup menu appears with 6 or 7 options:

CH1	Only CH1 is measured and displayed
CH2	Only CH2 is measured and displayed
ALTERNATE	CH1 and 2 are measured and displayed alternating.
CHOP	CH1 and 2 are measured and displayed simultaneously
ADD	CH1 and 2 are measured simultaneously and added. Only the sum is displayed
COMPARE	CH1 is measured and displayed. The latest measured signal at CH2 is displayed, to compare it with CH1.
XY-PLOT	This is a special display mode. CH1 and CH2 are measured in CHOP mode. CH1 is displayed at the x-axis and CH2 at the y-axis of the screen.



Note When the CHOP or XY-PLOT mode is selected and TIME/DIV is at the fastest option, the second fastest option of TIME/DIV will be taken automatically during measuring.

Setting up a channel

The two channels of the HANDYSCOPE oscilloscope are completely configurable. Both channels are identical, each has its own menu in the main menu. Since the possibilities for both channels are equal, only CH1 is explained here. Where CH1 is described, the same applies to CH2.

With the choice **CH1 SET** (or **CH2 SET**) from the main menu a pop-up menu is called.

With the choices from this menu CH1 can be configured.

Setting the sensitivity of a channel

By activating the choice **VOLTS/DIV** a popup menu appears, displaying 6 voltage ranges.

When e.g. **10** has been activated, a voltage range of 2.5 volts/div will be set, that is, the maximum voltage to be measured is ± 10 volt. The **AUTO** option provides for setting the voltage-axis automatically.

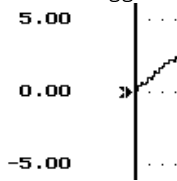
The following ranges can be selected: 0.5, 1, 2, 5, 10 and 20 volt.

Setting the vertical position of a channel

The vertical position of a channel on the screen can be adjusted.

When doing this, the signal is shifted graphically, **after** sampling. This method only has an effect on the displayed signal. If the signal clipped, because it is larger than the selected input range, and is positioned with the software method, the signal remains clipped, but the clipping is now visible on the display. The signal appears to be 'cut off' at either the upper side or the lower side. This positioning method is used for placing two overlapping signals away from each other.

The positioning method is not done by means of a popup menu. At the lefthand side of the screen (in CHOP, ALT, ADD and X-Y for CH2 at the righthand side) two small arrows are visible. The small arrow next to the border of the window is for the position of a channel, the outer arrow is for the trigger level, which is discussed later.



There are two or four arrows, depending on whether one or two channels are displayed. Only one offset (arrow) can be active, displayed normally. All other arrows are displayed inverted. In the above illustration the channel offset arrow for CH 1 is active.

The arrow indicates the offset, the 0 volt level. The offset can be set as follows:



Note To change the offset, first remove all popup menus

SHIFT	↑	Move the offset level up
SHIFT	↓	Move the offset level down
SHIFT	home	Move the offset level to the center
SHIFT	→	Select the next offset or trigger level
SHIFT	←	Select the previous offset or trigger level
SHIFT	page up	Move the offset level to the top
SHIFT	page down	Move the offset level to the bottom
SHIFT	+	Increase the step size (2 times)
SHIFT	-	Decrease the step size (2 times)



Note the step size for changing the offset level or trigger level can be changed with <Shift-+> and <Shift-—>

The offset level can also be changed with the mouse. Place the mouse

cursor on top of the arrow of the offset level to be changed and drag it to the correct position. Drag: press the left mouse button and keep it pressed, while moving the mouse. When the arrow is at the correct position, release the mouse button. The mouse cursor will disappear while dragging the arrow and appear again when the button is released.

Enlarging or reducing the vertical axis

The measured values can be enlarged or reduced by software. Because of that it is possible to magnify the vertical axis. It is also possible to invert the measured values. After activation of the choice **GAIN** in the menu CH1 SET, a dialog box appears in which a multiplication factor can be entered. The factor must be between -5 .. -0.1 or 0.1 .. 5. The measured factors are multiplied by the factors and then displayed. The numbers along the vertical axis are also changed.

The GAIN can be used to invert a signal. Suppose you are measuring two signals and wish to subtract the signal on CH1 from the signal on CH2. To do that you have to set the GAIN for CH1 to -1 and the gain for CH2 to +1. By selecting a measuring mode ADD (CH2 - CH1) is displayed.



Note By choosing a gain of -1, two signals can be subtracted from each other.

An other possibility is displaying two signals of a different size at the same size, to simplify comparison. Suppose you have a 5 volt square wave signal on CH1 and a 4 volt square wave signal on CH2. For displaying both signals at the same size, set the gain for CH1 to 0.8 and for CH2 to 1. Both signals are displayed then at the same size.

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

By means of the choice **UNIT** from the CH1 SET menu the units of measure of the vertical axis can be changed.

After activation, a popup menu with 11 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. The possible choices are:
VOLT, AMPERE, DEGREE F, DEGREE C, WATT, PERCENT, METER, KILOGRAM, NEWTON, COLOMB and BAR.

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

By activating the choice **UNITS/VOLT** from the menu CH1 SET, a multiplication factor can be entered for the units of the vertical axis. A dialog box appears in which the factor is 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/VOLT enter the value 10, to get correct numbers along the vertical axis.

Another application is the previously mentioned example with the 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 **OFFSET** from the CH1 SET 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 again the temperature measurement. Suppose the output voltage of the temperature probe at 0°C is +10 Volt. By entering a UNITS/OFFSET of (-10 x 5), the zero level of the axis is changed in a way that the absolute temperature is directly readable. The extra factor 5 is necessary because of the factor entered with UNITS/VOLT.

Setting up the time axis

Along the horizontal axis of the screen the measuring time is projected.

Setting the timebase

By activating the choice **TIME/DIV**, a popup menu appears, displaying 12 time ranges.

When, for example, the **0.2 sec** choice has been activated, one time division will be equal to 0.2 sec. The screen contains 12 time divisions.

One time division consists of 40 samples. The screen contains $12 * 40 = 480$ sample values. By activating the **0.2 sec** choice one sample is taken every $0.2/40 = 5$ msec.

With the **0.2 sec** choice the minimum duration of the sampling process is $12 * 0.2 \text{ sec} = 2.4 \text{ sec}$. When a key is pressed within these 2.4 sec, the sampling process will be interrupted and the program will respond to the key press.

The following time-axis ranges can be selected: 0.5 msec, 1 msec, 2 msec, 5 msec, 10 msec, 20 msec, 50 msec, 0.1 sec, 0.2 sec, 0.5 sec, 1 sec and 2 sec.



Note On a XT computer the fastest, and sometimes the second fastest timebase setting as well, will not be available. In the CHOP and XY-PLOT measuring modes the fastest and second fastest timebase settings are equal.

Magnification of the horizontal axis

By activating the **TIME-MAG** choice, a popup menu appears, displaying 12 software time-axis magnification areas.

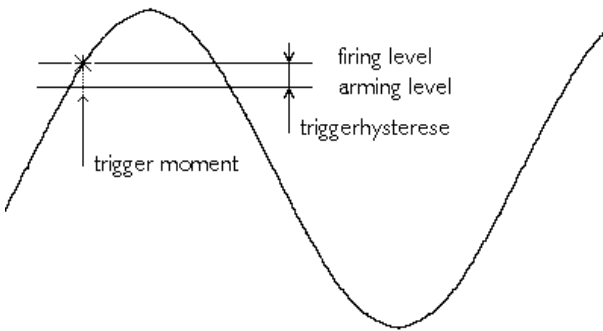
When, for example, the **2*** choice has been activated, the time-axis will be magnified 2 times. The magnified part of the screen is indicated by two interconnected arrows at the bottom of the screen. The time-axis magnification area can be moved using the **<PgUp>** and **<PgDn>** keys or by mouse. Place the mouse cursor on the interconnected ar-

rows, press the left mouse button and drag the arrows left or right. With the <+> and <-> keys the step size can be increased and decreased, respectively.

The following time-axis magnification areas can be selected: 1, 2, 3, 4, 5, 6, 8, 10, 12, 15, 16 and 20 times.

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 circuitry: the arming level and the firing level. When the input signal passes the arming level, the trigger system is armed. If the 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 latter 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

By activating the **TRIG CH1 / TRIG CH2** choice, the text in the menu changes from TRIG CH1 to TRIG CH2 or vice versa.

In the TRIG CH1 position triggering takes place on CH1 and in the TRIG CH2 position on CH2.

Setting the trigger hysteresis

By activating the choice **HYSTERESIS**, a popup menu appears, displaying 12 trigger hysteresis options.

