User manual

TP112

a multifunctional
PC measuring instrument

2 analog inputs, 12 bits

8 digital outputs

8 digital inputs

TiePie engineering

User manual

TP112

a multifunctional
PC measuring instrument

2 analog inputs, 12 bits

8 digital outputs

8 digital inputs

TiePie engineering

Contents

Declaration of Confirmity

TiePie engineering
Battenserreed 2
9023 AR Jorwerd
Nederland

EG-verklaring van overeenstemming

Wij verklaren geheel onder eigen verantwoordelijkheid, dat het produkt

TP112

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

EN55011, EN55022, EN50081-1 en EN50082-1

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 Confirmity

We declare, on our own responsibility, that the product

TP112

for which this declaration is valid, is in compliance with

EN55011, EN55022, EN50081-1 and EN50082-1

according 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

TP112

für das diese Erklärung gültig ist, mit

EN55011, EN55022, EN50081-1 und EN50082-1,

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 responsibilité, que le produit

TP112

pour lequel cette déclaration est valide, est conforme aux:

EN55011, EN55022, EN50081-1 et EN50082-1

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 responsabilitià che il prodotto:

TP112

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

EN55011, EN55022, EN50081-1 e EN50082-1

conformente 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

TP112

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

EN55011, EN55022, EN50081-1 y EN50082-1

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 kompatibilet (EMC).

Identification af apparat:
Kategori: Måleinstrument
Model/type: TP112

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

EN55011, EN55022, EN50081-1 og EN50082-1

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 tuotteemme

TP112

jota tämä selvitys koskee, on huväksytty

EN55011, EN55022, EN50081-1 ja EN50082-1

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

Safety

Before you start working with the TP112, first read these safety rules.

- Avoid working **alone**.
- Check the probes/testleads for damages. **DO NOT** use them if they are damaged.
- Take care when measuring voltages higher than 25 V AC or 60 V DC.
- Always choose the right function and range when measuring.
- The TP112 is grounded through the grounding conductor of the power cord of the PC it is placed in. Plug the power cord in a proper, grounded outlet before making connections to the inputs and outputs of the TP112. Proper grounding is essential for safe measuring.
- If the PC with the TP112 is not grounded, all accessible conductive parts can render an electrical shock.
- To avoid explosion, do not use the TP112 in an explosive atmosphere, the TP112 uses relays for initial calibration.

Chapter 1
Introduction

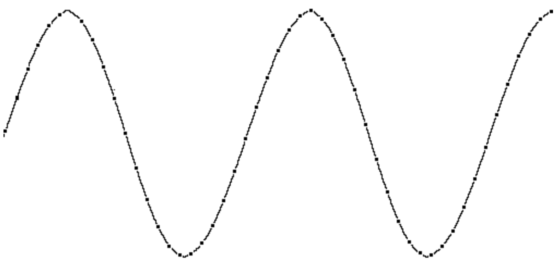
NOTE before using the TP112, first read the chapter about Safety.

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.

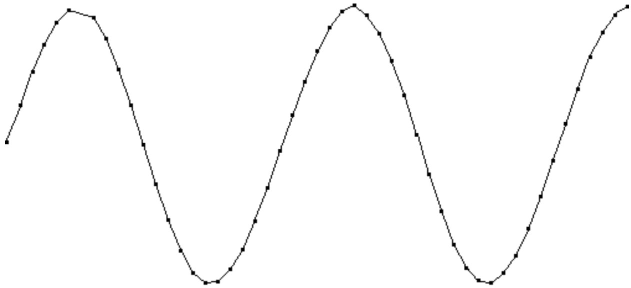
The TP112 is a 2 channel, 12 bits, 1 Msamples/sec interface card, with 8 digital inputs and 8 digital outputs. With the acompanying software, the TP112 can be used as a digital storage oscilloscope, a spectrum analyzer, a voltmeter or a transient recorder. All instruments measure by sampling the input signals, digitalize 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.



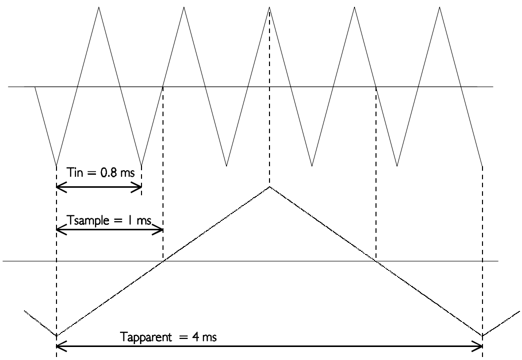
In the latter illustration a sine wave signal is sampled with 50 samples. By connecting the adjacent samples, the original signal can be reconstructed. See also the next illustration.



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.

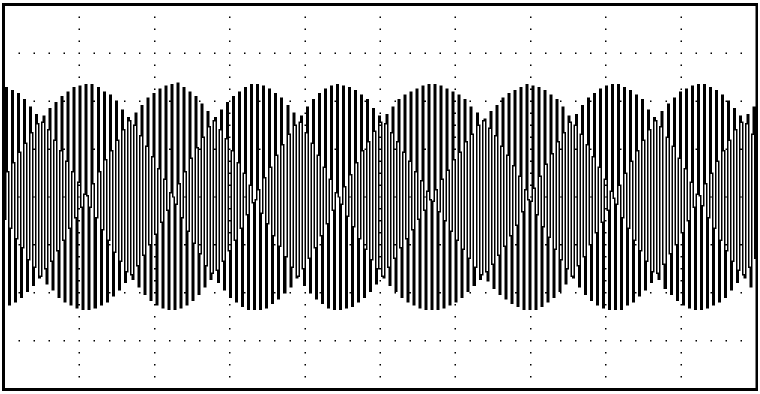
Aliasing

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 frquency of 1 kHz. The dotted signal is the result of the reconstruction. From that triangular signal the periodical time is 4 ms, which corresponds with an apparent frequency (alias) of 250 Hz (1.25 kHz - 1 kHz).
To avoid aliasing, the sample frequency must be higher than 2 times the maximum frequency of the input signal.

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



This time it is a sine wave signal with a frequency of 257 kHz, which is sampled at a frequency of 50 kHz. The minimal sampling frequency should have been 514 kHz. For proper analysis, the sampling frequency should have been 5 Mhz.

Digitising

After taking a sample of the input signal, it is digitised. This is done with an Analog to Digital Convertor, 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:

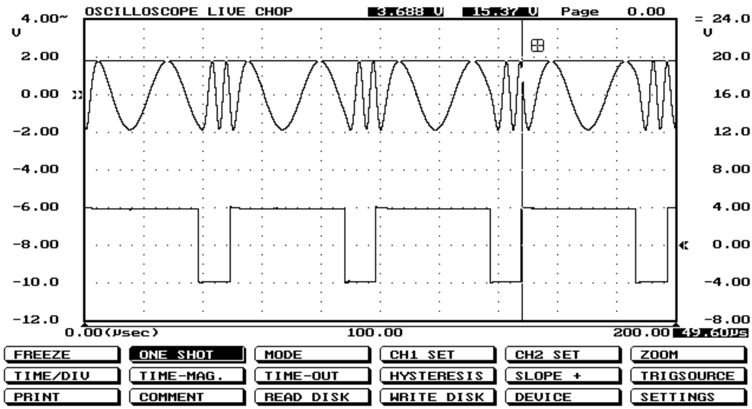
$$\text{number of quantifying steps} = 2^{\text{number of bits}}$$

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 quatifying 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 TP112 oscilloscope has two separated inputs (channels), for which the sensitivity of each channel can be set.

The TP112 is equipped with two types of oscilloscopes, LIVE and HOLD. The oscilloscope LIVE measures 1000 samples, 2 pages. The oscilloscope HOLD measures 15000 samples, 30 pages.

The sensitivity range for the channels goes from 125 V full scale to 20 V full scale. The timebase goes from 40 µsec/div to 5 sec/div.

The display of the oscilloscope is built of 500 x 256 (wxh) pixels, which corresponds with 10x8 (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.

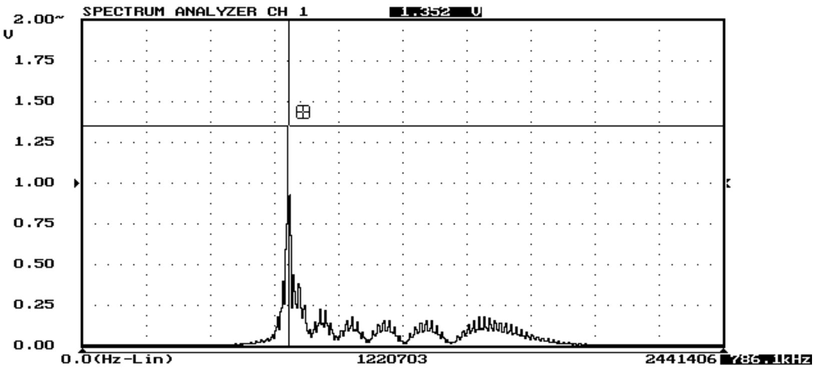
Not all electrical systems can be characterised in the time domain. Circuits like filters, amplifiers, oscillators, mixers, modulators and detectors can be characterised best by their frequency behaviour. That frequency behaviour is best obtained by observing the electrical signals in the frequency domain. 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).



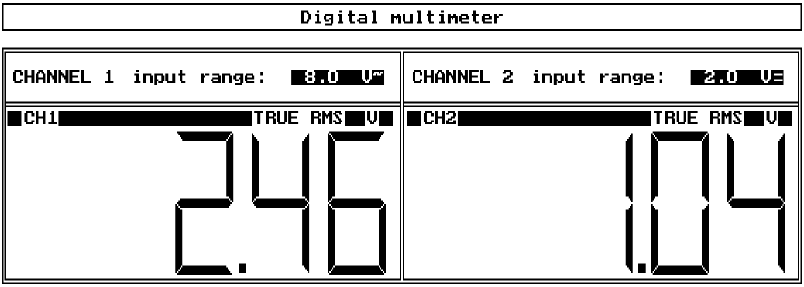
The TP112 is equipped with a two channel spectrum analyzer. The two channels can be set independently. The spectrum analyzer takes 1024 samples, of which a spectrum of 512 spectral components is calculated by means of FFT.

The sensitivity of the two channels can be set from 1.25 V full scale to 20 V full scale. The frequency range of the spectrum analyzer goes from 0.0125 Hz to 500 kHz.

The display of the spectrum analyzer is built of 500 x 256 (wxh) pixels, which corresponds with 10x8 (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 5 digit 7 segment display. The two independent channels each have their own displays.



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 125 mV full scale and 20 V full scale.

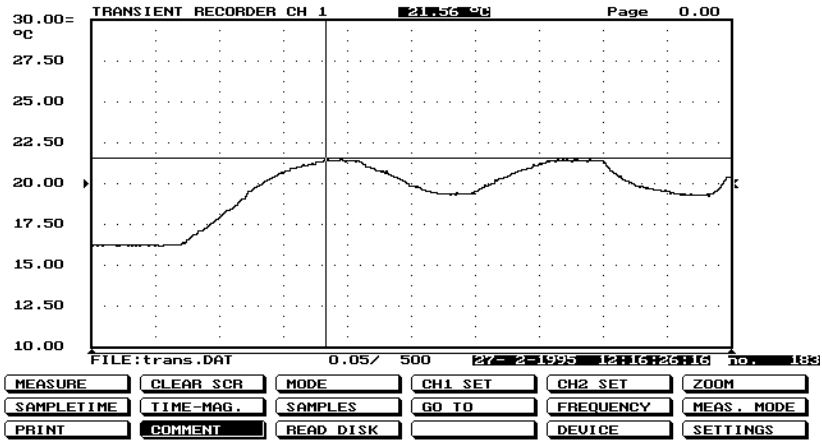
Besides this, the voltmeter has the possibility to measure at certain times and send the data to a printer or write it on disk.

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 sec x 30000 samples = 9000000 seconds (104.16 days).

The sensitivity of the channels is adjustable between 125 mV full scale and 20 V full scale.



The display of the transient recorder is built of 500 x 256 (wxh) pixels, which corresponds with 10x8 (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.

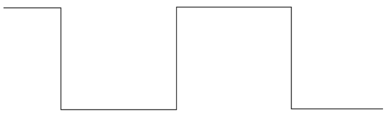
The transient recorder measures in one of the following ways:

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

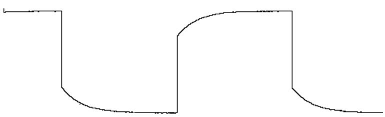
The TP112 is shipped with two probes. These are 1x/10x selectable passive probes. This means that the input signal is passed through directly or 10 times attenuated.

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

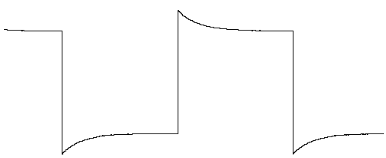
Therefore the probe is equiped with a setscrew, with which the parallel capacity of the attenuation network can be altered. To adjust the probe, switch the probe to the x10 and attach the probe to a 1 kHz square wave signal. Then adjust the probe for a square front corner on the square wave displayed.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



correct



undercompensated



overcompensated

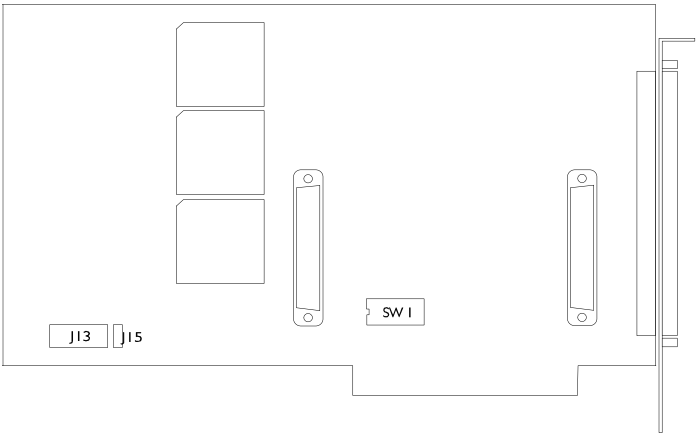
Safety

Before you start working with the TP112, first read these safety rules.