For example, when the **0.5 DIV** option has been activated, the trigger hysteresis will be 0.5 division of the voltage range set. This means that the signal has to come out at a minimum of 0.5 division below the trigger level set (for a positive trigger slope). When the signal has much noise the trigger hysteresis can be set high to ensure that triggering is satisfactory.

When a signal has less noise, the trigger hysteresis can be set low allowing triggering at low signal levels. For setting the trigger level, see: offset and trigger levels.

The following trigger hysteresises can be selected: 0 DIV, 0.125 DIV, 0.25 DIV, 0.5 DIV, 0.75 DIV, 1.0 DIV, 1.25 DIV, 1.5 DIV, 1.75 DIV, 2.0 DIV, 2.25 DIV and 2.5 DIV.

Setting the trigger slope

By activating the choice **SLOPE+ / SLOPE-**, the text in the menu choice changes from SLOPE+ to SLOPE- or vice versa.

In the SLOPE+ position triggering takes place on a rising slope of the signal to be measured and in the SLOPE- position on a falling slope.

Adjusting the trigger level

The adjustment of the trigger level is not done by means of a popup menu. At the lefthand side of the screen (in CHOP, ALT, ADD and X-Y for CH2 at the righthand side) two small arrows are visible. The small arrow next to the border of the window is for the position of a channel, the outer arrow is for the trigger level.

There are two or four arrows, depending on whether one or two channels are displayed. Only one arrow can be active, displayed normally. All other arrows are displayed inverted. The arrow indicates the trigger level. The trigger level can be set as follows:



Note To change the trigger level, remove all popup menu's

SHIFT	↑	Move the trigger level up
SHIFT	↓	Move the trigger level down
SHIFT	home	Move the trigger level to the center
SHIFT	→	Select the next offset or trigger level
SHIFT	←	Select the previous offset or trigger level
SHIFT	page up	Move the trigger level to the top
SHIFT	page down	Move the trigger level to the bottom
SHIFT	+	Increase the step size (2 times)
SHIFT	-	Decrease the step size (2 times)



Note the step size for changing the offset level or trigger level can be changed with **<Shift-+>** and **<Shift-—>**

The trigger level can also be changed with the mouse. Place the mouse cursor on top of the arrow of the trigger level to be changed and drag it to the correct position. Drag: press the left mouse button and keep it pressed, while moving the mouse. When the arrow is at the correct position, release the mouse button. The mouse cursor will disappear while dragging the arrow and appear again when the button is released.

Setting 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 circuitry 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, 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.

By activating the choice **TIME-OUT** from the main menu, a popup menu appears with 12 choices for the trigger time-out: 0 DIV, 10 DIV, 20 DIV, 50 DIV, 100 DIV, 150 DIV, 200 DIV, 250 DIV, 300 DIV, 350 DIV, 400 DIV and 450 DIV.

For example, when the **50 DIV** option has been activated, the signal will be controlled for 50 time divisions maximally. If the signal does not comply with the trigger level set, the trigger slope and the trigger hysteresis within 50 time divisions, it will nevertheless be measured after 50 time divisions.

Stopping the display

By activating the **LIVE/FREEZE** choice, the text in the menu changes from LIVE to FREEZE or vice versa.

In the LIVE position the oscilloscope continuously takes 480 samples and displays them on the screen.

In the FREEZE position no samples are taken and the samples last taken are displayed on the screen. A **ONE SHOT** menu is added in the FREEZE position.

By activating the **ONE SHOT** menu the text in the menu changes from **ONE SHOT** to **MEASURE**. After measuring the signal the text changes back to **ONE SHOT**. When a key is pressed during measuring (MEASU-

RE), the sampling process will be terminated immediately. Samples taken up to the moment the key is pressed can be viewed (the remaining samples are 0).

Magnifying a part of the screen

By activating the choice **ZOOM**, a popup menu appears, displaying 2 options.

The **FULL** choice displays the signal fully. (The x-axis and y-axis magnifications are set at 1 and the offset levels at 0.)

With the **WINDOW** option a rectangular magnification area can be indicated using the <←↑↓→> keys or the mouse. One corner has to be indicated first, followed by the <Enter> key or the left mouse button. Then the second corner has to be indicated, followed by the <Enter> key or the left mouse button. Cancelling the zooming can always be done by means of the <Esc> key or the right mouse button.

Differences with the oscilloscope

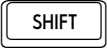

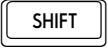

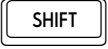

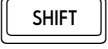

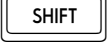

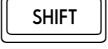

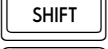

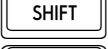

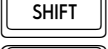

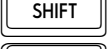



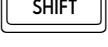

The storage oscilloscope HOLD is quite similar to the oscilloscope LIVE, there are a few differences, which will be described here. For the rest is referred to the chapter Oscilloscope (LIVE).

- The HOLD scope only works in ONE SHOT mode and does not measure 1 but 10 pages of 480 samples. With the <PgUp> and <PgDn> keys, the pages are selected. The step size can be set using the <+> or <-> keys.
- The HOLD scope does not provide for XY-PLOT
- After measuring, a cross hair appears, to simplify voltage and time measurements

The main menu of the HOLD device differs slightly from the main menu of the LIVE device. At the position where in the LIVE device the LIVE / FREEZE choice is found, the HOLD device has a choice ONE SHOT.

Measuring with the cross hair

After the measured signal is displayed, a crosshair appears. With the crosshair absolute and relative time and voltage measurements can be performed. Controlling the crosshair is done with the following keys:

		Setting the absolute zero
		Setting the relative zero
		Move the crosshair to zero
		Move the crosshair up
		Move the crosshair down
		Move the crosshair to the left
		Move the crosshair to the right
		Mirror the crosshair on the relative x-axis
		Mirror the crosshair on the relative y-axis
		Decrease the step size twice
		Increase the step size twice
		Switching between crosshair controll and offset/ trigger level controll.

Absolute zero is that point at the screen where voltage and time are zero (left most, in the center of the screen). Relative zero is that point set by moving the crosshair to the intended point and subsequently pressing **<Shift-Del>**.

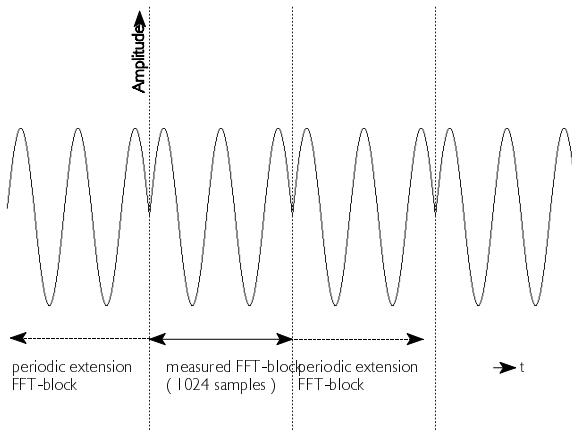
The time and voltage values according the crosshair position are displayed on the screen, the voltage value in the upper right corner of the screen and the time value at the right most end of the time axis. The values are displayed inverted.

The crosshair can also be moved by means of the mouse. This is done by placing the mouse cursor at the crosshair, press the left mouse button and move the mouse, keeping the left mouse button pressed. While positioning the crosshair with the mouse, the mouse cursor disappears. When the correct position is achieved, the left mouse button can be released and the mouse cursor appears again.

The (storage)oscilloscope measures and displays signals in the time domain: the voltage is plotted against the time. It is also possible to display measurement data in the frequency domain, the data is then plotted against the frequency. For this method the spectrum analyzer is used.

Description of FFT

The spectrum analyzer is using the Fast Fourier Transform (FFT) principle. For this transform, 1024 samples are taken with a previously set sample speed. With these 1024 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.

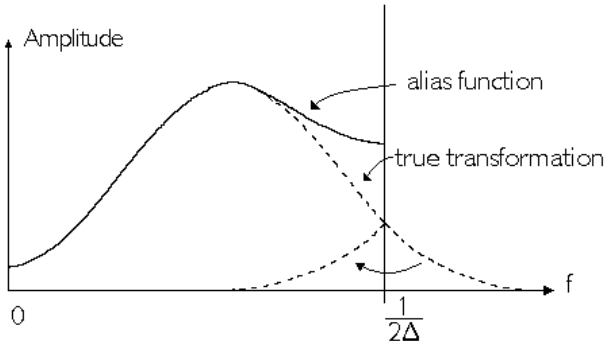


From the signal obtained like this, the spectrum is calculated. The FFT calculation results in 512 spectral components. These spectral components together form the frequency spectrum of the input signal. 480 of the 512 spectral components are displayed, in 12 divisions of 40 spectral components each.