- Avoid working **alone**.
- Check the probes/testleads for damages. **DO NOT** use them if they are damaged.
- Take care when measuring voltages higher than 25 V AC or 60 V DC.
- The maximum input signal size is 200 V AC. Applying higher voltages may damage your TP112.
- Always choose the right function and range when measuring.
- The TP112 is grounded through the grounding conductor of the power cord of the PC it is placed in. Plug the power cord in a proper, grounded outlet before making connections to the inputs and outputs of the TP112. Proper grounding is essential for safe measuring.
- If the PC with the TP112 is not grounded, all accessible conductive parts can render an electrical shock.

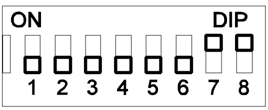
Chapter 3
Hardware installation

The TP112 is an 8 bits interface card which can be placed in any free 8 or 16 bits ISA slot of an IBM compatible PC, XT or AT. The card does not use DMA. With the IRQ2, IRQ3, IRQ4, IRQ5, IRQ6 and IRQ7 jumpers an interrupt can be selected. The TP112 uses 8 I/O addresses of the PC.



The only thing that has to be set is the base I/O address of the card. It can be set to any address between $000 and $3F8, in steps of 8 addresses. For that the address lines A3 .. A9 of the PC are necessary. The address lines A0 .. A2 are used for selecting the next 8 addresses of the card. Setting the base I/O address is done with dipswitch SW1.

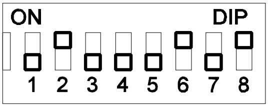
The factory setting is $300, which corresponds with a dipswitch setting as in the next illustration.



Calculation of address $300:

```
A0  A1  A2  A3 | A4  A5  A6  A7 | A8  A9
x   x   x   0  | 0   0   0   0  | 1   1
      $0       |       $0       |   $3
```

If the address $300 is already in use in your PC, you can set the TP112 to another address. Here follows an example for address $288



Calculation of the address $288:

```
A0  A1  A2  A3 | A4  A5  A6  A7 | A8  A9
x   x   x   1  | 0   0   0   1  | 0   1
      $8       |       $8       |   $2
```

Note If the base I/O address is changed, it also has to be changed in the software. See Setting the TP112 address.

Chapter 4

Software

Installation of the software

The software of the the TP112 is shipped on one diskette. The software runs in **protected mode**.

Note The protected mode software only works on a PC AT compatible computer with a **80286** or (upward) compatible processor and **2 MByte RAM** available.

Hint If your computer does not meet these specifications, a real mode version of the software is available. The real mode software works on all PC, PC XT and PC AT compatible computers with at least 640 KByte RAM. To get the real mode version of the software, please contact TiePie engineering

The protected mode software supports 15 measurement pages in the HOLD device, the real mode software supports 5 measurement pages.

In this manual the protected mode version of the software is described.

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 in a safe place and use the copy.

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

Hint A hard disk is much faster than a diskette and has usually more free space available for storing 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:

TP112.EXE	the program
TP112ENG.HLP	the English help file
TP5ENG.INF	information for the help file
TP112.FNT	fontfile for printing output
TP112.DEF	file with program settings
DMPI16BI.OVL	protected mode overlay
RTM.EXE	protected mode Run Time Manager

An example, where the diskette is in drive A: and the software has to be installed in a directory called TP112 on the C: drive.

```
C:
MD TP112
CD TP112
COPY A:*.*
```

The software is now installed on the harddisk. After starting the program, the DEFAULT DIRECTORIES can be set. This indicates where the program has to store and look for help files, instrument setting files and measurement data files. You can set the DEFAULT DIRECTORIES with the menu SETTINGS, in the lower right corner of the screen.

Structure of the program

The program is divided into 5 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 instrument can be active at any time.

From each instrument any other instrument can be made active. Each instrument can store measurement data on disk and read from disk. It is also possible to print out the data.

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

```
TP112 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 denotes the device that will be active after starting the program. Valid values are:

- LIVE the oscilloscope
- HOLD the storage oscilloscope
- SPECTRUM the spectrum analyzer
- VOLT the true RMS voltmeter
- TRANS the transient recorder

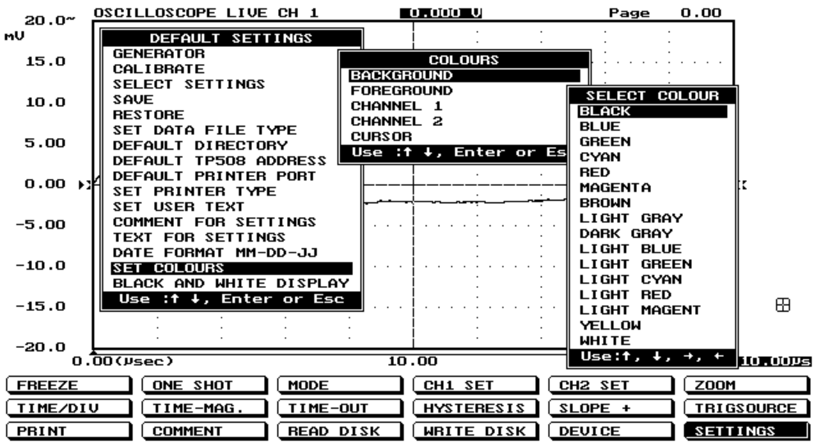
par2 denotes the name of the instrument settings file that has to be loaded. (see for saving and restoring instrument settings later in this chapter)

Example:

```
TP112 trans A:setting3.set
```

The program is now started as a transient recorder with the settings from the file A:setting3.set.

An invalid device name or file name is ignored. The parameters par1 and par2 may be exchanged, but have to be separated by a space.



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

Especially for controlling the program by means of a mouse, the screens of the instruments are provided with so called 'hot spots'. These are areas at which clicking the left mouse button causes an action to take place. A popup menu corresponding to the item displayed at that position of the screen appears. At the back of this manual two pages are included, illustrating the positions of the hot spots of the different instruments. The instruments LIVE, HOLD, SPECTRUM, TRANS use the same positions for the hot spots, for the device VOLT the hot spots are positioned at different places.

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 advance menu is built of three rows of 6 columns, the simple menu is built of two rows of 4 columns. As an example the advance main menu of the device LIVE, the oscilloscope, is given here.

LIVE		MODE	CH1 SET	CH2 SET	ZOOM
TIME/DIV	TIME-MAG.	TIME-OUT	HYSTERESIS	SLOPE +	TRIGSOURCE
PRINT	COMMENT	READ DISK	WRITE DISK	DEVICE	SETTINGS

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 next 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
RETURN	activates the selected choice
esc	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
home	Move the cursor to the top of the menu
end	Move the cursor to the bottom of the menu
SHIFT →	Move the popup menu to the right
SHIFT ←	Move the popup menu to the left
SHIFT ↑	Move the popup menu up
SHIFT ↓	Move the popup menu down
RETURN	Accept the selected choice
esc	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 <→←↑↓> keys in combination with the <Shift> key, the new position of the menu will be saved in the file TP112.DEF.

Help screens

The program contains integrated help screens. The screens can be invoked at several places in the program, by pressing function key <F1>. 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 configure the program, activate the choice SETTINGS from the main menu. A popup menu will appear.

```
DEFAULT SETTINGS
SIMPLE MENU
SELECT SETTINGS
SAVE
RESTORE
SET DATA FILE TYPE
DEFAULT DIRECTORY
DEFAULT TP112 ADDRESS
DEFAULT PRINTER PORT
SET PRINTER TYPE
SET USER TEXT
COMMENT FOR SETTINGS
TEXT FOR SETTINGS
DATE FORMAT MM-DD-YY
SET COLOURS
COLOUR DISPLAY
Use :↑ ↓, Enter or Esc
```

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 colours 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 TP112.DEF, which is read at the start of the program.

Setting the main menu type

With the choice SIMPLE MENU it is possible to select the simple main menu. This menu contains the following choices:

MEASUREMENT TYPE	With this choice saved instrument settings can be read.
START/STOP MEASUREMENT	With this choice the instrument can be stopped and started again.
REFERENCE ON / OFF	With this choice a reference signal can be put on or removed from the display. The reference signal is read from disk using READ MEASUREMENT.
WIPE SCREEN	With this choice the contents of the screen are wiped out.
SAVE MEASUREMENT	With this choice the current measured data can be stored on disk.
PRINT MEASUREMENT	With this choice the current measurement data can be printed out.

MISCELLANEOUS

With this choice three new choices are available:
ADVANCE MENU to select the advance menu with 18 choices
SAVE SETTINGS to save the current instrument setting to disk
QUIT PROGRAM to end the program.

Setting the file type for the measurement data

It is possible to store measurement data on disk. There are two ways available to store the data: BYTE and ASCII.

When using BYTE, the data is stored in binary. The file is built of 4 byte long records. The first two bytes are for channel 1, the last two are for channel 2.

When using ASCII, the measurement data is stored in ASCII. The file exists of four columns, separated by a comma. Each row contains one measurement for channel 1 and channel 2. The first column contains the sample number, the second column contains the sample time, the third column contains the value of channel 1 and the fourth column contains the value of channel 2.

Files in ASCII format are much larger than files in BYTE format, up to 7 times. An advantage of ASCII files is the fact that the files can easily be read in other programs, like spreadsheet programs and word processors.

Hint If you want to process data in a spreadsheet program, set the data file type to ASCII

The data file type can be set by activating the choice SET DATA FILE TYPE from the SETTINGS menu. A popup menu will appear with the two choices: BYTE and ASCII.

Setting the location of the files

By activating the choice DEFAULT DIRECTORY from the settings menu it is possible to set the directories for the files the program works with. By means of a popup menu with 6 choices, the locations for the program files and settings files, the print files and the data files for each instrument can be found or will be written. Each choice will place a dialog box in which the directory for the specific files has to be entered.

Setting the TP112 I/O address

With the choice DEFAULT TP112 ADDRESS from the settings menu, the I/O base address of the TP112 can be set for the program. After activating the choice, a dialog box appears, in which the address can be entered. The address can be entered decimal as well as hexadecimal. When entering hexadecimal, the address must be preceded by a '$' sign. When entering decimal, $300 must be entered as 768.

Setting the printer port

With the choice DEFAULT PRINTER PORT the port to which the printer is connected can be set. By means of a popup menu the choice can be made from PRN, LPT1, LPT2 and LPT3.

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

Setting the printer type

The software supports two types of printers: Epson FX compatible printers and HP laserjet/deskjet compatible printers. The printer type can be set by activating the choice SET PRINTER TYPE from the settings menu. A popup menu will appear with the choices EPSON and HP LASER.

Default text on the printer output

All printer output can be provided with default text. Three lines of text can be placed on each printout. They appear on the upper left corner of the printout. These lines can contain e.g. name and address of the company. The text can be entered by activating the choice SET USER TEXT from the SETTINGS menu. A dialog box appears in which the three lines of text can be entered. The text is saved in the file TP112.DEF, which is read at the start of the program.

Setting the date format

The program uses date indications at several places, like at printer output or on the screen of the transient recorder. There are two ways to represent the date: day-month-year (DD-MM-YY) and month-day-year (MM-DD-YY). The preferable notation can be selected by activating DATE FORMAT from the SETTINGS MENU. Behind DATE FORMAT the current setting is indicated.

Setting the screen colours

If your PC is equipped with a EGA or VGA graphics card, it is possible to adjust the screen colours for the program. After activating the choice SET COLOURS from the SETTINGS menu, a popup menu ap-pears, with the 'objects' for which the colour can be changed:

- the background of the screen
- the foreground of the screen (grid, text, menus)
- the signal of channel 1
- the signal of channel 2
- the crosshair

After activating an 'object' another popup menu appears with a list of 16 colours. In this popup menu the following keys are valid:

↑	go one colour up in the list
↓	go one colour down in the list
←	choose a darker shade of the colour
→	choose a lighter shade of the colour
RETURN	accept this colour
esc	leave the menu without changes

The screen colours are adapted immediately. Selecting a colour can also be done by mouse. Pushing the mouse forward and backwards will pick another colour, moving the mouse left and right will pick another shade. The colour is accepted by pressing the left mouse button.

If you do have an EGA or VGA graphics card, but no colour monitor, the colours are displayed in shades of grey.This can be confusing, since some colours produce almost the same shades of grey. To avoid this problem, you can tell the program that a black and white monitor is used, instead of a colour monitor. This can be done by activating the choice COLOUR / BLACK AND WHITE from the SETTINGS menu.

Hint When you have a monochrome monitor or an LCD, select BLACK AND WHITE MONITOR

Saving instrument settings

If you often use the same settings for an instrument, you can save the setup on disk. By loading the file with the setup, you can set your instrument very easy. The setups for all five instruments are saved.

To save an instrument setup, first you have to set the instrument completey. After the instrument is set, you can save the setup.

With TEXT FOR SETTINGS a dialog box appears, in which three lines of explanatory text for the setup to be saved can be entered. You can enter e.g. for what type of measurements this setup is to be used.

After the text is entered, the dialog box is closed with <Esc>

The popup menu SETTINGS will become active again. Choose from the menu the choice COMMENT FOR SETTINGS. A dialog box appe-ars.

In this dialog box a name for the setup can be entered. The maximum length for the name is 25 characters. This name is used for selecting the correct saved setup, when loading a setup. After the name is entered, the box can be closed by pressing <Esc>.

Then activate the choice SAVE from the SETTINGS menu. A dialog box appears, in which the filename for the settings file can be entered. The filename will automatically get an extension .SET. If you have en-tered the filename, the file with the instrument setup is saved on disk.

To make an instrument setting which will be used as a 'quit program' from SELECT SETTINGS (menu SETTINGS from the advance main menu) or MEASUREMENT TYPE (simple menu) any instrument setup will do, but the filename for the settings has to be QUIT. If a file named QUIT is read by one of the two here mentioned menus, the program will quit. Reading that file using RESTORE will not quit the program.

Loading instrument settings

Saved instrument settings can be read in three ways: at the start of the program as a parameter, through the choice RESTORE from the SET-TINGS menu and through the choice SELECT SETTINGS from the SETTINGS menu.

After activating the choice RESTORE a popup menu with a list of the present settings files appears. By means of the <→←↑↓> keys the file to be loaded can be selected. By pressing <Enter> the selection is accepted. Before loading an extra confirmation is requested. The selection can be cancelled by pressing <Esc>.

After activating the choice SELECT SETTINGS both a popup menu and a text window appear. In the popup menu the names of the settings, entered with the choice COMMENT FOR SETTINGS, are listed. In the text window, the accompanying text, entered with TEXT FOR SETTINGS, is shown.

Storing measurement data on disk

With the choice WRITE DISK from the main menu, the latest measurement data can be stored on disk. The data can be read later for examination or comparison with new measurements. The data can also be read into another program, e.g. a spreadsheet program, when the data is stored in ASCII.

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.

After activating the choice, a dialog box appears, in which the name for the data file has to be entered. No extension is needed, the program provides for a standard extension. It is possible to enter a full path, if not, the default directory is used.

After confirmation of the name, two files are written, one with measurement data, in a file with the extension .DAT and one with the ac-companying instrument information, in a file with the extension .GEG.

Loading saved measurement data

Previously saved measurement data can be read by activating the choice READ DISK from the main menu. After activation, a popup menu with a list of files with stored data appears. With the <→←↑↓> keys the correct file can be selected. Pressing <Enter> will ask for an extra confirmation and then load the data.

Both the measurement data and the instrument information are read. If the files are not written by the active instrument, only the data file is read or an error warning is displayed.

During selection of the correct file, the following keys are available:

- **F2** View the file information (size, date and time)
- **F3** View the file (only text files)
- **F4** Delete the file
- **F5** Rename the file
- **F6** Copy the file
- **F7** 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.

Note After reading stored measurement data, the oscilloscope will go to FREEZE. (See chapter Oscilloscope)

Note The voltmeter stores data in a different way. It can not read data.

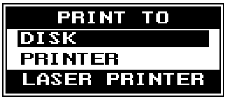
Printing measurement data

There are two ways to print measurement data.

The first way is by pressing <Shift-PrtSc>. A hardcopy of the screen is made, without extra information. After pressing, an extra confirmation is requested. On the printout, the user text entered in the SETTINGS menu, the comment text entered with the choice COMMENT from the main menu and the signal display are found.

The second method is by activating the choice PRINT from the main menu. The full page printout contains more information than the printout made with <Shift-PrtSc>. It is provided with the current time and date of the printout and the time and date of the measurement.

Activating the choice gives the following popup menu:



Choosing DISK will print to a file on disk. After activating, a name for the print file is asked for. The print file can be sent to a printer with the DOS command `COPY /B` *`filename`* `PRN`, where *`filename`* stands for the name you entered.

With the choice PRINTER, the data is sent directly to the printer.

The bottom line of the popup menu displays the installed printer.

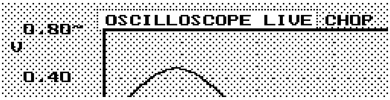
Note The voltmeter has a different way to print data. See the chapter on Voltmeter for more information.

Comment text can be added to the printout by activating the choice COMMENT from the main menu. Three lines of text can be entered in the dialog box that appears. The three lines will appear at the bottom of the printout. The comment text is also saved with WRITE TO DISK, so each measurement has its own comment text.

Note If settings are changed after saving the data on disk, these changes are not saved, except for the comment.

Choosing an instrument

By activating the choice DEVICE from the main menu, a popup menu with the available measuring instruments appears. With this the requested instrument is selected. The active instrument is displayed in the upper left corner of the screen.



The popup menu is also called by placing the mouse cursor on the device indication and pressing the left mouse button.

Ending the program

From anywhere in the program, if no popup menu is active, by pressing <Esc> a popup menu is displayed on the screen. This popup menu contains a list of measuring instruments and a choice QUIT, this choice terminates the program.

The same list is available through the main menu choice DEVICE.

Chapter 5

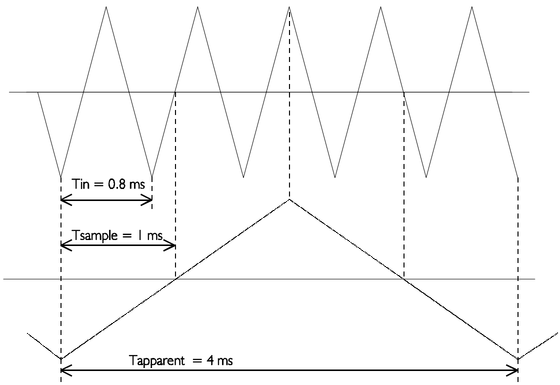
Oscilloscope (device: LIVE)

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

The TP112 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 500 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 a too low and therefore 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 frquency 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 frequen-

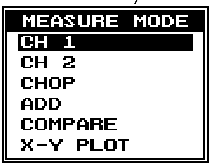
cy (alias) of 250 Hz (1.25 kHz - 1 kHz).

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

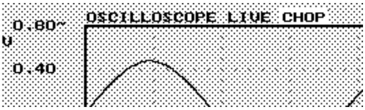
The TP112 has several modes to display the two channels. The mode can be set by activating the choice MODE from the main menu.



A popup menu MEASURE MODE will appear, with which the preferred mode can be selected. The channels can be displayed in the following ways:

CH1	Only channel 1 is measured and displayed
CH2	Only channel 2 is measured and displayed
CHOP	Channel 1 and 2 are measured and displayed simultaneously.
ADD	Channel 1 and 2 are measured simultaneously and added. Only the sum is displayed.
COMPARE	Channel 1 is measured and displayed. The latest measured signal at channel 2 is also displayed, to be able to compare it with channel 1.
XY-PLOT	This is a special display mode. Channel 1 and 2 are measured in CHOP mode. Channel 1 is displayed on the X axis of the screen, channel 2 on the Y axis of the screen.

The selected measure mode is displayed in the upper left corner of the screen, right of the instrument indication.

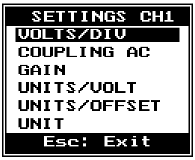


The measure mode menu can also be called by placing the mouse cursor on top of the measure mode indication and pressing the left mouse button.

Setting up a channel

The two channels of the TP112 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, channel 1 is explained here. Where channel 1 is described, the same applies to channel 2.

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



With the choices from this menu channel 1 can be configured.