The HANDYSCOPE 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

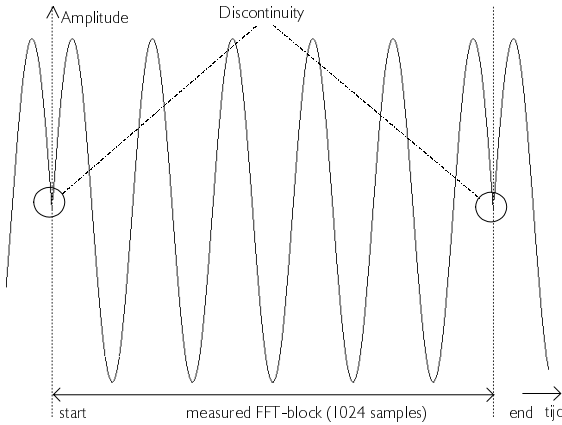
Like the oscilloscope, the spectrum analyzer needs a high enough sampling frequency to avoid aliasing. The result of aliasing when using FFT is that the frequency curve is folded back on itself. See also the next illustration.



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

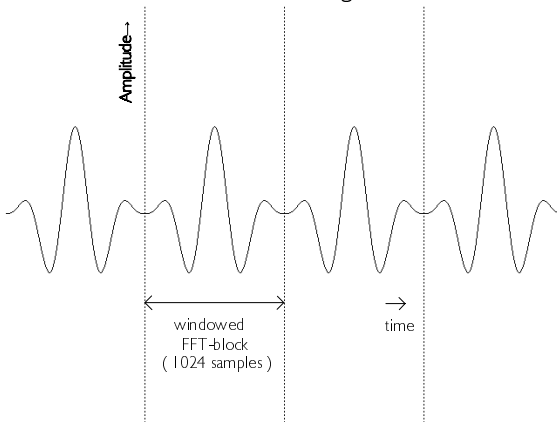
Windowing

FFT treats the FFT-block of 1024 samples 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.

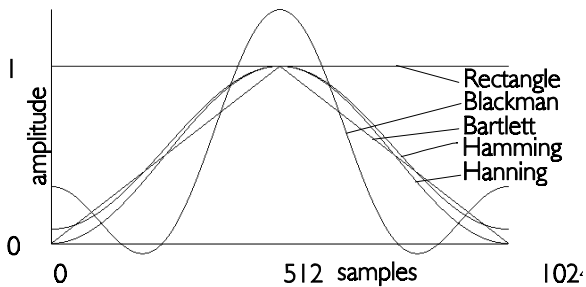
The smearing, caused by the calculation errors, can be decreased by placing a window on the FFT-block of 1024 samples, in such a way that the ends of the FFT-block are smoothly tapered to zero and discontinuities do not occur when the FFT treats the windowed block as one period of a periodic signal. 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. See also the following illustration.



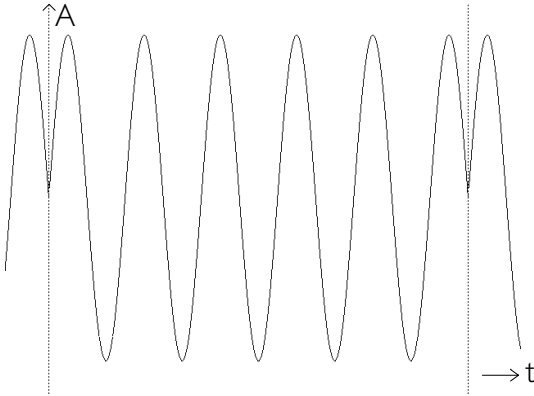
The software supports five different windows:

- rectangular (all data remains the same)
- Hanning
- Hamming
- Blackman
- Bartlett

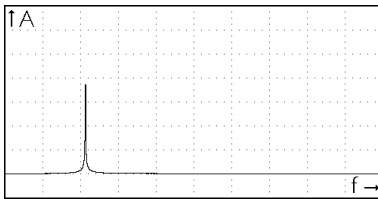
A graphical representation of the different windows is given in the next illustration.



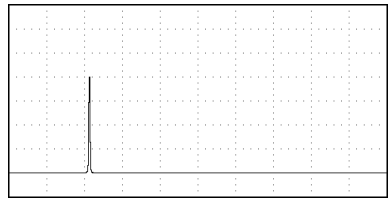
The effects of the different windows are displayed next, together with the sinewave signal the spectrum is taken from. The sinewave does not fit in the FFT-block and will therefore cause calculation errors.



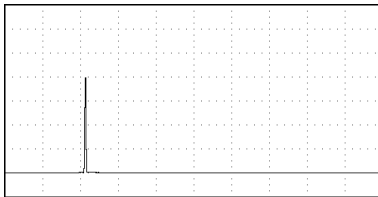
The measured sine wave



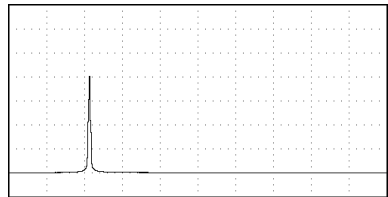
Rectangular window



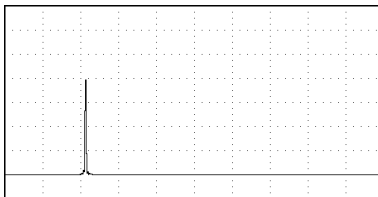
Hanning window



Hamming window



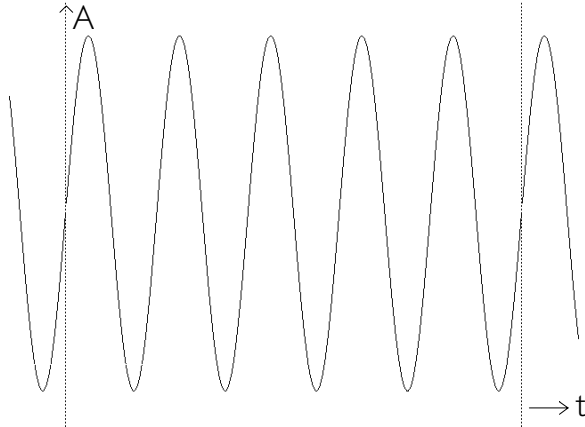
Blackman window



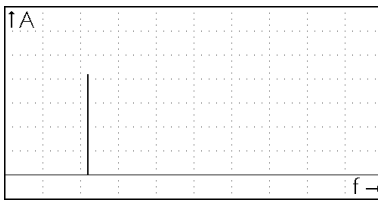
Bartlett window

When using the rectangular window, the amplitude of the spectrum is lower than with the other windows. Smearing is larger with the rectangular window.

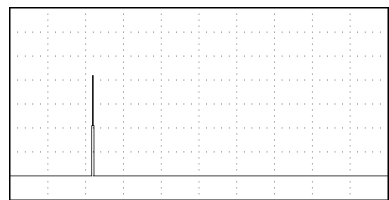
Again some example illustrations, now with a sine wave exactly fitting in the FFT-block.



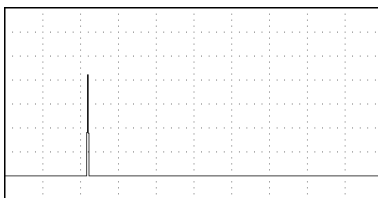
The measured sine wave



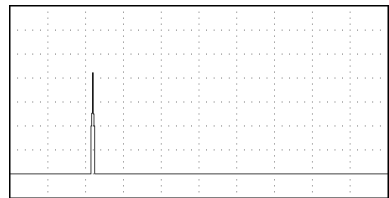
Rectangular window



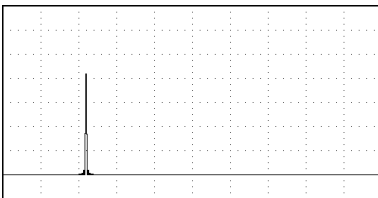
Hanning window



Hamming window



Blackman window



Bartlett window

Similarities with the oscilloscope

The spectrum analyzer has some similarities with the oscilloscope. The bottom row in the main menu contains choices that are similar to the ones in the oscilloscope devices. They will not be discussed here.

Displaying channels

The spectrum analyzer has several ways to display the two channels. By activating **MODE** from the main menu, a popup menu containing five choices appears. The spectrum analyzer does not have the display option X-Y PLOT. The other choices are equal to the choices for the oscilloscope. See the chapter Oscilloscope for an explanation of the possibilities.

Setting up a channel

The two channels of the spectrum analyzer can be set up to your own demands. For both channels equal settings are available, each through their own choice in the main menu. The available settings are equal to the settings for the oscilloscope channels, except for a few points. These points will be discussed here, the other points will be referred to the chapter about the oscilloscope. All that is discussed here, is valid for both channels.

Activating the choice **CH1 SET** from the main menu, gives a popup menu which is almost equal to that of the oscilloscope, except for the last line. The oscilloscope provides for a choice **UNIT**, the spectrum analyzer for a choice **LINEAR/dB**.

Setting the sensitivity

The sensitivity of a channel can be set in exactly the same way as with the oscilloscope. It can be done by means of the choice **VOLTS/DIV**.

Adapting the vertical axis

With the spectrum analyzer you can decide whether the vertical axis will be linear or logarithmic. This is done with the setting **LINEAR** or **dB** from the menu CH1 SET. The current setting is displayed by the text of the choice. Activating the choice will make the text and setting change.

On a logarithmic axis, 0 dB corresponds with 1 Volt.

Setting up the frequency axis

By activating the choice **FREQUENCY**, a popup menu appears, displaying 12 frequency ranges.

For example, when the **1200 Hz** option is activated, the maximum frequency displayed by the spectrum analyzer will then be 1200 Hz. The resolution is $(1200\text{Hz}/480)$ 2.5 Hz.

The following frequency ranges can be selected: 12 Hz, 24 Hz, 48 Hz, 120 Hz, 240 Hz, 480 Hz, 1200 Hz, 2400 Hz, 4800 Hz, 12 kHz, 18 kHz and 36 kHz.



Note On a XT computer, the highest, and sometimes the second highest frequency range are not available. In ALT and CHOP mode, the highest frequency range equals the second highest frequency range.

Magnifying the frequency axis

By activating the **FREQ-MAG** choice, a popup menu appears, displaying 12 magnification areas.

When, for example, the **2*** option has been activated, the frequency-axis will be magnified 2 times. The magnified part of the screen is indicated by two interconnected arrows at the bottom of the screen.

The frequency-axis magnification area can be moved using the **<PgUp>** and **<PgDn>** keys or the mouse by placing the mouse cursor on the interconnected arrows and clicking the left mouse button (while moving

the mouse button must be kept pressed). With the <+> and <-> keys the step size can be increased and decreased, respectively.

The following frequency-axis magnification areas can be selected: 1, 2, 3, 4, 5, 6, 8, 10, 12, 15, 16 and 20 times.

Selecting the window

By activating the choice **WINDOW**, a popup menu appears, displaying 5 window types.

These windows are used to reduce calculating errors during the FFT transformation. See at the beginning of this chapter for an explanation about FFT and windows.

The following window types can be selected: rectangle(= no window), Hanning, Hamming, Blackman, Bartlett.

Averaging of spectra

When the spectrum of a non-periodic signal (for example noise) is measured, **AVERAGE** can be used.

By activating the choice **AVERAGE**, a popup menu appears, displaying 8 options. When **AVERAGE** is set to 10, 10 spectra will be measured, calculated and averaged. After calculating each spectrum the results are displayed immediately. The averaging process can be interrupted by pressing the <**Esc**> key or the right mouse button.

The following choices for averaging are available: 1, 2, 5, 10, 20, 50, 100 and 200.

Starting a spectrum measurement

The sampling process is started by activating the choice **MEASURE**.

On the screen the text "MEASURING, PLEASE WAIT ..." appears. After the sampling process the spectrum is calculated using FFT and the message "CALCULATE SPECTRUM 1, PLEASE WAIT ... (Esc; Exit)" appears on the screen. When there are more calculations set by means of the **AVERAGE** menu, the first text will be displayed on the screen followed by the second one.

Measuring a harmonic distortion

By activating **DISTORTION**, a frequency can be indicated by means of the <←↑↓→> keys or the mouse.

The indicated frequency is considered the fundamental frequency.

The total harmonic distortion relative to this fundamental frequency is determined on the basis of the formula:

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

V_1 is the RMS amplitude of the fundamental frequency. $V_2 \dots V_{10}$ are the RMS amplitudes of each harmonic. The RMS amplitude of each harmonic is shown in a table.

Calculate the spectrum from the HOLD signal

By activating **HOLD->FFT**, the spectrum of the first 1024 measuring points of the signal measured with the HOLD device is calculated. The frequency-axis and the voltage-axes are adjusted to the value of the HOLD device.

Chapter 7

Voltmeter (device: VOLT)

The HANDYSCOPE is equipped with a two channel voltmeter with digital display.

The voltmeter functions as follows:

- 200 points are sampled.
- The measured data is processed, e.g. for calculating the RMS value or the mean value. 9 options can be carried out.
- The calculated values are displayed, e.g. add CH1 and CH2 and display on CH1. 16 different display modes are available.

Because of the computations and display modes there are many options (± 100) to measure a signal with the voltmeter.

Setting up a channel

As noticed, the channels can be configured in many ways. Both channels have equal settings, so only CH1 is discussed here. Activating the choice **CH1 SET** brings up a popup menu in which the settings for CH1 are available.

Setting the input range

By activating the **VOLTS/DIV** choice, a popup menu appears, displaying 6 voltage ranges.

When the **10** option has been activated, a sensitivity of ± 10 volt full scale will be set. This corresponds with 2.5 V/div. The **AUTO** option provides for setting the voltage-axis automatically.

The following voltage ranges can be selected: 0.5, 1, 2, 5, 10, and 20 volt.

Changing the units of measure

The units of measure displayed, can be configured to your own demands. By activating the choice **UNIT**, a popup menu with the choices. When the requested option is activated, this unit will be placed under the display behind the MEASURING mode. At an ACQUIRE TO DISK (or an ACQUIRE TO PRINTER) the unit is placed behind the MEASURING mode as well.

Changing the units per volt

By activating the choice **UNITS/VOLT**, you can enter a factor for the display. The value filled in will be multiplied with the measured value.

Example: A thermocouple generating 1 volt per 5 degrees is measured. Then the number of UNITS/VOLT can be set at 5 allowing the direct reading of the correct number of degrees.

When an oscilloscope probe which attenuates 10 times is used, the value 10 can be entered with the UNITS/VOLT option, so the correct voltage values can be read directly.

Relative measurements

By activating the **RELATIVE = ON/OFF** choice, the text changes from RELATIVE = ON to RELATIVE = OFF or vice versa.

The RELATIVE = ON option results in decreasing the displayed value with the value given at the SET REL. VALUE option.

Example: MEASURING is set on MEAN and DISPLAY is set on CH 1. When a DC voltage of 10 volt is offered, the voltmeter will point at 10 volt. When the value 10 is filled in with the SET REL. VALUE option and RELATIVE = ON is set, the voltmeter will point at 0 volt.

With the RELATIVE = OFF option the measured value is not influenced.

Comparison measurements

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

- > then HI
- < then LO
- > < COMP
- < > LO HI

These measurements require a HIGH and a LOW value. With the choices **SET HIGH VALUE** and **SET LO VALUE** the two values can be set. Activating the choices will bring up a dialog box in which the size of the level can be entered in volts.

Setting the impedance

The dBm and POWER measurements require a reference resistance. The value of the resistance can be set by activating **SET IMPEDANCE** from the CH1 SET menu.

The value has to be entered in Ohm. The default value is 600 Ω .

Setting the gate time

By activating the choice **SET GATE TIME**, a gate time value can be selected. The gate time set is used for measuring the frequency. When the gate time is set at 0.1 sec, the measuring time is 0.1 sec and the frequency resolution 10 Hz. At a gate time of 1 sec the frequency resolution is 1 Hz and determining the frequency takes 1 sec. The voltmeter will then respond less quickly on the key board and the mouse.

Processing the measured data

By activating **MEASURING**, a popup menu appears, displaying 9 calculations.

The option chosen determines how the data of CH1 (or CH2) will be calculated.

The following options can be selected:

True RMS: The true RMS value of the sampled data is calculated. The true RMS value (Root Mean Square) is the value of a casual voltage form, identical to a DC voltage dissipating the same power in a resistance.

An AC voltage of 1 volt (RMS) is identical to a DC voltage of 1 volt. (Most -cheap- voltmeters only point the right value at a true sine voltage when an AC voltage is measured. The HANDYSCOPE always points the correct value at the true RMS handling.)