Setting the sensitivity of a channel

With the choice VOLTS/DIV the sensitivity of a channel can be set. The input range of a channel is adapted with this setting. Selection of the input range is done with a popup menu with 13 choices. If e.g. the choice 0.5 is made, the sensitivity will be set to 0.5 Volts/division, giving a full scale range of ±2 volts. If AUTO is chosen, the software sets the sensitivity according to the size of the input signal.

The popup menu can also be called by placing the mouse cursor next to the vertical axis, between the numbers and then pressing the left mouse button. When two channels are displayed, the left axis is for channel 1 and the right axis is for channel 2.

An other way of setting the sensitivity is done by means of the function keys <F5>, <F6>, <F7> and <F8>. With <F5> the sensitivity of channel 1 is increased, with <F6> decreased. With <F7> the sensitivity of channel 2 is increased, with <F8> decreased.

Note In AUTO ranging mode, it takes the oscilloscope 1 to 5 measurements to set the sensitivity properly, depending on the input signal.

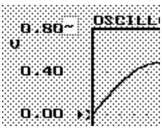
The available sensitivitiy ranges are: 1.25 V, 2.50 V, 5 V, 10 V and 20 V full scale.

Setting the signal coupling of a channel

By activating the choice **COUPLING AC/DC**, the text changes from COUPLING AC to COUPLING DC or vice versa. In the hardware, a capacitor is switched into (AC) or out of (DC) the signal path.

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

The current coupling is displayed in the upper part of the vertical axis. For DC the sign **=** is used and for AC the sign **~**.



The coupling can also be set by placing the mouse cursor on top of the coupling indicator and then press the left mouse button.

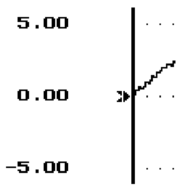
An other way to set the coupling is by means of the keys <a>, <d>, <A> and <D>. The key <a> sets the coupling of channel 1 to AC, the key <d> sets the coupling of channel 1 to DC. The key <A> sets the coupling of channel 2 to AC, the key <D> sets the coupling of channel 2 to DC.

Setting the vertical position of a channel

The vertical position of a channel on the screen is adjustable with a software method.

Using the software method, 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 software positioning method is not done by means of a popup menu. At the lefthand side of the screen (in CHOP, ADD and X-Y for channel 2 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 channel 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)
SHIFT	ins	Switching between crosshair and offset level controll.

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.

Note When the offset level of a channel is changed, the trigger level will change accordingly, providing the same triggering while positioning.

Enlarging or reducing the vertical axis

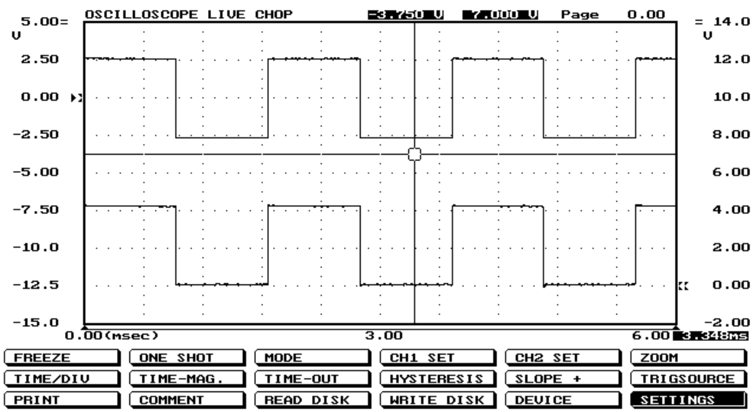
The measured values can be enlarged or reduced by software. Because of that it is possible to magnify the vertical axis. It is also possible to invert the measured values. After activation of the choice GAIN in the menu CH1 SET, a dialog box appears in which a multiplication factor can be entered. The 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 channel 1 from the signal on channel 2. To do that you have to set the GAIN for channel 1 to -1 and the gain for channel 2 to +1. By selecting a measuring mode ADD (channel 2 - channel 1) is displayed.

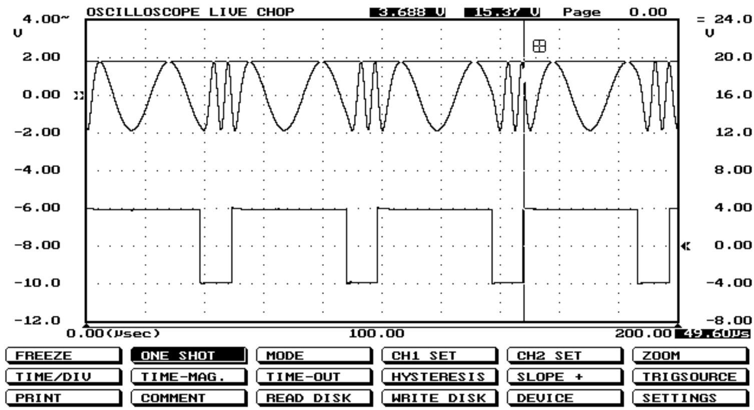
Note By choosing a gain of -1, two signals can be subtracted from each other.

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 channel 1 and a 4 volt square wave signal on channel 2. For displaying both signals at the same size, set the gain for channel 1 to 0.8 and for channel 2 to 1. Both signals are displayed then at the same size.

This is displayed in the next illustration. The upper signal is channel 1, the lower signal is channel 2. The left voltage axis is for channel 1, the right voltage axis is for channel 2.



A third application is displaying two signals in a kind of 'split screen mode'. This can be achieved by setting the GAIN of both channels to 0.5 and moving channel 1 2 divisions up by means of the software offset and channel 2 2 divisions down by means of the software channel offset. See also the following illustration.

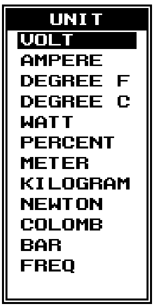


Changing the units of the vertical axis

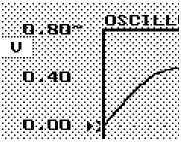
Along the vertical axis, normally the voltage of the input signal is projected. The unit of measure is then Volt and the numbers along the axis correspond with the size of the input signal.

The TP112 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 12 choices appears. The unit that is selected, is displayed at the top of the vertical axis. It is also placed on the paper when printed out. When activating the empty choice, a dialog box appears, in which a custom unit can be entered, with a maximum length of 5 characters.



The UNIT popup menu can also be called by placing the mouse cursor on top of the unit indication and pressing the left mouse button.

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 can be entered. The numbers along the vertical axis are multiplied by the factor and replaced by the result.

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

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

It is also possible to add an offset to the numbers of the vertical axis. In the dialog box called by activating UNITS/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 aplication 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, the zero level of the axis is changed in a way that the absolute temperature is directly readable.

Setting up the time axis

Along the horizontal axis of the screen the measuring time is projected. The oscilloscope always measures two screens (pages) of data. The default displayed page is the page after the trigger point, post trigger. The other page is the page before the trigger moment, pre trigger. Switching between both pages is done with <PgUp> and <PgDn>. <PgUp> gives the post trigger page and <PgDn> gives the pre trigger page. With <+> and <-> the stepsize is changed.

The data remains on the screen until the next pages are measured.

Setting the time base

By means of the choice TIME/DIV from the main menu the time base is set. Activating the choice gives a popup menu with 18 ranges. If the range 0.2 SEC is chosen, one time division corresponds with 0.2 second. The complete screen (1 page) contains 10 divisions, which corresponds with a time of 2 seconds.

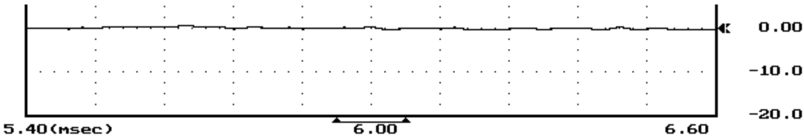
The popup menu for the time base setting is also called by placing the mouse cursor on top of one of the numbers along the vertical axis and pressing the left mouse button.

Each division contains 50 samples. A complete page contains 500 samples. With the choice 0.2 sec/div each 0.2/50 = 4 msec a sample is taken. The complete sampling process takes 2 pages x 10 divisions x 0.2 sec/div = 4 seconds. If, within these 4 seconds, a key is pressed, the sampling process stops and the action requested with the key is taken.

The following time base settings are available:
50 μs/div, 0.1 ms/div, 0.2 ms/div, 0.5 ms/div, 1 ms/div, 2 ms/div, 5 ms/div, 10 ms/div, 20 ms/div, 50 ms/div, 0.1 s/div, 0.2 s/div, 0.5 s/div, 1 s/div, 2 s/div and 5 s/div.

Magnification of the horizontal axis

By activating the **TIME MAG** choice from the main menu, a popup menu appears in which 6 time axis magnifications are available.
When the choice **10*** is activated, the time axis is magnified 10 times.

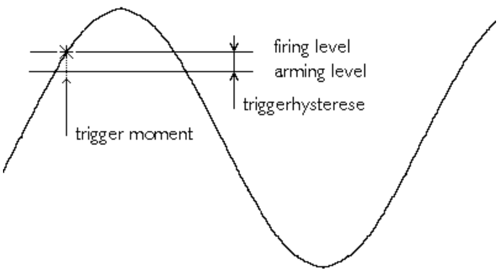


The magnified part of the screen is indicated by two interconnected arrows at the bottom of the screen. The magnified part of the screen can be moved using the <PgUp> and <PgDn> keys or the mouse. By placing the mouse cursor on the line connecting the two arrows, keeping the left mouse button pressed, the part can be moved. When using the keyboard, the step size can be changed using the <+> and <-> keys.

The magnification factors available are: 1, 2, 4, 5, 10 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 circuit. 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 circuit is armed. If the the input signal passes the firing level, the trigger circuit 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

The choice TRIGSOURCE from the main menu gives a choice of which channel has to be triggered. By activating the choice a popup menu is displayed with two sources of triggering: TRIG CH1 and TRIG CH2.

With TRIG CH1 and TRIG CH2, triggering is from channel 1 and channel 2, respectively.

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, ADD and X-Y for channel 2 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)
SHIFT	ins	Switching between crosshair and offset level controll.

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.

Note When the offset level of a channel is changed, the trigger level will change accordingly, providing the same triggering while positioning.

Setting the trigger hysteresis

By activating the choice HYSTERESIS from the main menu, a popup menu appears in which 12 choices for the trigger hysteresis are shown: 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.

If e.g. **0.25 DIV** is chosen, the trigger hysteresis is set to 0.25 division.

Setting the trigger slope

The trigger slope can be changed by activating the choice **SLOPE ±** from the main menu. When activating the choice, the trigger slope changes as well as the text in the menu. The text changes from SLOPE- to SLOPE + or vice versa. The text indicates the current trigger slope setting.

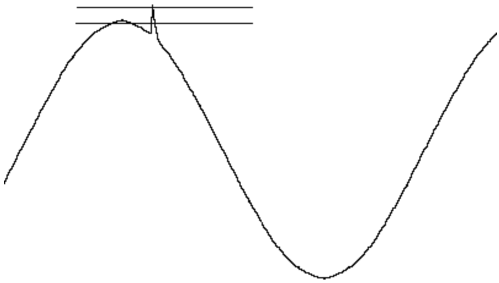
Setting the trigger time-out

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, 200 DIV, 500 DIV, 1000 DIV, 2000 DIV, 5000 DIV, 10000 DIV and INFINITE.
If e.g. **100 DIV** is activated and the time base setting is 0.1 ms/div, then there will be a delay of 100 * 0.1 ms = 10 ms. When **INFINITE** is activated, the delay time will be infinite.

Example: Suppose you have a signal, on which every now and then a glitch occurs. You want to examine that glitch, how to set the trigger?



You are interested in the glitch, so the slope has to be set to the rising slope. The trigger level has to be set such that the normal signal remains under the firing level, but the glitch exceeds the firing level. The trigger hysteresis has to be set such that the (signal + glitch) is larger than the trigger hysteresis, to provide for a trigger. The time-out has to be set to a long time, at least longer than the time it takes for the next expected glitch. If the time-out is set too short, the display will be replaced too soon by a new one, removing the last measured glitch from the screen.

Stopping the display

To avoid a new screen of measurement data replacing an old screen, measuring and displaying of data can be stopped. This is done by activating the choice LIVE from the main menu. When activated, the sampling process and displaying are stopped. The latest measured and displayed data will remain on the screen, for examination. It can e.g. be magnified. When activated, the text in the menu will change from LIVE to FREEZE, to indicate the display is 'frozen' and a menu choice is added to the main menu: ONE SHOT.

The sampling process can be started again by activating the same choice again, now called FREEZE. FREEZE will change to LIVE again and the ONE SHOT disappears again.

When measuring is stopped by switching from LIVE to FREEZE, measurements can still be made by activating ONE SHOT. Once two pages of measurement data are taken. Then the process stops again. During measuring the text ONE SHOT changes to MEASURE. When measuring is complete and the signal is displayed, ONE SHOT will return.

The sampling proces can be stopped during sampling (MEASURE) by pressing a key.

Magnifying a part of the screen

To examine a certain part of the screen thoroughly, it can be magnified. Activating the choice ZOOM will give a popup menu with two choices: FULL and WINDOW.

By activating WINDOW, an area of the screen can be selected, to be displayed full screen. The numbers along the axes will be adjusted accordingly. Selection of the area can by done by keyboard or by mouse. At first, one corner of the area has to be pointed out. With the mouse or with the <→←↑↓> keys the cursor is placed at the proper position. By pressing <Enter> or the left mouse button the first corner is selected. Then the second corner has to be pointed out and selected with <Enter> or the left mouse button. During selection of the second corner, a rectangle marks the area. Selecting the area can always be cancelled by pressing <Esc> while pointing to the corners.

Opmerking With the oscilloscope LIVE, zooming only works correct in FREEZE. In LIVE the signal is not displayed correct since the trigger level is not adapted according the vertical position.

The choice FULL displays the complete signal. The magnify factors for the horizontal and vertical axis are set to 1 and the software offset for the channels to 0.

Measuring with the crosshair

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:

SHIFT home	Setting the absolute zero
SHIFT del	Setting the relative zero
SHIFT end	Move the crosshair to zero
SHIFT ↑	Move the crosshair up
SHIFT ↓	Move the crosshair down
SHIFT ←	Move the crosshair to the left
SHIFT →	Move the crosshair to the right
SHIFT page up	Mirror the crosshair on the relative x-axis
SHIFT page down	Mirror the crosshair on the relative y-axis
SHIFT -	Decrease the step size twice
SHIFT +	Increase the step size twice
SHIFT ins	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.

Chapter 6

Storage oscilloscope (device: HOLD)

Differences with the oscilloscope

The TP112 storage oscilloscope 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 storage oscilloscope only works in ONE SHOT mode and does not measure 2 pages but 30 pages of data. With the <PgUp> and <PgDn> keys, the pages are selected. The page-number is indicated at the upper right corner of the screen. Pages with a negative number are pre-trigger pages.
- The storage oscilloscope does not provide for X-Y PLOT.
- The storage oscilloscope can store data on disk while measuring.

The main menu of the storage oscilloscope differs some from the main menu of the oscilloscope.

ONE SHOT	NO DISK	MODE	CH1 SET	CH2 SET	ZOOM
TIME/DIV	TIME-MAG.	TIME-OUT	HYSTERESIS	SLOPE +	TRIGSOURCE
PRINT	COMMENT	READ DISK	WRITE DISK	DEVICE	SETTINGS

At the position where the oscilloscope has the menu choice LI-VE/FREEZE, the storage oscilloscope has the choice ONE SHOT and at the position where the oscilloscope in FREEZE mode has the choice ONE SHOT, the storage oscilloscope has a choice NO DISK/AUTO DISK.

Automatic storage of measurement data

The storage oscilloscope can store measurement data automatically on disk. This is set by activating the choice NO DISK from the main menu. The text changes from NO DISK to AUTO DISK. As soon as a measu-rement is started using ONE SHOT, entering a filename for the data is requested.