PEAK TO PEAK

The peak to peak voltage of the sampled data is determined. The peak to peak voltage of a DC voltage is zero. The peak to peak voltage of a true sine with a RMS value of 1 volt is 2.828 volt.

MEAN:

The mean value of the sampled data is determined. The mean value of the DC voltage of 1 volt is 1 volt. The mean value of a true sine with a RMS value of 1 volt is 0 volt.

MAX:

The maximum value of the sampled data is determined. The maximum value of a DC voltage of 1 volt is 1 volt. The maximum value of a true sine with a RMS value of 1 volt is 1.41 volt.

MIN:

The minimum value of the sampled data is determined. The minimum value of a DC voltage of 1 volt is 1 volt. The minimum value of a true sine with a RMS value of 1 volt is -1.41 volt.

dBm:

The value of the sampled data is converted into decibels. Converting is done by means of the formula:

$$dBm = 10 * \log \frac{1000 * value^2}{reference\ resistance}$$

With "value" the RMS value of the measured data is meant. The reference resistance can be set with the SET IMPE-DANCE option.

POWER: The value of the sampled data is converted into a power. Converting is done by means of the formula:

$$POWER = \frac{value^2}{reference\ resistance}$$

With "value" the RMS value of the measured data is meant. The reference resistance can be set with the SET IMPEDANCE option.

CREST: The value of the sampled data is converted into a crest factor. Converting is done by means of the formula:

$$CREST = \frac{peak\ value}{RMS\ value}$$

The "peak value" is the peak voltage relative to the zero line of the signal.

FREQ: The frequency of the offered signal is determined. The accuracy and the speed of calculating the frequency is determined by the "gate time".

Displaying the measured value

By activating the **DISPLAY** choice, a popup menu appears, displaying 16 options.

The option chosen determines how the data of CH1 or CH2 will be displayed. The options are:

CH1: Display data of CH1.

CH2: Display data of CH2.

CH1*CH2: Display data of CH1 multiplied with data of CH2.

CH1/CH2: Display data of ch 1 divided by data of CH2.

CH2/CH1: Display data of ch 2 divided by data of CH1.

CH1-CH2: Display data of ch 1 minus data of CH2.

CH2-CH1: Display data of ch 2 minus data of CH1.

CH1+CH2: Display data of ch 1 plus data of CH2.

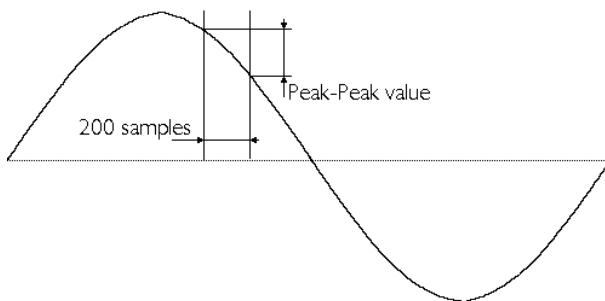
> then: The data is compared with the HIGH VALUE. When the data is higher than HIGH VALUE, **PASS** is displayed. When the data is smaller than HIGH VALUE, **LO** is displayed.

- < then:** The data is compared with the LOW VALUE. When the data is lower than LOW VALUE, **PASS** is displayed. When the data is higher than LOW VALUE, **HI** is displayed.
- >< COMP:** The data is compared with the LOW VALUE and the HIGH VALUE. When the data is higher than HIGH VALUE, **HI** is displayed. When the data is lower than LOW VALUE, **LO** is displayed. When the data is lower than or equal to HIGH VALUE **AND** higher than or equal to LOW VALUE, **PASS** is displayed.
- <> Hi Lo:** The data is compared with the LOW VALUE and the HIGH VALUE. When the data is higher than HIGH VALUE, **PASS** is displayed. When the data is lower than LOW VALUE, **PASS** is displayed. When the data is lower than or equal to HIGH VALUE **AND** higher than or equal to LOW VALUE, **LO** is displayed.
- MAX:** Display the maximum value of the data.
- MIN:** Display the minimum value of the data.
- LOG(1/2):**
- $$20 * \log \frac{\textit{measured value CH1}}{\textit{measured value CH2}}$$
- LOG(2/1):**
- $$20 * \log \frac{\textit{measured value CH2}}{\textit{measured value CH1}}$$

Setting the frequency range

The voltmeter has a limited frequency range in which the measurements are performed correctly. This is caused by the fact that the voltmeter takes 200 samples, using a certain sampling frequency.

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 short in relation to the periodical time of the input signal, **aliasing** can occur (see also the chapter about the oscilloscope).

To avoid errors like the one mentioned above, the frequency range of the voltmeter has to be set properly. The sampling frequency is then changed. By activating the **FREQUENCY** choice, a popup menu appears, displaying 7 center frequencies. A frequency range is determined from $0.2 \times f_{\text{center}}$ to $10 \times f_{\text{center}}$.

For example, when **50 Hz** is chosen, signals with a frequency between 10 Hz to 500 Hz will be calculated correctly.

The following center frequencies can be selected: 10 Hz, 50 Hz, 100 Hz, 500 Hz, 1 kHz, 5 kHz and 10 kHz.

Zeroing the voltmeter

The **CALIBRATE** choice allows the zeroing of the voltmeter. By activating CALIBRATE, the following text appears on the screen

Connect probe input to ground and press key (Esc: Exit)

When a key is pressed, the mean value of all input voltage ranges will be measured. This value will be subtracted from the following measurements.

Perform measurements

By activating the **MEASURE/FREEZE** choice from the main menu, the text in the menu changes from MEASURE to FREEZE or vice versa.

In the MEASURE position 200 sample values are taken and from that a determined quantity will be calculated. The quantity to be determined can be set with the CHI SET menu, MEASURING option. The last measured values are displayed in the FREEZE position.

By activating the choice **ONE SHOT** from the main menu one measurement is taken and from that the data will be calculated and displayed. This menu is only available when the MEASURE/FREEZE choice is in the FREEZE position.

Storing measurement values on disk or paper

The voltmeter can store measurement values on disk or print them on paper. This is done in a completely different way to 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.

Setting the time between two measurements

By activating the choice **TIME** from the main menu, the time between two measurements can be set. A dialog box appears in which the time can be entered, in seconds. The time has to be between 0 and 300 seconds.



Note When a time **shorter than 0.5 seconds** is entered, the acquisition speed is determined by the changes of display 1 or display 2. Each time a measurement value at the screen changes, the measured value is stored on disk or printed.

Start an acquisition

By activating the choice **ACQUIRE**, a popup menu is displayed, from which a selection can be made whether the data has to be stored on disk or sent to the printer.

The choice **DISK** will cause the measurement data to be stored on disk. After activating the choice, a dialog box appears in which a name for the file has to be entered. If a file with that name exists, you will be asked whether that file can be overwritten. The data in that file will be lost. After confirmation, measuring starts. At the fixed times a measurement is taken and stored on disk. Measuring can be cancelled by pressing <**Esc**>.

The choice **PRINTER** will cause the measurement data to be sent to a printer. After activating, measuring starts. At the fixed times a measurement is taken and the value is sent to the printer. Measuring can be cancelled by pressing <**Esc**>.

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 higher than the selected input range, '-----' is displayed. This will also be stored on disk or printed.

Changing the display characters

By activating the **DIGIT SET** choice, a popup menu appears, displaying 6 digit thicknesses.

For example, when the **4-PIXELS** choice has been activated, the digit thickness will be 4 pixels.

The following digit thicknesses can be selected: 2, 4, 6, 8, 10 and 12 pixels.

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

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



Note The transient recorder displays a measured sample immediately, the oscilloscope takes all samples and then displays the complete page.

The transient recorder main menu has several items that are similar to the oscilloscope, therefore they are not discussed in this chapter.

Setting the measuring type

By activating **MEAS. MODE**, a popup menu appears, displaying 5 measuring methods.

MOMENTAL	the momentary value will be measured
true RMS	the true RMS (= effective) value will be measured
MEAN	the mean value will be measured
MAX	the maximum value of the signal will be measured
MIN	the minimum value of the signal will be measured

The true RMS, MEAN, MAX and MIN values are calculated from ± 200 samples. To avoid measuring errors the frequency of the measured signal must be around the center frequency set (See also Setting the frequency range in the chapter Voltmeter).



Note When true RMS, MEAN, MIN or MAX is selected, the minimum sampling time will be 0.5 sec.

Setting the frequency range

By activating the **FREQUENCY** choice, a popup menu appears, displaying 7 center frequencies. A frequency range is determined from $0.2 \times f_{\text{center}}$ to $10 \times f_{\text{center}}$.

For example, when the **50 Hz** option has been activated, the signals with a frequency of 10 Hz to 500 Hz will be calculated correctly.

The frequency setting is only effective when True RMS (= effective), MEAN, MAX or MIN values are measured.

The following center frequencies can be selected: 10 Hz, 50 Hz, 100 Hz, 500 Hz, 1 kHz, 5 kHz and 10 kHz.



Note When the 10 Hz center frequency has been selected the minimum sampling time will be 1.5 sec.

Setting the sampling time

By activating **SAMPLE TIME** the sampling time between two measured values can be given.

For the sampling time a value between 0.01 and 300 sec can be given. Then the total measuring time can be $300 \times 30,000 = 9,000,000$ sec (= 104.16 days).



Note On a PC-XT the minimum sampling time is 0.05 sec.



Note When no momentary values are taken (see MEAS. MODE) the minimum sampling time is 0.5 sec, or 1.5 sec at a frequency setting of 10Hz

Setting the number of samples

By activating the **SAMPLES** menu, the number of samples can be given. For the number of samples a value between 1 and 30,000 can be given. Then the total measuring time can be $300 \cdot 30,000 = 9,000,000$ sec (= 104.16 days).

Clear the screen

By activating the **CLEAR SCR** choice, the screen of the transient recorder is erased.

Start a measurement

When all settings are set, a measurement can be started, by activating **MEASURE** from the main menu. First a name has to be entered for the file the data will be stored in. If the name of an existing file is entered, you will be asked whether this file can be overwritten by the new data. The old data will be destroyed.

When the name is entered, sampling starts. This is indicated by a window with the text:

```
MEASURING..., Esc: interrupt measuring.
```

At fixed times a sample is taken, until the number of samples to be taken is reached. While sampling, the sample number, the time and the measured value are displayed inverted.



Note When the **HANDYSCOPE** is connected to a relative slow computer, it may occur that, depending on the selected sampletime, the sample number, time and value are not being displayed. Only if enough time is available, the items are displayed.

When the total measurement time is completed, the window with the text disappears and the data is saved on disk.

A crosshair, used for measuring, now appears. See the chapter oscilloscope HOLD for information about controlling the crosshair. During measuring with the crosshair, the value, the sample number and the time of the selected sample are displayed.

When more than 480 samples are taken, the keys <PgUp> and <PgDn> provide switching between the pages with data. The current page number is displayed in the upper right corner of the screen. With <+> and <-> the step size can be changed, <+> increases the step size, <-> decreases the step size.

As with the oscilloscope, the data can be shifted up and down on the screen. This is done in the same way as with the oscilloscope.

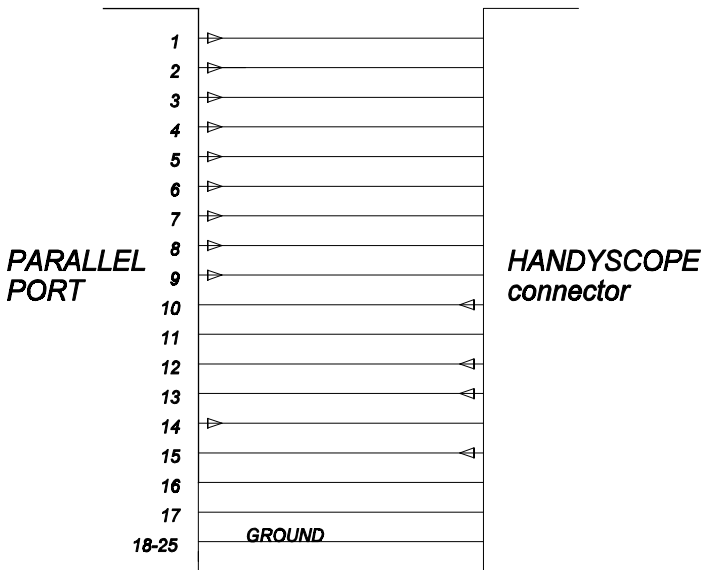


Note When switching between pages of data, or shifting the signal, each time the complete signal is read from disk, which may take some time.

Go to a specific sample

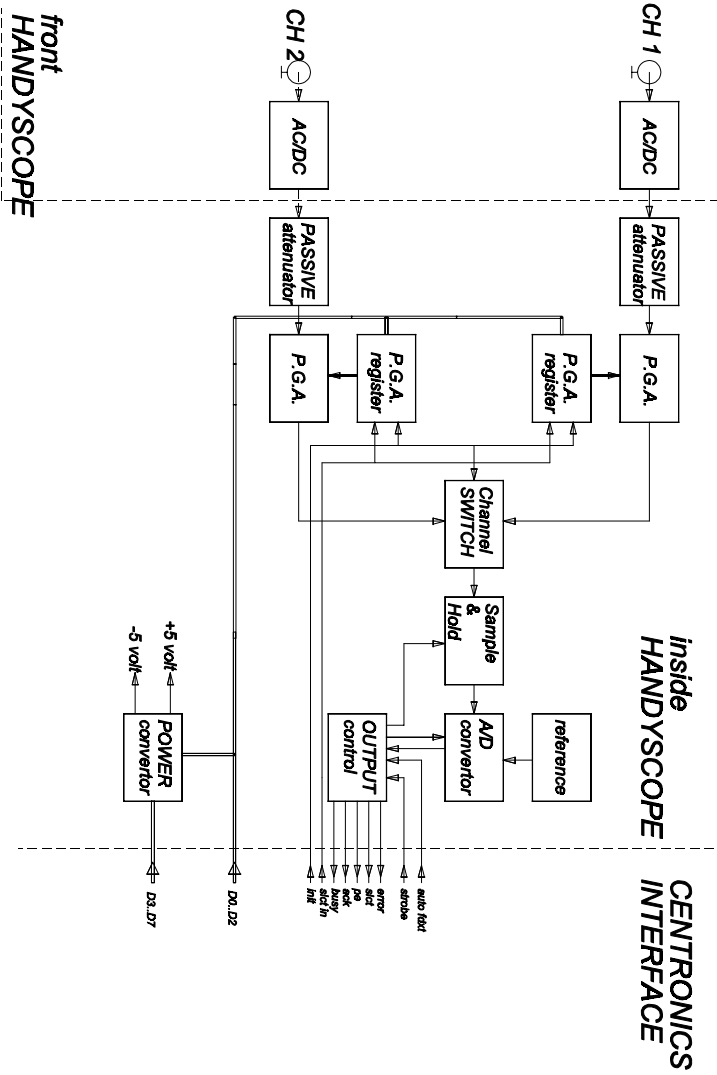
By activating the **GO TO** choice a sample number to be jumped to can be given. For example, when a measurement has been made involving 15,000 values, then the 15,000th sample value can be jumped to straight away by entering 15,000.

Connections



Pin definition HANDYSCOPE

1:	A1	10:	databit 10 or 5 or 1
2:	select gain factor 10*	11:	databit 11 or 6
3:	select gain factor 2*	12:	databit 9 or 4 or 0
4:	select gain factor 2*	13:	databit 8 or 3
5:	power supply +3 .. 5 volt (4 mA)	14:	A0
6:	power supply +3 .. 5 volt (4 mA)	15:	databit 7 or 2
7:	power supply +3 .. 5 volt (4 mA)	16:	CH1/CH2
8:	power supply +3 .. 5 volt (4 mA)	17:	CLOCK
9:	power supply +3 .. 5 volt (4 mA)	18-25:	ground



The signal to be measured is offered to the HANDYSCOPE by means of the BNC connectors on the front. AC or DC voltage can be coupled through to the signal to be measured using an AC/DC switch. With the switch in the AC position the DC voltage component in the input signal is blocked.

Then the signal is attenuated by a passive attenuator which also protects the HANDYSCOPE against too high input voltages. Behind the attenuator a so-called PGA (Programmable Gain Amplifier) is located. This is an active amplifier of which the gain can be programmed by software. Here the different input sensitivities are created.

The gain is determined by the digital value contained in the PGA register. The PGA's are available for both CH1 and CH2.

The outputs of both PGAs go to an analog multiplexer enabling CH1 or CH2 to couple through to the next device, the SAMPLE AND HOLD. The SAMPLE AND HOLD provides for not changing the input voltage of the A/D convertor during converting, this prevents conversion errors.

The analog output signal of the SAMPLE AND HOLD is converted into a 12 bits binary code by the A/D convertor.