```
      Syntax: drive:\path\filename (no .ext !)

  DATA_1

                     F1: Help  Enter: Accept  Esc: Exit
```

For the name, 5 characters are accepted, the rest is ignored. The software adds three characters for a file serial number. When the name is entered, measuring can start. There is a delay until the input signal meets the trigger conditions. When the trigger conditions have been met and a signal is measured, the data is written on disk. This is indicated by a text window, in the lower left corner of the screen.

```
AUTO DISK WRITING...,  Esc: Interrupt measuring.
FILE: C:\TIEPIE\TP508\DATA_1
```

When the data (measurement data, time and date of measuring) are written, the storage oscilloscope will wait for the next signal to meet the trigger conditions. If that occurs, new samples are taken and stored on disk in a new file. The file serial number will be increased by one from the previous number. The measuring and storing of data can be stopped by pressing <Esc>. READ DISK is used to load stored data for examination.

This system is very usefull when e.g an interference glitch has to be measured. Suppose at unpredictable times a glitch occurs. By setting the trigger delay to INFINITE and selecting AUTO DISK, the storage oscilloscope may wait days for the glitch, but when it occurs, it is measured and stored. The time and date of the occurrence are stored and appear on a printout.

Note When the sensitivity of a channel is set to AUTO ranging, the oscilloscope uses 1 to 5 measurements, depending on the input signal, to find a suitable sensitivity.

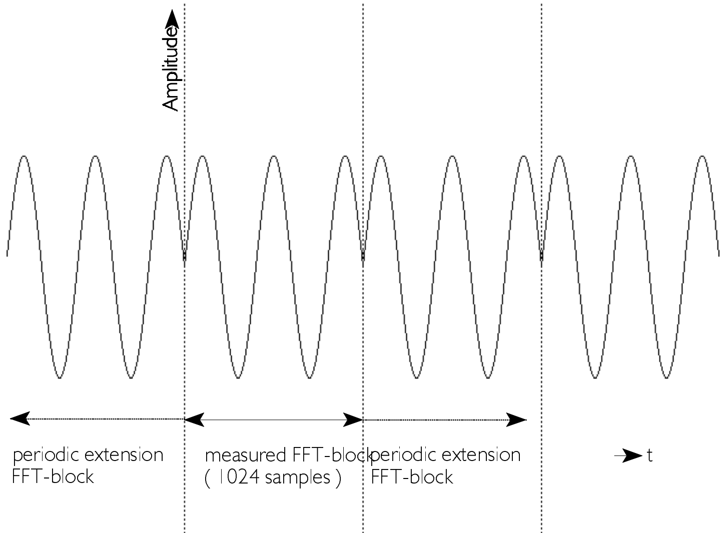
Chapter 7

Spectrum analyzer (device: SPECTRUM)

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.

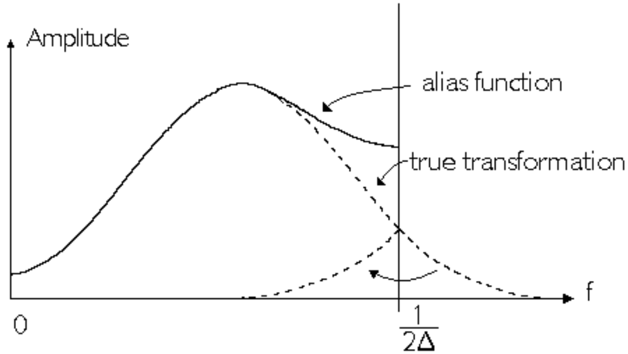


From the signal obtained like this, the spectrum is calculated. The FFT calculation results in 512 spectral components. These spectral components together form the frequency spectrum of the input signal. 500 of the 512 spectral components are displayed, in 10 divisions of 50 spectral components each.

The TP112 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

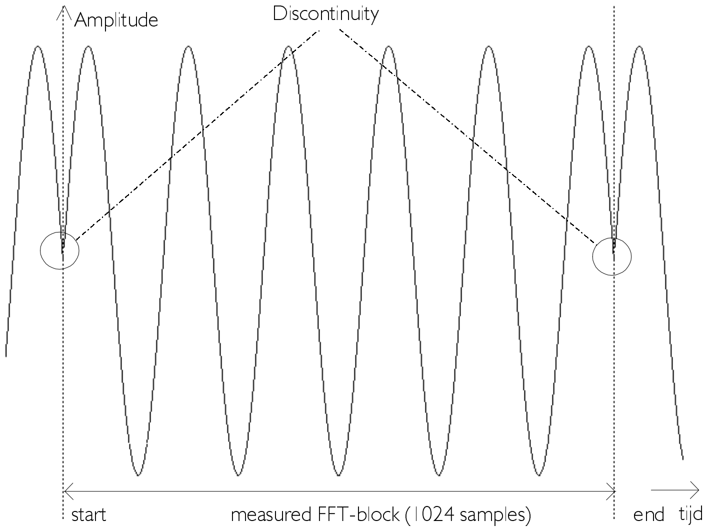
Like the oscilloscope, the spectrum analyzer needs a high enough sampling frequency to avoid aliasing. The result of aliasing when using FFT is that the frequency curve is folded back on itself. See also the next illustration.



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

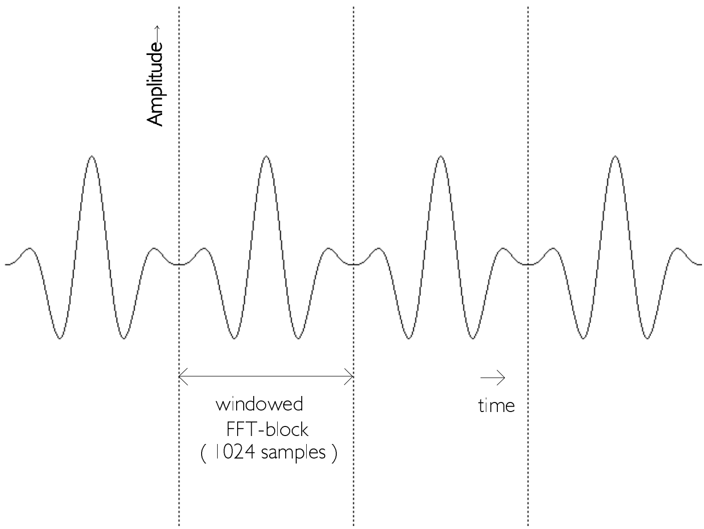
Windowing

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.

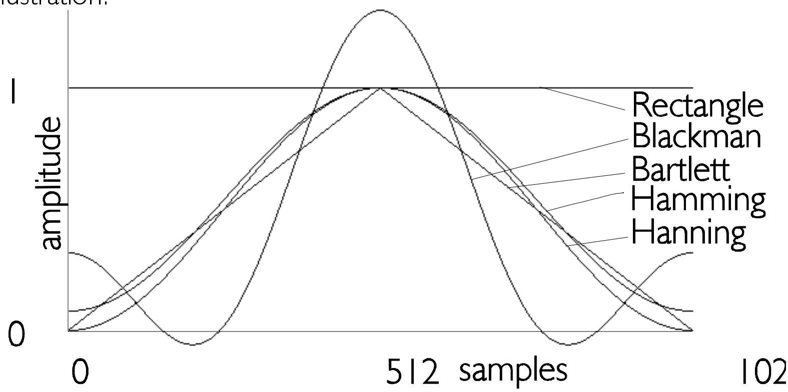
The smearing, caused by the calculation errors, can be decreased by placing a window on the FFT-block of 1024 samples, in such a way that the ends of the FFT-block are smoothly tapered to zero and discontinuities do not occur when the FFT treats the windowed block as one period of a periodic signal.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



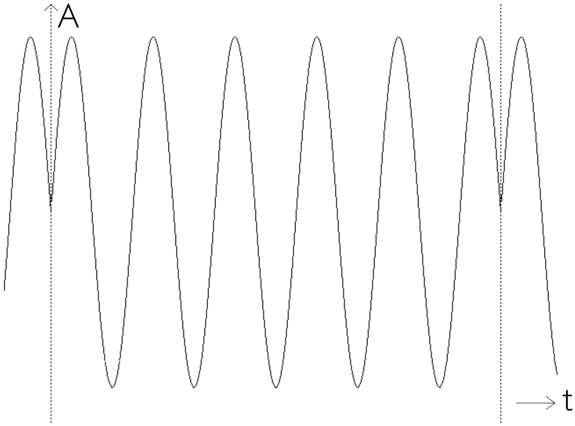
The software supports five different windows:

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

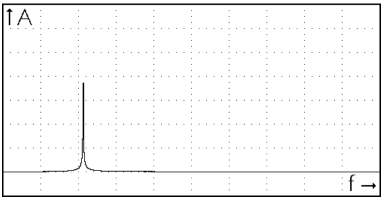
A graphical representation of the different windows is given in the next illustration.



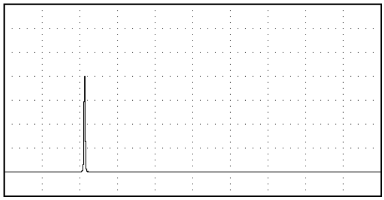
The effects of the different windows are displayed next, together with the sinewave signal the spectrum is taken from. The sinewave does not fit in the FFT-block and will therefore cause calculation errors.

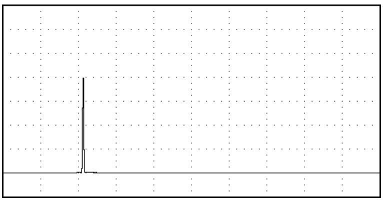


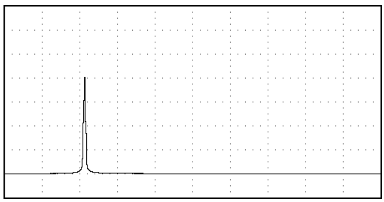
The measured sine wave

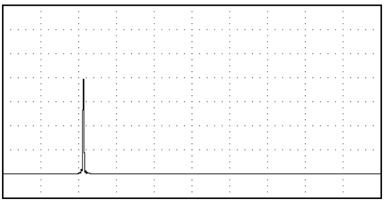


Rectangular window


Hanning window

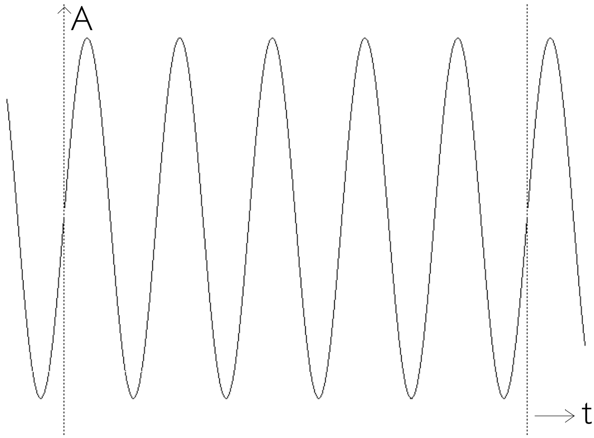

Hamming window


Blackman window

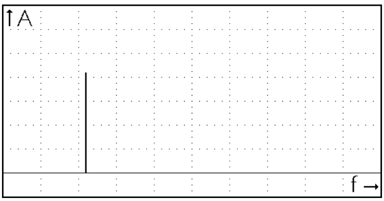

Bartlett window

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

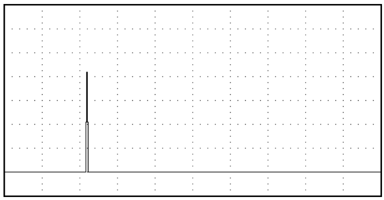
Again some example illustrations, now with a sine wave exactly fitting in the FFT-block.



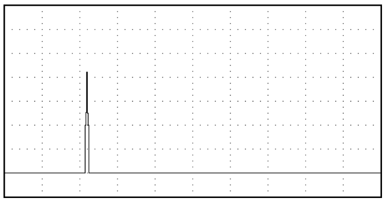
The measured sine wave



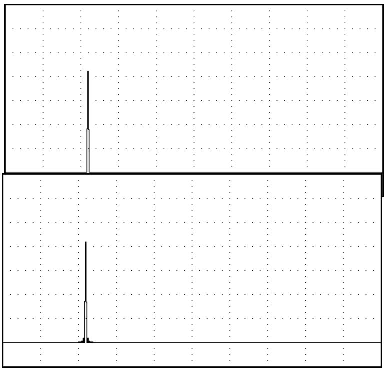
Rectangular window



Hanning window



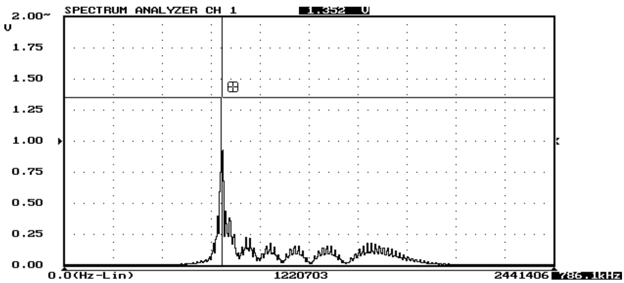
Blackman window



Bartlett window

The spectrum analyzer screen

The screen of the spectrum analyzer is divided in two parts. The upper part is the display. Here the spectrum of the input signal is displayed.



The lower part contains the main menu of the spectrum analyzer.

This has some similarities with the main menu of the oscilloscope and some differences. The lower row of choices is the same as the lower row in the oscilloscope menu and is not discussed in this chapter.

Displaying channels

The spectrum analyzer has several ways to display the two channels. By activating the choice 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 channles 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 or using the mouse or the function keys.

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 / 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 the setting change.

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

Setting the method of measuring

By activating the choice CH1 NORMAL MODE or CH2 NORMAL MODE from the menu FFT OPTIONS the methode of measuring for a channel can be set. Both channels behave alike, so only channel 1 will be discussed here. When the choice CH1 NORMAL MODE is activated, the text will change to CH1 MAX MODE and CH1 will measure and display only the maximum values of each spectral component. When CH1 NORMAL MODE is selected, each new spectrum will be displayed.

The text in the menu indicates the selected measuring method.

Setting up the frequency axis

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

Setting the frequency range

By activating the choice **FREQUENCY** from the main menu, the frequency range of the spectrum analyzer can be set. After activation, a popup menu containing 16 frequency ranges appears.

When e.g. **1250 Hz** is activated, the maximum frequecy the spectrum analyzer can display will be 1250 Hz. The frequency resolution will be 1250 Hz / 512 = 2.44 Hz. The available frequency ranges are:
6 Hz, 12 Hz, 24 Hz, 48 Hz, 120 Hz, 240 Hz, 480 Hz, 1200 Hz, 2400 Hz, 4800 Hz, 12 kHz, 24 kHz, 48 kHz, 117.2 kHz, 234.4 kHz and 468.8 kHz.

Note Since only 500 of the 512 points of the spectrum are displayed, the numbers along the frequency axis are not exactly the same as the numbers in the menu. 125 kHz e.g. will be 122070 Hz.

Note When changing the frequency range of the spectrum analyzer, the sampling frequency is changed.

Magnifying the frequency axis

By activating the choice **FREQ. MAG** from the main manu, a popup menu appears with which a choice can be made from 6 frequency axis magnifications. If the choice **10*** is activated, the frequency axis will be magnified 10 times.

The magnified part of the screen is indicated by two interconnected arrows at the bottom of the screen. The magnified part of the screen can be moved using the <PgUp> and <PgDn> keys or the mouse. By placing the mouse cursor on the line connecting the two arrows, keeping the left mouse button pressed, the displayed signal can be moved. When using the keyboard, the step size can be changed using the <+> and <-> keys.

The magnification factors available are: 1, 2, 4, 5, 10 and 20 times.

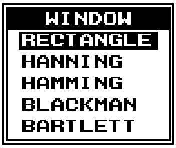
Adapting the frequency axis

By activating F-AXIS LIN / F-AXIS LOG from the main menu, the frequency axis can be set to linear or logarithmic.

The text in the menu indicates the selected setting.

Selecting the window

The spectrum analyzer has five different windows to reduce errors when calculating the spectrum. At the beginning of this chapter an explanation about the use of windows can be found. By activating the choice WINDOW TYPE from the menu FFT OPTION the following popup menu will appear.



Through this menu, the window to be used for the FFT calculation is set.

Averaging of spectra

When the spectrum of a non-periodic signal (e.g. a signal with a lot of noise) is measured, more spectra can be measured and averaged. The advantage is that the noise component disappears and the true signal remains.

Averaging can be set by activating the choice NUMBER OF AVERAGE from the menu FFT OPTION. A popup menu will appear, containing the choices 1, 2, 5,10, 20, 50, 100 and 200.

When AVERAGE 10 is activated, 10 spectra are measured and averaged. After calculating each spectrum the result is displayed. The averaging process can be cancelled by pressing <Esc>. Averaging is switched off by selecting the choice 1.

Starting a spectrum measurement

When all settings are made, the measurement can be started. There are two methods.

Measuring once

By activating ONE SHOT from the main menu, one measurement will be performed.

After activating, a measurement is made. This is indicated by a window:

```
MEASURING
                              (Esc: Exit)
```

Then the measured data will be windowed and the FFT will be calculated. This is indicated by:

```
CALCULATING SPECTRUM
                              (Esc: Exit)
```

When the FFT calculation is finished, the spectrum will be displayed.

When averaging is selected, after displaying the first spectrum measuring starts again. Then the second spectrum will be calculated, indicated by:

```
CALCULATING SPECTRUM 2
                              (Esc: Exit)
```

When all spectra are measured and calculated, measuring and calculating stops.

Continuous measuring

By activating FREEZE / LIVE from the main menu, spectra will be measured continuously. When a spectrum is measured, calculated and displayed, the process is started again.

The current setting is indicated by the text in the menu. By activating the choice, both the setting and the text change.

The continuous process is cancelled by pressing <Esc>.

Note Calculating a spectrum is time consuming. The time needed for calculating a spectrum is strongly dependent on the calculating power of the PC.

When the spectrum analyzer has stopped measuring, a crosshair appears. For controlling the crosshair, refer to the chapter Storage Oscilloscope: **Measuring with the cross hair**.

Measuring a harmonic distortion

With the spectrum analyzer it is possible to measure the harmonic distortion of a signal. When the choice **DISTORTION** from the main menu is activated, a crosshair and a text window containing the text `'Select fundamental frequency (use <- or -> and Enter)'` appear.

With the <→←> keys and the mouse, the crosshair can be placed at a frequency. For controlling the crosshair, refer to the chapter Storage Oscilloscope: **Measuring with the cross hair**. The selected frequency will be used as the fundamental frequency. The total harmonic distortion related to this frequency is calculated using the formula:

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

V_1 is the RMS amplitude of the fundamental frequency, V_2 .. V_{10} are the RMS amplitudes of each higher harmonic. The RMS amplitudes of the harmonics and the harmonic distortion of the fundamental frequency are calculated and displayed in a table. The harmonic distortion is calculated for both channels.

Calculate the spectrum from the HOLD signal

By activating the choice **HOLD -> FFT** from the main menu, a spectrum is calculated from 1024 samples of the last measured signal on the storage oscilloscope (device HOLD). The frequency- and voltage axis of the spectrum analyzer screen are adapted according to the axes of the oscilloscope.

After activating the choice, a popup menu appears, with which is determined wich part of the HOLD signal is used for the spectrum calculation.

The first possibility is **HOLD t=0**, with which a spectrum is calculated from 1024 samples from time t=0.

The second possibility is **HOLD crosshair**, with which a spectrum is calculated from 1024 samples from the point the crosshair is located. If the crosshair is located less than 1024 samples from the end of the HOLD signal, the spectrum is calculated from the last 1024 samples of the HOLD signal.

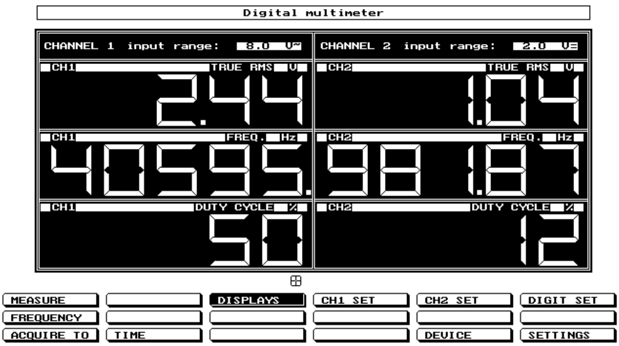
Chapter 8
Voltmeter (device: VOLT)

The TP112 is equipped with a 2 channel voltmeter with digital display. The voltmeter functions as follows:

- 200 points are sampled from the input signal.
- The measured data is processed, e.g. for calculating the RMS value or the mean value. 10 different operations are available.
- The calculated values are displayed, e.g. add CH1 and CH2 and display on channel 1. 16 different display methods are available.

Because of the different computations and display methods, about 100 different ways to measure and display signals are available.

The voltmeter screen



Like the other instruments, the screen of the voltmeter is divided in two parts: the diplay and the main menu.

The display has two to six large readouts, one to three for each channel. The settings of the channels are displayed as well. The voltmeter main menu differs a lot from the other instruments.

Setting up a channel

As noticed before, the channels can be configured in many ways. Both channels have equal settings, so only channel 1 is discussed here. Activating the choice CH1 SET brings up a popup menu in which the settings for channel 1 are available.

Setting the input range

By activating INPUT RANGE, a popup menu appears, containing 6 choices, 5 input ranges and an AUTO range function.

When e.g. the choice 2.5 V is activated, the range of the voltmeter is 2.5 volt full scale. In AUTO ranging, the voltmeter determines the best setting depending on the input signal.

The popup menu can also be called by placing the mouse cursor on top of the input range indicator and pressing the left mouse button.

Another way of setting the sensitivity is done by means of the function keys <F5>, <F6>, <F7> and <F8>. With <F5> the sensitivity of channel 1 is increased, with <F6> decreased. With <F7> the sensitivity of channel 2 is increased, with <F8> decreased.

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

Note When auto ranging, the voltmeter requires 1 to 5 measurements, depending on the input signal, to find the proper setting

Setting the signal coupling

By activating the choice COUPLING AC/DC, the text changes from COUPLING AC to COUPLING DC or vice versa. In the hardware, a capacitor is switched into (AC) or out of (DC) the signal path.

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

The coupling can also be set by placing the mouse cursor on top of the coupling indicator and then press the left mouse button.

Another way to set the coupling is by means of the keys <a>, <d>, <A> and <D>. The key <a> sets the coupling of channel 1 to AC, the key <d> sets the coupling of channel 1 to DC. The key <A> sets the coupling of channel 2 to AC, the key <D> sets the coupling of channel 2 to DC.

Displaying the measured value

For displaying the measured and calculated values, 16 methods are available. By activating the choice **DISPLAY** from the CH1 SET menu, a popup menu appears, showing the available methods. These are:

CH1: The measured value of CH1 is displayed

CH2: The measured value of CH2 is displayed

CH1*CH2: The values of CH1 and CH2 are multiplied and the result is displayed.

CH1/CH2: The value of CH1 is divided by the value of CH2. The result is displayed.

CH2/CH1: The value of CH2 is divided by the value of CH1. The result is displayed.

CH1-CH2: The value of CH2 is subtracted from the value of CH1. The result is displayed.

CH2-CH1: The value of CH1 is subtracted from the value of CH2. The result is displayed.

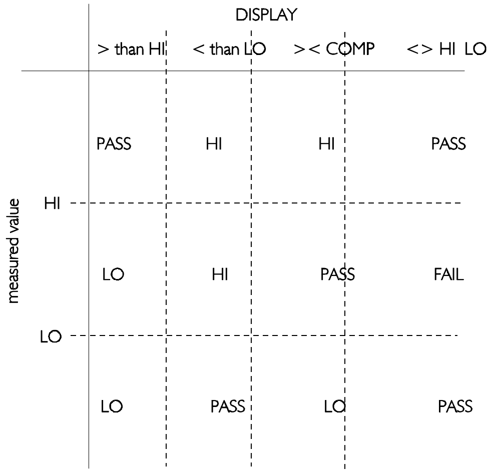
CH1+CH2: The values of CH1 and CH2 are added and the sum is displayed.

> then HI: The value is compared to the HIGH VALUE (see SET HIGH VALUE from menu SET CH1). If the value is higher than the HIGH VALUE, then PASS is displayed, else LO is displayed. See also the next illustration.

< then LO: The value is compared to the LOW VALUE (see SET LO VALUE from menu SET CH1). If the value is lower than the LOW VALUE, then PASS is displayed, else HI is displayed. See also the next illustration.

>< COMP: The value is compared to the HIGH VALUE and the LOW VALUE. If the value is higher than the HIGH VALUE, HI is displayed. If the value is lower than the LOW VALUE, LO is displayed. If the value is lower than or equal to the HIGH VALUE **and** higher than or equal to the LOW VALUE, PASS is displayed. See also the next illustration.

<> HI LO: The value is compared to the HIGH VALUE and the LOW VALUE. If the measured value is higher than the HIGH VALUE, PASS is displayed. If the value is lower than the LOW VALUE, PASS is displayed. If the value is lower than or equal to the HIGH VALUE **and** higher than or equal to the LOW VALUE, FAIL is displayed. See also the next illustration.



MAX: The maximum measured value is displayed.
MIN: The minimum measured value is displayed.
LOG(1/2): The result of the calculation

$$20 \ast \log \frac{measured\ value\ CH1}{measured\ value\ CH2}$$

is displayed.

LOG(2/1): The result of the calculation

$$20 * \log \frac{measured\ value\ CH2}{measured\ value\ CH1}$$

is displayed.

Displaying the measured values

The voltmeter can display the measured values several different ways, on 2 to 6 displays. By activating the choice **DISPLAYS** from the main menu a popup menu appears in which the number of displays can be set and the configuration of the displays can be set.

Setting the number of displays

By activating **NUMBER OF DISPLAYS** from the DISPLAYS menu, a popup menu appears in which the number of displays can be set. The possible options are 2, 4 and 6 displays, which means 1, 2 or 3 displays per channel. When **6 DISPLAYS** is chosen, three displays per channel are available, each separately configurable. The left three displays(display 1, 3 and 5) are for channel 1, the right three display (display 2, 4 and 6) are for channel 2.

Configuring the displays

With the choices **SET DISPLAY 1 .. 6** the settings for the displays are avaliable, via popup menus. Displays 1 and 2 have most settings, 2 to 6 have less possibilities. Display 1 is discussed here, if a discussed option is not available to certain displays, it is mentioned.

Processing the measured value

The software can process the measured value in 10 different ways. By activating the choice **MEASURING** from the menu SET DISPLAY 1 a popup menu appears with the following computations:

TRUE RMS: The true RMS value of the input signal is calculated. The true RMS value (Root Mean Square) is the value of a random voltage, corresponding to a DC voltage, dissipating the same power in a resistance in the same amount of time. So an AC voltage of 1 Volt RMS corresponds to a DC voltage of 1 Volt. (Most —cheap— voltmeters only display, when measuring AC voltage, the correct value when the input signal is a true sine wave and usually in a limited frequency range. The TP112 always displays the correct value.)

PEAK-PEAK: The peak-peak value of the input signal is determined. The peak-peak value of a DC voltage is zero. The peak-peak value of a true sine wave with a RMS value of 1 volt is 2.828 volt.

MEAN: The mean value of the input signal is determined. The mean value of a varying voltage corresponds with the value of a DC voltage transporting the same load. The mean value of a 1 volt DC voltage is 1 volt. The mean value of a true sine wave AC voltage is 0.

MAX: The maximum of the input signal is determined. The maximum value of a 1 volt DC voltage is 1 volt. The maximum value of a 1 volt AC sine wave voltage is 1.414 volt.

MIN: The minimum of the input signal is determined. The minimum value of a 1 volt DC voltage is 1 volt. The minimum value of a 1 volt AC sine wave voltage is -1.414 volt.

dBm: The value of the input signal is converted to dB. This is done using the formula:

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

'value' is the RMS value of the input signal. The reference resistance can be set using the choice SET IMPEDANCE from the CH1 SET menu.

POWER: The value of the input signal is converted to a power, using the formula:

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

'value' is the RMS value of the input signal. The reference resistance can be set using the choice SET IMPE-

DANCE from the CH1 SET menu.

CREST: The value of the input signal is converted to a crest factor, using the formula:

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

'peak value' is the highest voltage of the input signal with respect to the zero level.

FREQ.: The frequency of the input signal is determined. The frequentie range runs from ± 4.5 Hz to 100 kHz. At frequencies lower than 4.5 Hz, 0 Hz is displayed. To measure frequencies higher than 100 kHz the UNITS/VOLT for that display can be set to 0.001, kHz are displayed then. The unit has to be changed to kHz using UNITS.

DUTY CYCLE: The duty cycle of the input signal is determined. The duty cylce indicates how much of a signal is low in relation to the periodic time of the signal. The number is given in percent.

Note When frequency measurements are set, the voltmeter gets slower at each display, since for determining the frequency more measurements are needed.

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 appears, showing 13 choices:
VOLT, AMPERE, DEGREE F, DEGREE C, WATT, PERCENT, METER, KILOGRAM, NEWTON, COLOMB, BAR, HERTZ and '   '.

The empty choice can be used to enter a self defined unit. After activating, a dialog box appears in which the unit, maximum 5 characters, can be entered.

The popup menu can alo be called by placing the mouse cursor on top of the unit indicator and pressing the left mouse button.

If one of the choices is activated, the unit is indicated at the top of the display.

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

Changing the units per volt

By activating the choice UNITS/VOLT from the menu CH1 SET, a multiplication factor can be entered for the units of measure. A dialog box appears in which the factor can be entered. The measured value is multiplied by the factor and displayed.

If you are using a 10x probe, you can by means of UNITS/VOLT enter the value 10, to get a correct display.

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.

Relative measurements

The voltmater can take absolute measurements and relative measurements. When measuring relative, the measurements are related to a settable value. This value is subtracted from the measured value.

The value can be set by activating the choice SET REL. VALUE. A dialog box appears in which the value can be entered. The value has to be entered in volts.

For measuring relative, the choice RELATIVE = OFF has to be activated. The text will change to RELATIVE = ON and relative measurements are switched on. It is also indicated in the display, when relative measurements are switched on.

Activating RELATIVE = ON will change the text to RELATIVE = OFF and switch off relative measurements.

Relative measurements can be used for e.g. with the previously mentioned temperature measurement. Suppose the output voltage of the temperature probe at 0°C is +10 Volt. By entering a REL. VALUE of 10 volt, the absolute temperature is directly readable.

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.

These options are only available to Display 1 and Display 2.

Note For easy entering of LOW and HIGH VALUES, the software does not check whether the HIGH VALUE is higher than the LOW VALUE. So it is possible to enter a LOW VALUE which is higher than the HIGH VALUE. Measurements using the HIGH and LOW VALUE will then give useless results.

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 following dialog box appears:

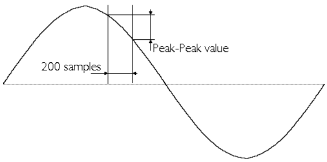
```
reference impedance for dBm and POWER measuring  (0.001...20000.000)

600.000_
```

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

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 choice **FREQUENCY** from the main menu, a popup menu appears, in which 10 frequencies are available. The frequencies indicate the center frequency of a frequency range in which the voltmeter measures properly. The size of the frequency range is determined from 0.2 x f_{center} .. 10 x f_{center}. When AUTO FREQ is chosen, the voltmeter determines the best frequency setting by itself. For each measurement the frequency has to be determined, resulting in a longer time before updating the displays.

The center frequencies available are: 10 Hz, 50 Hz, 100 Hz, 500 Hz, 1 kHz, 5 kHz, 10 kHz, 50 kHz and 100 kHz and AUTO FREQ.

Setting sounds

With comparison measurements, for each display value a sound signal can be set. Then also can be indicated audible whether a measurement is within the set limits or not.

By activating the choice SOUND from the main menu, a popup menu appears from which for each display value a sound signal can be set. For each display value can be chosen from: OFF, 100 Hz, 200 Hz, 500 Hz, 1 kHz, 2 kHz, 5 kHz and 10 kHz.

With the choice OFF no sound signal generated for the concerning value, with the other choices a tone with the corresponding frequency is generated.

Perform measurements

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

When the device Voltmeter is switched on, measurements take place. The measured values are displayed continuously. The text MEASURE in the main menu indicates measuring is taking place.

Measuring can be stopped by activating MEASURE. The text changes to FREEZE and measuring stops. Also a choice ONE SHOT appears.

Now a single measurement can be performed, by activating ONE SHOT, there is one measurement. During this measurement, the text changes from ONE SHOT to MEASURE. When measuring is finished, the text changes back to ONE SHOT.

Note One measurement consists of three successive measurements, from wich the measured values are averaged. Between the measurements the display is updated. If the voltmeter has to perform an AUTO ranging operation during a ONE SHOT measurement, measurements are taken until AUTO ranging is finished and three measurements are taken in the final input range setting.

Continuous measuring can be switched on by activating FREEZE. Measuring starts and the text changes to MEASURE.

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.

Setting the hysteresis

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

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

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.

Next follows an example of how the data is stored or printed.