The REFERENCE device provides for a stable reference voltage, this is necessary for the A/D convertor to convert accurately.

The OUTPUT CONTROL enables starting a conversion and determines what group bits of the A/D convertor can be read in by the PC.

The POWER CONVERTOR provides for the +5 and -5 volt power supply of the HANDYSCOPE.

Measuring directly on the LINE VOLTAGE can be dangerous. The OUTSIDE of the BNC CONNECTORS at the HANDYSCOPE are connected with the GROUND of the computer. Use a good isolation transformer or a differential probe when measuring on the LINE VOLTAGE or on GROUNDED POWER SUPPLIES.

Description of the hardware

The HANDYSCOPE contains a fast 12 bits A/D convertor with a SAMPLE AND HOLD amplifier. Each channel is executed with a separate programmable amplifier. All programmable amplifiers have a register (the PGA register). They can each be set at 0.5, 1, 2, 5, 10 and 20 volt bipolar voltage (full scale).

The HANDYSCOPE is connected to the parallel printer port of the PC and needs no external power supplies. The necessary power is delivered by the data bits (D0..D7) of the parallel printer port. By setting the data bits to **1** the HANDYSCOPE is supplied with power. The voltage of the data bits varies from 2.4 to 4.8 volt. This voltage is converted into a stabilized voltage of +5 and -5 volt in the HANDYSCOPE.

The data bits D0..D2 are used for setting the input sensitivity of the HANDYSCOPE.

For controlling the HANDYSCOPE the lines **INIT**, **-AUTO FDXT**, **-STROBE** and **-SLCT IN** are used.

The control lines **-AUTO FDXT** and **-STROBE** are used for controlling the A/D convertor.

The control line **-SLCT IN** is used for loading the PGA register and the control line **INIT** for selecting CH1 (**INIT** is low) or CH2 (**INIT** is high).

Setting the input sensitivity

For setting the input sensitivity of the HANDYSCOPE two registers are used. Each register contains a bit pattern of 3 bits wide indicating the input sensitivity of the channel. Each channel has a separate input sensitivity setting. When measuring two channels alternately, the input sensitivity has only to be set once for each channel. CH1 can be set at a sensitivity of, for example, 20 volt while CH2 can be set at, for example, 0.5 volt.

The data bits D0..D2 are used for setting the input sensitivity. The table at the next page indicates how the input sensitivity can be set.

data bits			input sensitivity
D0	D1	D2	
0	0	0	0.5 volt
0	0	1	1.0 volt
0	1	1	2.0 volt
1	0	0	5.0 volt
1	0	1	10 volt
1	1	1	20 volt

The table above concerns both CH1 and CH2.

The control line **-SLCT IN** is used for loading the PGA register of CH1 or CH2.

The control line **INIT** is used for selecting CH1 or CH2.

When **INIT** is high, CH1 is on.

When **INIT** is high and **-SLCT IN** is taken from low to high, the data bits D0..D2 will be loaded in the PGA register of CH1.

When **INIT** is low, CH2 is on.

When **INIT** is low and **-SLCT IN** is taken from low to high, the data bits D0..D2 will be loaded in the PGA register of CH2.

Reading data

The control lines **-AUTO FDXT** and **-STROBE** are used for starting a conversion of the A/D convertor and for reading the data bits of the A/D convertor (AD_D11..AD_D0).

When **-AUTO FDXT** and **-STROBE** are both low, the data bits AD_11..AD_7 of the A/D convertor will be available on **-BUSY**, **ACK**, **PE**, **SLCT** and **ERROR**.

When **-AUTO FDXT** is high and **-STROBE** low, the bits AD_6..AD_2 of the A/D convertor will be available on **-BUSY**, **ACK**, **PE**, **SLCT** and **ERROR**.

When **-AUTO FDXT** is low and **-STROBE** high, the bits AD_1 and AD_0 of the A/D convertor will be available on **ACK** and **PE**.

Starting a conversion is done at the moment **-AUTO FDXT** is taken from low to high and **-STROBE** is low.

The conversion value of the previous conversion can be read during converting.

Control lines		data of the A/D convertor				
-AUTO FDXT	-STROBE	-BUSY	ACK	PE	SLCT	ERROR
0	0	AD_D11	AD_D10	AD_D9	AD_D8	AD_D7
0	1	AD-D6	AD_D5	AD_D4	AD_D3	AD_D2
1	0		AD_D1	AD_D0		
1	1					

Description of the parallel printer port

The HANDYSCOPE is connected to the parallel printer port of the computer. In case you want to program the HANDYSCOPE by yourself look at the control lines of the parallel printer port, which are shortly mentioned next.

A personal computer cannot contain more than 3 parallel printer ports. These printer ports are default at the basic I/O port addresses \$3bc, \$378 and \$278 (\$ means hexadecimal).

Because the HANDYSCOPE is connected to the parallel printer port, programming the HANDYSCOPE is not easy. Some bits are inverted at the parallel printer port, reckon with this when programming. In the examples below is assumed that the HANDYSCOPE will be connected to the parallel printer port being on the I/O port address \$3bc. In the examples the programming language Turbo Pascal is used.

Programming the input sensitivity

The input sensitivity of each channel is determined by the bit combination of the first 3 data bits of the parallel port. These 3 bits have to be loaded in the PGA register. The control line **INIT** indicates if CH1 is active (**INIT** is high = 1) or CH2 (**INIT** is low = 0). The registers are loaded when **-SLCT IN** is taken from low to high. **-SLCT IN** will be low by setting the third bit on the I/O port address \$3be at 1 (= high). **-SLCT IN** will be high by setting the third bit on the I/O port address \$3be at 0 (= low). In Turbo Pascal this is programmed as followed:

Loading the PGA register of CH1:

```
port[$3bc]:=253;    { this is the input sensitivity of 10 volt}
port[$3be]:=15;    {-SLCT IN will be low and INIT will be high}
                    { CH1 is active}
port[$3be]:=4;     {-SLCT IN will be high, so the PGA register}
                    { of CH1 is loaded with the bit pattern 101}
port[$3bc]:=255;   { set all databits at 1 again for a correct}
                    { power supply}
```

Loading the PGA register of CH2:

```
port[$3bc]:=248;   { this is the input sensitivity of 0.5 volt}
port[$3be]:=11;   { -SLCT IN and INIT will be low}
                    { CH2 is active}
port[$3be]:=0;    {-SLCT IN will be high, so the PGA register}
                    { of CH2 is loaded with the bit pattern 000}
port[$3bc]:=255;   { set all databits at 1 again for a correct}
                    { power supply}
```

Programming the A/D convertor

The parallel printer port has the ability of reading in 5 bits at the same time. For reading **ERROR**, **SLCT**, **PE**, **ACK** and **-BUSY** are used. The **-BUSY** signal is inverted in the parallel port and is used for reading bits 11 and 6. These bits are inverted inside the HANDYSCOPE, so the fact that the **-BUSY** signal is inverted can be overlooked in the software. The **-AUTO FDXT** and **-STROBE** signals are for controlling the A/D convertor. The signals **-AUTO FDXT** and **-STROBE** are set at the same time as **INIT** and **-SLCT IN**, keep this in mind.

Starting a conversion and reading the previous conversion value of CH1 is as followed:

```
port[$3be]:=5;    {start new conversion and load previous values}
AD_MI      :=port[$3bd];    {read D2..D6}
port[$3be]:=4;
AD_LO      :=port[$3bd];
port[$3be]:=6;    {set D0..D1 at ERROR, SLCT, PE, ACK, -BUSY}
AD_LO      :=port[$3bd];    {read D0..D1}
port[$3be]:=7;    {set D7..D11 at ERROR, SLCT, PE, ACK, -BUSY}
AD_HI      :=port[$3bd];    {read D7..D11 in}
AD_VALUE   :=(AD_HI shl 4)+(AD_MI shr 1)+(AD_LO shr 5);
```

Starting a conversion and reading the previous conversion value of CH2 is as followed:

```
port[$3be]:=1;    {start new conversion and load previous values}
AD_MI      :=port[$3bd];    {read D2..D6}
port[$3be]:=0;
AD_LO      :=port[$3bd];
port[$3be]:=2;    {set D0..D1 at ERROR, SLCT, PE, ACK, -BUSY}
AD_LO      :=port[$3bd];    {read D0..D1}
port[$3be]:=3;    {set D7..D11 at ERROR, SLCT, PE, ACK, -BUSY}
AD_HI      :=port[$3bd];    {read D7..D11}
AD_VALUE   :=(AD_HI shl 4)+(AD_MI shr 1)+(AD_LO shr 5);
```

Programming example

Reading data from the HANDYSCOPE (Turbo Pascal).

```
program test;

uses crt;

const HsIOaddress = $378;

var Ch1Value, Ch2Value : integer;

procedure SetInputSensitivityCh1( Sens : byte );
begin
  port[HsIOaddress ]:=Sens;           {          input sensitivity}
  port[HsIOaddress+2]:=0 +4;         {          INIT high, select CH1}
  port[HsIOaddress+2]:=11+4;        {          -slct in low}
  port[HsIOaddress+2]:=0 +4;        {-SLCT IN high and load gain}
  port[HsIOaddress ]:=255;         {          turn power on}
end;

procedure SetInputSensitivityCh2( Sens : byte );
begin
  port[HsIOaddress ]:=Sens;           {          input sensitivity}
  port[HsIOaddress+2]:=0;            {          INIT low, select CH2}
  port[HsIOaddress+2]:=11;          {          -SLCT IN low}
  port[HsIOaddress+2]:=0;          {-SLCT IN high and load gain}
  port[HsIOaddress ]:=255;         {          turn power on}
end;

procedure Read12bitData( var Ch1Value, Ch2Value : word);
var i, Lo, Mi, Hi, Value, Dummy : integer;
const Counter = 4;
begin
  port[HsIOaddress+2]:=7;
  port[HsIOaddress+2]:=5;           { start conversion at ch 1}
  i := Counter;
  repeat
    dec( i );
  until i <= 0;                    {          delay 10 microsec}

  port[HsIOaddress+2]:=7;
  Hi :=port[HsIOaddress+1];         {          read D7..D11 of CH2}
  port[HsIOaddress+2]:=5;
  Mi :=port[HsIOaddress+1];         {          read D2..D6of CH2}
  port[HsIOaddress+2]:=0;
  Dummy:=port[HsIOaddress+1];       {          read nothing}
  port[HsIOaddress+2]:=2;
  Lo :=port[HsIOaddress+1];         {          read D1..D0 of CH2}
  Ch2Value:=(Hi and 248) * 16 +
            (Mi and 248) div 2 +
            (Lo and 96) div 32;

  port[HsIOaddress+2]:=3;           { start conversion at ch 2}
  i := Counter;
  repeat
    dec( i );
  until i <= 0;                    {          delay 10 microsec}

  Hi :=port[HsIOaddress+1];         {          read D7..D11 of CH1}
```

```

port[HsIOaddress+2]:=1;
Mi :=port[HsIOaddress+1];           {      read D2..D6 of CH1}
port[HsIOaddress+2]:=4;
Dummy:=port[HsIOaddress+1];       {      read nothing}
port[HsIOaddress+2]:=6;
Lo :=port[HsIOaddress+1];         {      read D1..D0 of CH1}
Ch1Value:=(Hi and 248) * 16 +
           (Mi and 248) div 2 +
           (Lo and 96) div 32;

end;

{ M A I N   P R O G R A M }

begin
port[HsIOaddress]:=255;           {      turn power on}
SetInputSensitivityCh1(248);      { sensitivity 0.5 volt}
SetInputSensitivityCh2(255);     { sensitivity 20 volt}
repeat
  Read12bitData( Ch1Value, Ch2Value );
  writeln( Ch1Value:8, Ch2Value:8 );
until KeyPressed;
end.

```

Chapter 11

Specifications

Hardware

A/D convertor

sampling frequency		100 kHz
conversion time		10 μ sec
input sensitivity (for CH1 and 2)		
	500 mvolt	(resolution 244 μ volt)
	1 volt	(resolution 488 μ volt)
	2 volt	(resolution 976 μ volt)
	5 volt	(resolution 2.44 mvolt)
	10 volt	(resolution 4.88 mvolt)
	20 volt	(resolution 9.76 mvolt)
maximum overload voltage		200 volt AC 50 Hz
maximum overload voltage 1:10 probe		600 volt AC 50 Hz
input impedance		1 Mohm/20 pF
coupling		AC/DC
resolution		12 bits 0.025 %
linearity		10 bits
accuracy		0.25 % \pm 1 LSB
bandwidth		DC to 50 kHz

General

operation temperature		0°C .. +50 °C
storage temperature		-10°C .. +70 °C
relative humidity		5 % .. 90 %
dimensions		
	height	1.5 inch / 40 mm
	length	5.7 inch / 145 mm
	width	3.3 inch / 85 mm
weight		approx. 350 gram
cable length		approx. 67 inch (1.7 m)

Spectrum analyzer

frequency range	0.025 - 12 Hz (resolution 0.025 Hz)
	0.05 - 24 Hz (resolution 0.05 Hz)
	0.1 - 48 Hz (resolution 0.1 Hz)
	0.25 - 120 Hz (resolution 0.25 Hz)
	0.5 - 240 Hz (resolution 0.5 Hz)
	1.0 - 480 Hz (resolution 1.0 Hz)
	2.5 - 1200 Hz (resolution 2.0 Hz)
	5.0 - 2400 Hz (resolution 5.0 Hz)
	10.0 - 4800 Hz (resolution 10.0 Hz)
	25.0 - 12000 Hz (resolution 20.0 Hz)
<i>(not at a PC-XT)</i>	37.5 - 18000 Hz (resolution 37.5 Hz)
<i>(maybe not at a PC-XT)</i>	5.0 - 36000 Hz (resolution 75.0 Hz)
frequency-axis magnification	2 ... 20 times
y-axis	linear/dB
windows	rectangle window
	Hanning window
	Hamming window
	Blackman window
	Bartlett window
distortion	measuring 10 harmonics of the ground frequency
hardcopy	matrix and laser printer

Oscilloscope

time base	<i>(not at a PC-XT)</i>	0.5 msec/div
	<i>(maybe not at a PC-XT)</i>	1.0 msec/div
		2.0 msec/div
		5.0 msec/div
		0.01 sec/div
		0.02 sec/div
		0.05 sec/div
		0.10 sec/div
		0.20 sec/div
		0.50 sec/div
		1.00 sec/div
		2.00 sec/div
accuracy of time base		5 %
time-axis magnification		2..20 times
setting accuracy time-axis magnification		0.25 %
y-axis magnification		-5..5 times
triggering	CH1, CH2, positive slope, negative slope	
trigger sensitivity		0.25..2.25 DIV
trigger resolution		0.01 times full scale
trigger time-out		0..50 DIV
external triggering		keyboard
offset setting	CH1 and CH2 separately	
offset range		0..100 %
offset resolution		0.01 times full scale
invert	CH1 and CH2 separately	
add		CH1 + CH2
compare		CH1 relative to CH2
xy-mode	CH1 = x-axis, CH2 = y-axis	
hardcopy		matrix and laser printer

True RMS voltmeter

measuring method	true RMS, mean, min, max, momentary, dBm, power, crest factor and frequency values
display abilities:	CH1, CH2, CH1*CH2, CH1/CH2, CH2/CH1, CH1-CH2, CH2-CH1, CH1+CH2, > then, < then, >< COMP, <>HI LO, MAX, MIN, LOG (CH1/CH2) and LOG (CH2/CH1)
display height	± 1..2 inch (±25..50 mm) depending on the monitor
frequency range	10 Hz, 50 Hz, 100 Hz, 500 Hz, 1 kHz, 5 kHz, 10 kHz
voltage range	500 mvolt (resolution 244 μvolt) 1 volt (resolution 488 μvolt) 2 volt (resolution 976 μvolt) 5 volt (resolution 2.4 mvolt) 10 volt (resolution 4.8 mvolt) 20 volt (resolution 9.7 mvolt)
maximum overload voltage	200 volt peak to peak
maximum overload voltage 1:10 probe	600 volt peak to peak
data acquire	to disk and to printer

Transient recorder

measuring time between two measuring points	0.01..300 sec at a PC-XT the minimum time is 0.05 sec
maximum number of measuring points to be taken	30,000
time-axis magnification	2..20 times
gain factor	-5..5 times
offset factor	-10,000..10,000
measuring method	true RMS, mean, min, max and momentary
hardcopy	matrix and laser printer

If you have any suggestions and/or remarks regarding the program, the HANDYSCOPE or the manual, please contact:

✉ **TiePie engineering**
P.O. Box 290
8600 AG SNEEK
The Netherlands

△ TiePie engineering
Koperslagersstraat 37
8601 WL SNEEK
The Netherlands
Tel.: +31 515 415 416
Fax: +31 515 418 819

Email: info@tiepie.nl

Site: www.tiepie.nl