```
MEASURING CHANNEL 1 : MEAN      (Hz)
MEASURING CHANNEL 2 : TRUE RMS  (V )
DISPLAY CHANNEL 1   : CH1     (REL. ON!)   1.0000
DISPLAY CHANNEL 2   : CH2
IMPEDANCE CHANNEL 1 :   600.0000
IMPEDANCE CHANNEL 2 :   600.0000
LOW VALUE CHANNEL 1 :   -20.0000
LOW VALUE CHANNEL 2 :   -20.0000
HIGH VALUE CHANNEL 1:    20.0000
HIGH VALUE CHANNEL 2:    20.0000
DATE     TIME     CHANNEL 1     CHANNEL 2
14-02-95 13:41:50  -------          2.15
14-02-95 13:41:55   -0.723          2.15
14-02-95 13:42:00   -0.684          2.14
14-02-95 13:42:05   -0.201          2.15
14-02-95 13:42:10   -0.119          2.15
```

Changing the display characters

By activating DIGIT SET, a popup menu appears in which 5 character sizes are available. With these the thickness of the segments of the display characters can be changed.

If e.g. 4 PIXELS is activated, the thickness of the segments will be 4 pixels. The available thicknesses are: 2, 4, 6, 8 and 10 pixels.

Chapter 9

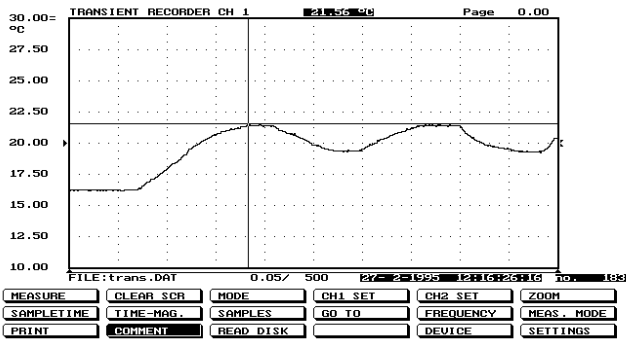
Transient recorder (device: TRANS)

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 each measured sample immediately, the oscilloscope first takes all samples and then displays the complete page.

The transient recorder display



Like with the other instruments, the screen is divided in two parts, a display part and the main menu.

The menu contains a number of choices also available in the oscilloscope and therefore will be not discussed here. The other choices (see the next illustration, the not-inverted parts) are discussed here.

Setting the measuring type

The transient recorder can measure in five different ways. By activating MEAS. MODE from the main menu, a popup menu appears, with the following choices:

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

The TRUE RMS, MEAN, MAX and MIN value are calculated from 200 samples. To avoid measuring errors, the frequency range has to be set properly (see also 'Setting the frequency range' in the chapter Voltmeter).

Note When measure mode TRUE RMS, MEAN, MAX or MIN is selected, the minimum sample time is 0.5 sec.

Setting the frequency range

The proper frequency range for the measurements can be set by activating FREQUENCY from the main menu. A popup menu appears, containing 9 center frequencies. The frequencies indicate the center frequency of a frequency range in which the transient recorder measures properly. The size of the frequency range is determined from $0.2 \times f_{center}$ .. $10 \times f_{center}$.

The center frequencies available are: 10 Hz, 50 Hz, 100 Hz, 500 Hz, 1 kHz, 5 kHz, 10 kHz, 50 kHz and 100 kHz.

Note When the center frequency is set to 10 Hz, the minimum sampling time is 1.5 sec.

Setting the sampling time

The time between two samples can be set using SAMPLETIME from the main menu. After activating, a dialog box appears in which the time can be entered. The value has to be in the range from 0.01 sec to 300 sec. With the number of samples at 30000, the maximum measuring time will be 300 * 30000 = 9000000 seconds (= 104.16 days).

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

Note When measure mode TRUE RMS, MEAN, MAX or MIN is selected, the minimum sample time is 0.5 sec.
When the center frequency is set to 10 Hz, the minimum sampling time is 1.5 sec.

Setting the number of samples

The number of samples to take can be set by activating the choice SAMPLES from the main menu. A dialog box appears in which the number of samples can be entered. The value must be in the range from 0 to 30000. With a sample time of 300 sec, the maximum measuring time will be 300 * 30000 = 9000000 seconds (= 104.16 days).

Note All measurement values are first stored in memory of the PC. If insufficient memory is available, the program will display a warning: `Error: Not enough memory, decrease number of samples`. You have to enter a smaller number of samples or free memory by removing memory resident programs (TSR's).

Clear the screen

Old measurement data can be removed from the screen by activating the choice CLEAR SCREEN from the main menu.

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: `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 Sometimes 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 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 500 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, each time the complete signal is read from disk, may take some time.

Go to a specific sample

By activating GO TO from the main menu, a sample number can be entered. The page containing the sample with the entered number will be displayed. When e.g. a measurement is taken with 15000 samples, the sample with the number 14000 can be brought onto the display by entering 14000.

Timebase oscilloscope and transient recorder

sample speed	timebase	resolution
1 MSamples/sec	50 μsec/div	1 μsec
500 kSamples/sec	0.1 msec/div	2 μsec
250 kSamples/sec	0.2 msec/div	4 μsec
100 kSamples/sec	0.5 msec/div	10 μsec
50 kSamples/sec	1 msec/div	20 μsec
25 kSamples/sec	2 msec/div	40 μsec
10 kSamples/sec	5 msec/div	100 μsec
5 kSamples/sec	10 msec/div	200 μsec
2.5 kSamples/sec	20 msec/div	400 μsec
1 kSamples/sec	50 msec/div	1 msec
500 Samples/sec	0.1 sec/div	2 msec
250 Samples/sec	0.2 sec/div	4 msec
100 Samples/sec	0.5 sec/div	10 msec
50 samples/sec	1 sec/div	20 msec
25 samples/sec	2 sec/div	40 msec
10 samples/sec	5 sec/div	100 msec

sample speed	frequency range spectrum analyzer	resolution
1 MSamples/sec	500 kHz	976 Hz
0.5 MSamples/sec	250 kHz	488 Hz
0.2 MSamples/sec	125 kHz	244 Hz
50 kSamples/sec	50 kHz	97.6 Hz
500 kSamples/sec	25 kHz	48.8 Hz
250 kSamples/sec	12.5 kHz	24.4 Hz
125 kSamples/sec	5000 Hz	9.76 Hz
50 kSamples/sec	2500 Hz	4.88 Hz
25 kSamples/sec	1250 Hz	2.44 Hz
12.5 kSamples/sec	500 Hz	0.976 Hz
5 kSamples/sec	250 Hz	0.488 Hz
2.5 kSamples/sec	125 Hz	0.244 Hz
1.25 kSamples/sec	50 Hz	0.0976 Hz
500 Samples/sec	25 Hz	0.0488 Hz
250 Samples/sec	12.5 Hz	0.0244 Hz
125 Samples/sec	5 Hz	0.0097 Hz

sensitivity Volts/div	full scale sensitivity	resolution
312.5 mV/div	1.25 V	0.610 mV
625 mV/div	2.5 V	1.221 mV
1.25 V/div	5 V	2.441 mV
2.5 V/div	10 V	4.883 mV
5 V/div	20 V	9.766 mV

Appendix B
Technical information

Use of the addresses

The TP112 uses 8 I/O addresses of the PC. These addresses are successive. The base address can be set from $000 to $3F8.

The TP112 contains 3 FPGAs (Field Programmable Gate Arrays), which control the TP112 completely. Each FPGA has a control register in which a controll word can be stored. This controll word determines which DATA register will be active (256 DATA registers at most are available per FPGA). The DATA registers contain the TP112 settings.

At power on of the PC, several data and control registers of the TP112 are loaded with a fixed value.

The TP112 contains three CONTROL registers and a number of DATA registers. The CONTROL register determines which DATA register will be available. The CONTROL and DATA registers are write-only.

The I/O map is:
- base address +0: CONTROL register 1 (of FPGA 1).
- base address +1: DATA register indicated by control register 1.
- base address +4: CONTROL register 2 (of FPGA 2).
- base address +5: DATA register indicated by control register 2.
- base address +6: CONTROL register 3 (of FPGA 3).
- base address +7: DATA register indicated by control register 3.

Because of this way of interfacing, only 8 I/O addresses are used, while 3x256 = 768 I/O addresses are available in the TP112.

CONTROL register 1

The CONTROL register 1 is reached at I/O base address +0.

The byte of the CONTROL register means:

D2 ... D0	000:	Activate the POST-LO register.
	001:	Activate the POST-HI register.
	010:	Activate the OUTPUT register.
	011:	Activate the TIMEBASE register.
	100:	Activate the MODE register.
	101:	Activate the RESET register.
	110:	Activate the COUNT register.
	111:	Activate the WRITE TIMEBASE register.
D3 ... D7	:	Must be zero.

(FPGA 1 has 7 DATA registers.)

Example:

```
port[$300+0]:=010; {output register is active}
```

DATA registers of CONTROL register 1

The DATA registers of CONTROL register 1 are reached at I/O base address +1.

NB: First a byte has to be written in the CONTROL register indicating which DATA register will be coupled to the PC data bus. After writing the CONTROL register, the associated DATA register is connected to the PC data bus. Now data can be written to the DATA register.
Three CONTROL registers are available. Each control register can couple a number of DATA registers to the PC data bus.

POST-LO register (CONTROL 1 = 000)

In the POST-LO register the lower-byte of the post counter is stored.
The byte in the POST-LO register correspond with postcounter bits POST_D0 ... POST_D7.

POST-HI register (CONTROL 1 = 001)

In the POST-HI register, the higher-byte of the post counter is stored. The byte in the POST-HI register corresponds with post counter bits POST_D8 ... POST_D15.

The SRAM on the TP112 is controlled by the address counter. The address counter can be set to zero and the value of the address counter can be read. Before starting a measurement, the counter will be set to zero. Then the measurement is started. As soon as the trigger condition is met, the post counter (POST_D0 ... POST_D15) is started. The post counter will count until the number set in the LO-POST and HI-POST register is reached. Then measuring is stopped. The number in the LO-POST and HI-POST register determine the numbers measured after the triggerpoint.

OUTPUT register (CONTROL = 010)

In the OUTPUT register the status of the outputs is stored.

The byte of the OUTPUT register has the following meaning:

D0	0	the relay contact of output 0 is open
	1	the relay contact of output 0 is closed
D1	0	the relay contact of output 1 is open
	1	the relay contact of output 1 is closed
D2	0	the relay contact of output 2 is open
	1	the relay contact of output 2 is closed
D3	0	the relay contact of output 3 is open
	1	the relay contact of output 3 is closed
D4	0	the relay contact of output 4 is open
	1	the relay contact of output 4 is closed
D5	0	the relay contact of output 5 is open
	1	the relay contact of output 5 is closed
D6	0	the relay contact of output 6 is open
	1	the relay contact of output 6 is closed
D7	0	the relay contact of output 7 is open
	1	the relay contact of output 7 is closed

Example:

```
port[$300+0] := 2;     {select the OUTPUT register }
port[$300+1] := 1;     { output 0 on, the rest off }
```

TIMEBASE register (CONTROL = 011)

On the TP112, a timer IC 8253 is used. This IC contains three 16 bit counters (for more information, see the 8253 Data sheets). The counters are used for generating the timebase. The gates of counters 1 is controlled by a 4 MHz signal. The output of counter 1 is connected to the gate of counter 2. The output of counter2 is connected to the gate of counter 3. The sample speed can be set between 1 MHz and less than 1 sample a year.

The databits D0 and D1 of the timebase register control A0 and A1 of the 8253. The databits D2 and D4 are for selecting the timebase clock.

The byte of the TIMEBASE register means:

D0	0	:	A0 of timer 8253 is low
	1	:	A0 of timer 8253 is high
D1	0	:	A1 of timer 8253 is low
	1	:	A1 of timer 8253 is high
D2...D3	00	:	Sample clock is output of timer 1
	01	:	Sample clock is output of timer 2
	10	:	Sample clock is output of timer 3
	11	:	reserved
D4..D7		:	Reserved (must be 0)

MODE register (CONTROL = 100)

The byte of the MODE register means:

D0	0	:	Put sample clock off
	1	:	Put sample clock on
D1	0	:	Put sample clock through de PC
	1	:	Put sample clock internal
D2	0	:	Data from SRAM to PC bus
	1	:	Data from A/D convertor to SRAM and to PC bus
D3	0	:	Put the A/D convertor clock off

	1	:	Put the A/D convertors clock on
D4	0	:	Normal clock to A/D convertor 1
	1	:	Inverted clock to A/D convertor 1
D5	0	:	Normal clock to A/D convertor 2
	1	:	Inverted clock to A/D convertor 2
D6, D7	00	:	Trigger on channel 1
	01	:	Trigger on channel 2
	10	:	Trigger immediately
	11	:	reserved

RESET register (CONTROL = 101)

The byte of the RESET register means:

D0 ... D7	x	:	Reset address counter and post counter

COUNT register (CONTROL = 110)

This byte of the COUNT register means:

D0 ... D7	X	:	Increase the address counter by 1

WRITE TIMEBASE register (CONTROL = 111)

This byte of the WRITE TIMEBASE register means:

D0 ... D7		:	See 8253 datasheet

This byte is loaded in the IC 8253

CONTROL register 2 and 3

The CONTROL registers 2 and 3 are almost identical. CONTROL register 2 is used for controlling channel 1. CONTROL register 3 is used for controlling channel 2. CONTROL register 2 will be discussed here. Differences between CONTROL register 2 and 3 will be placed between brackets.

CONTROL register 2 is reached at I/O base address +4.
CONTROL register 3 is reached at I/O base address +6.

The byte of the CONTROL means:

D0 ... D2	000	:	Select LEVEL-1 register
	001	:	Select LEVEL-2 register
	010	:	Select TRIG-NIV register
	011	:	Select PGA register
	100	:	Select DATA-ZERO register
	101	:	Select IRQ-ENABLE register
	110	:	Select TRIG-RESET register
	111	:	Reserved.
D3 ... D7		:	Must be 0

DATA registers of CONTROL register 2 and 3

The DATA registers of CONTROL register 2 are reached at I/O base address +5.
The DATA registers of CONTROL register 3 are reached at I/O base address +7.

LEVEL-1 register (CONTROL = 000)

The LEVEL -1 register is for the first trigger level
The byte of the LEVEL-1 register means:

The byte is compared to the databits D4 .. D11 of AD convertor 1.

LEVEL-2 register (CONTROL = 001)

The LEVEL -2 register is for the second trigger level.
The byte of the LEVEL-2 register means:

The byte is compared to the databits D4 .. D11 of AD convertor 1.

In the TP112 two levels concerning the triggerlevel can be set: the arming and the firing level. Both levels are compared to the databits D4..D11 of the ADC. (Data bits D0 .. D3 are not used fot triggering) The result of this comparison can be higher or lower than the level set. Four comparisons are possibe. The four results are processed by the trigger logic. This makes triggering on a positive and a negative slope possible. The trigger logic generates a bit (the trigger bit) indicating the trigger status. If the bit is 0, the trigger conditions have not yet been met. If the bit is 1, the trigger conditions have been met.

TRIG-NIV register (CONTROL = 010)

The byte of the TRIG-NIV register means:

D0	0	:	Level -1 higher than A/D data.
	1	:	Level -1 lower than A/D data.
D1	0	:	Level -2 higher than A/D data.
	1	:	Level -2 lower than A/D data.
D2 ... D3	00	:	Trigger on digital inputs
	01	:	Trigger mode AND.
	10	:	Trigger mode OR.
	11	:	Reserved.
D4 ... D5	00	:	Trigger on digital input 0
	01	:	Trigger on digital input 1
	10	:	Trigger on digital input 2
	11	:	Trigger on digital input 3
D6..D7		:	reserved, must be 0

PGA register (CONTROL = 011)

With the PGA REGISTER, the input sensitivity and the coupling of each channel can be set.

The byte of the SERIAL register means:

D2..D0	000	:	Input sensitivity 1.25 Volt
	001	:	Input sensitivity 2.50 Volt
	010	:	Input sensitivity 5.00 Volt
	011	:	Input sensitivity 10.0 Volt
	100	:	Input sensitivity 20.0 Volt
D3	0	:	AC coupling
	1	:	DC coupling
D4..D7		:	Reserved.

DATA ZERO register (CONTROL = 100)

The byte of the DATA ZERO register means:

D1	0	:	A/D data is zero.
	1	:	A/D data is passed through.
D2 ... D7		:	Reserved.

IRQ ENABLE register (COTROL = 101)

The byte of the IRQ ENABLE register has the following meaning:

D1	0	:	The inputs 0..7 are not passed through to the interrupt line (inputs have an OR function)
	1	:	The inputs 0..7 are passed through to the interrupt line (inputs have an OR function)
D2..D7		:	reserved (must be 0)

TRIG-RESET register (CONTROL = 101)

The byte of the TRIG-RESET register means:

D0 ... D7	X	:	Reset the triggerbit (triggerbit is 0).

I/O base address + 0

A0 ... A7 of the address counter are read

D0	D1	D2	D3	D4	D5	D6	D7
A0	A1	A2	A3	A4	A5	A6	A7

I/O base address + 1

A8 ... A14 of the address counter are read

D0	D1	D2	D3	D4	D5	D6	D7
A8	A9	A10	A11	A12	A13	A14	x

I/O base address +2

EOC, TRIG1and TRIG2 of the address counter are read

D0	D1	D2	D3	D4	D5	D6	D7
EOC	TRIG1	TRIG2	0	0	0	0	0

EOC indicates the conversion to be ready
TRIG1 indicates the trigger condition of channel 1 is met
TRIG2 indicates the trigger condition of channel 2 is met

I/O base address + 3

This address is not used, all bits read zero

D0	D1	D2	D3	D4	D5	D6	D7
0	0	0	0	0	0	0	0

I/O base address + 4

D0..D7 of the ADC (channel 1) are read or D0..D7 of the SRAM (channel 1) are read. See MODE REGISTER of CONTROL REGISTER 1.

D0	D1	D2	D3	D4	D5	D6	D7
D0	D1	D2	D3	D4	D5	D6	D7

I/O base address + 5

D8..D11 of the ADC (channel 1) and digital inputs IN0 .. IN3 are read or D8..D11 and digital inputs IN0 .. IN3 of the SRAM (channel 1) are read. See MODE REGISTER of CONTROL REGISTER 1.

D0	D1	D2	D3	D4	D5	D6	D7
D8	D9	D10	D11	IN0	IN1	IN2	IN3

I/O base address + 6

D0..D7 of the ADC (channel 2) are read or D0..D7 of the SRAM (channel 2) are read. See MODE REGISTER of CONTROL REGISTER 1.

D0	D1	D2	D3	D4	D5	D6	D7
D0	D1	D2	D3	D4	D5	D6	D7

I/O base address + 7

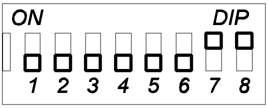
D8..D11 of the ADC (channel 2) and digital inputs IN4 .. IN7 are read or D8..D11 and digital inputs IN4 .. IN7 of the SRAM (channel 2) are read. See MODE REGISTER of CONTROL REGISTER 1.

D0	D1	D2	D3	D4	D5	D6	D7
D8	D9	D10	D11	IN4	IN5	IN6	IN7

Setting the I/O base address

The I/O base address can be set to any address between $000 and $3F8, in steps of 8 addresses. For that the address lines A3 .. A9 of the PC are necessary. The address lines A0 .. A2 are used for selecting the next 8 addresses of the card. Setting the base I/O address is done with dipswitch SW1.

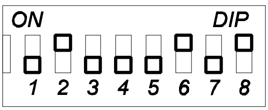
The factory setting is $300, which corresponds with a dipswitch setting as in the next illustration.



Calculation of address $300:

```
A0  A1  A2  A3 | A4  A5  A6  A7 | A8  A9
x   x   x   0  | 0   0   0   0  | 1   1
      $0       |       $0       |   $3
```

If the address $300 is already in use in your PC, you can set the TP112 to another address. Here follows an example for address $288



Calculation of the address $288:

```
A0  A1  A2  A3 | A4  A5  A6  A7 | A8  A9
x   x   x   1  | 0   0   0   1  | 0   1
      $8       |       $8       |   $2
```

Note If the base I/O address is changed, it also has to be changed in the software. See Setting the TP112 address.

Jumper J10: channel 1

When pins 1 and 2 are shorted, the input range of channel 1 is unipolar, when pins 2 and 3 are shorted, the input range is bipolar.

factory default is bipolar: pins 2 and 3 shorted.

	pins 1 and 2 shorted	pins 2 and 3 shorted
input range	0 .. 1.25 V	-1.25 V .. 1.25 V
input range	0 .. 2.50 V	-2.50 V .. 2.50 V
input range	0 .. 5.00 V	-5.00 V .. 5.00 V
input range	0 .. 10.0 V	-10.0 V .. 10.0 V
input range	0 .. 20.0 V	-20.0 V .. 20.0 V

Jumper J11: channel 2

When pins 1 and 2 are shorted, the input range of channel 2 is unipolar, when pins 2 and 3 are shorted, the input range is bipolar.

factory default is bipolar: pins 2 and 3 shorted.

	pins 1 and 2 shorted	pins 2 and 3 shorted
input range	0 .. 1.25 V	-1.25 V .. 1.25 V
input range	0 .. 2.50 V	-2.50 V .. 2.50 V
input range	0 .. 5.00 V	-5.00 V .. 5.00 V
input range	0 .. 10.0 V	-10.0 V .. 10.0 V
input range	0 .. 20.0 V	-20.0 V .. 20.0 V

Interrupt request

With the IRQ2, IRQ3, IRQ4, IRQ5, IRQ6 and IRQ7 jumpers an interrupt can be selected.
Choose a free interrupt and avoid conflicts with other hardware in your computer.

Pin assignments of the 37 pin connector

1	:	Input	A/D Ch1
2	:	Ground	
3	:	Input	A/D Ch2
4..6	:	Ground	
7	:	Output	+ output 0
8	:	Output	+ output 1
9	:	Output	+ output 2
10	:	Output	+ output 3
11	:	Output	+ output 4
12	:	Output	+ output 5
13	:	Output	+ output 6
14	:	Output	+ output 7
15	:	Input	Digital input 0
16	:	Input	Digital input 2
17	:	Input	Digital input 4
18	:	Input	Digital input 6
19	:	Input	Ground
20..25	:	Ground	
26	:	Output	- output 0
27	:	Output	- output 1
28	:	Output	- output 2
29	:	Output	- output 3
30	:	Output	- output 4
31	:	Output	- output 5
32	:	Output	- output 6
33	:	Output	- output 7
34	:	Input	Digital input 1
35	:	Input	Digital input 3
36	:	Input	Digital input 5
37	:	Input	Digital input 7

Appendix C
Specifications

Hardware

AD convertor

The TP112 is equipped with two independent channels. The specifications mentioned are for both channel 1 and channel 2.

samplefrequency	1 MHz
conversion time	1 μsec
memory	32 Kbyte
input sensitivity (for channel 1 and 2) unipolar and bipolar	
	1.25 volt (resolution 305.1 μvolt)
	2.50 volt (resolution 610.0 μvolt)
	5.00 volt (resolution 1.22 mvolt)
	10.0 volt (resolution 2.44 mvolt)
	20 volt (resolution 4.88 mvolt)
maximum input voltage	200 volt AC 50 Hz
input impedance	1 Mohm/20 pF
coupling	AC/DC
resolution	12 bits 0.025 %
linearity	12 bits
accuracy	0.1 % $\pm$ 1 LSB
bandwidth	DC to 500 kHz

Outputs

TP112-N	TTL level, 5 mA
TP112-R	0-5 A, 50 V max
	Output impedance 1 Ω
	Switching frequency 10 Hz
TP112-S	0-5 A, 50 V max
	Output impedance 1 Ω
	Switching frequency 1 kHz

Inputs

TP112-N	TTL level
TP112-R and TP112-S	0..1 Volt is logically 0
	3..24 Volt is logically 1
	Output impedance 1 kΩ
	Switching frequency 5 kHz

General

recommended ambient temperature		20 °C
dimensions	height	125 mm / 4.9 "
	length	250 mm / 9.8 "
	width	20 mm / 0.8 "
weight		approx. 250 gram/10 oz

Spectrum analyzer

frequency range	0.009	-	5	Hz (resolution	0.009 Hz)
	0.024	-	12.5	Hz (resolution	0.024 Hz)
	0.048	-	25	Hz (resolution	0.048 Hz)
	0.097	-	50	Hz (resolution	0.09 Hz)
	0.244	-	125	Hz (resolution	0.24 Hz)
	0.488	-	250	Hz (resolution	0.48 Hz)
	0.976	-	500	Hz (resolution	0.97 Hz)
	0.002	-	1.25	kHz (resolution	2.44 Hz)
	0.004	-	2.5	kHz (resolution	4.88 Hz)
	0.009	-	5	kHz (resolution	9.76 Hz)
	0.024	-	12.5	kHz (resolution	24.4 Hz)
	0.048	-	25	kHz (resolution	48.8 Hz)
	0.097	-	50	kHz (resolution	97.6 Hz)
	0.244	-	125	kHz (resolution	244 Hz)
	0.488	-	250	kHz (resolution	488 Hz)
	0.976	-	500	kHz (resolution	976 Hz)

frequency axis magnification	2 ... 20 times
frequency accuracy	0.01 % full scale
y-axis	linear/dB
windows	rectangle window
	Hanning window
	Hamming window
	Blackman window
	Bartlett window
distortion	calculation over 10 harmonics relative to the base frequency
hardcopy	matrix- and laser printer
averaging	1 ... 200 spectra

Oscilloscope

timebase: 50 µsec, 0,1 msec, 0,2 msec, 0,5 msec, 1 msec, 2 msec, 5 msec, 10 msec, 20 msec, 50 msec, 100 msec, 200 msec, 500 msec, 1 sec, 2 sec and 5 sec per division.

accuracy timebase	0.01 %
time axis magnification	2 ... 20 times
setting accuracy timebase magnification	0.25 %
y-axis magnification	-5 ... 5 times
triggering	channel 1, channel 2, rising and falling slope
trigger sensitivity	0.25 ... 2.25 DIV
trigger resolution	0.01 times full scale
trigger time-out	0 ... infinite
external triggering	keyboard
offset adjustment	channel 1 and 2 independent
offset range	-200 ... 200 %
offset resolution	0.01 times full scale
invert	channel 1 and 2 independent
add	channel 1 + 2
compare	channel 1 relative to channel 2
xy-mode	channel 1 = x-axis, channel 2 = y-axis
hardcopy	matrix- and laser printer

TRUE RMS voltmeter

measuring method	true RMS, mean, peak-peak, min, max, momental, dBm, power, crest factor, frequency and duty cycle
accuracy	2 % of full scale + 1 LSB
display methods:	CH1, CH2, CH1*CH2, CH1/CH2, CH2/CH1, CH2-CH1, CH1+CH2, > then, < then, >< COMP, <>HI LO, MAX, MIN, LOG(CH1/CH2) and LOG(CH2/CH1)
display height	± 25 .. 50 mm, depending on the monitor
frequency range	10 Hz, 50 Hz, 100 Hz, 500 Hz, 1 kHz, 5 kHz, 10 kHz, 100 kHz
data aqcuire	to disk and printer

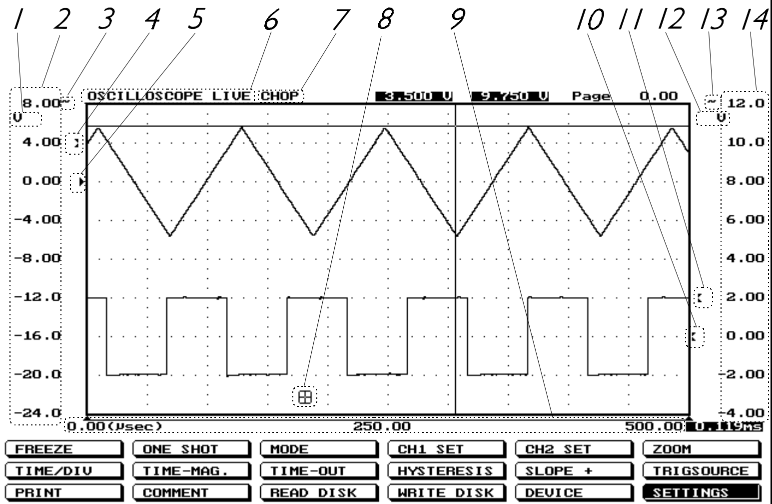
Transient recorder

sampling time	0.01 sec ... 300 sec
	At a PC-XT, the minimum time is 0.05 sec
maximum number of samples	30,000
time axis magnification	2 ... 20 times
y-as magnification	-5 ... 5 times
offset adjustment	channel 1 and 2 independently
offset range	-200 ... 200 %
offset resolution	0,01 times full scale
invert	channel 1 and 2 independent
add	channel 1 + 2
compare	channel 1 relative to channel 2
measure method	true RMS, mean, min, max and momental
hardcopy	matrix- and laser printer

Position of the hot spots

Clicking the left mouse button at the indicated places causes a change of:

1 *Units of measure channel 1*
UNITS
2 *Vertical sensitivity channel 1*
VOLTS/DIV
3 *Signal coupling channel 1*
AC/DC
4 *Trigger level channel 1 (drag)*
5 *Software offset channel 1 (drag)*
6 *Measurement instrument*
DEVICE
7 *Display mode*
MODE
9 *Time base setting*
SEC/DIV
10 *Software offset channel 2 (drag)*
11 *Trigger level channel 2 (drag)*
12 *Units of measure channel 2*
UNITS
13 *Signal coupling channel 2*
AC/DC
14 *Vertical sensitivity channel 2*

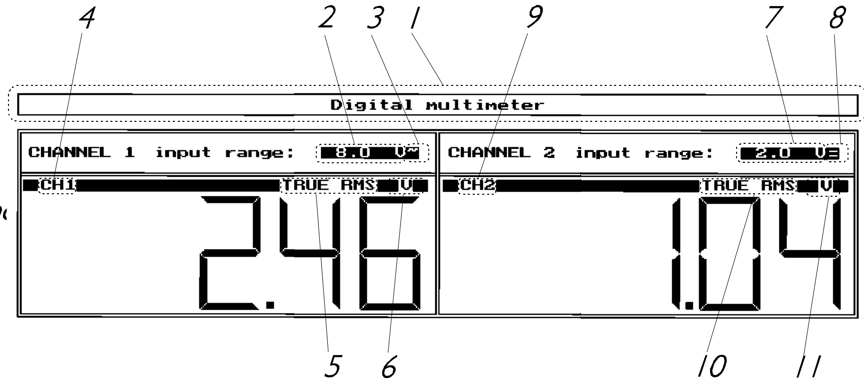


Clicking the left mouse button at the indicated places causes a change of:

1 *Measurement instrument*
DEVICE

2 *Input range channel 1*
INPUT RANGE
3 *Signal coupling channel 1*
AC/DC
4 *Display method channel 1*
DISPLAY
5 *Measurement method chann*
MEASURE
6 *Unit of measure channel 1*
UNITS

7 *Input range channel 2*
INPUT RANGE
8 *Signal coupling channel 2*
AC/DC
9 *Display method channel 2*
DISPLAY
10 *Measurement method channel 2*



notes:

If you have any suggestions and/or remarks concerning the program, the TP112 or the manual, please contact:

TiePie engineering
PO Box 290
8600 AG SNEEK

Visitors address:

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

E-mail: tiepie@tiepie.nl
Site: http://www.tiepie.nl