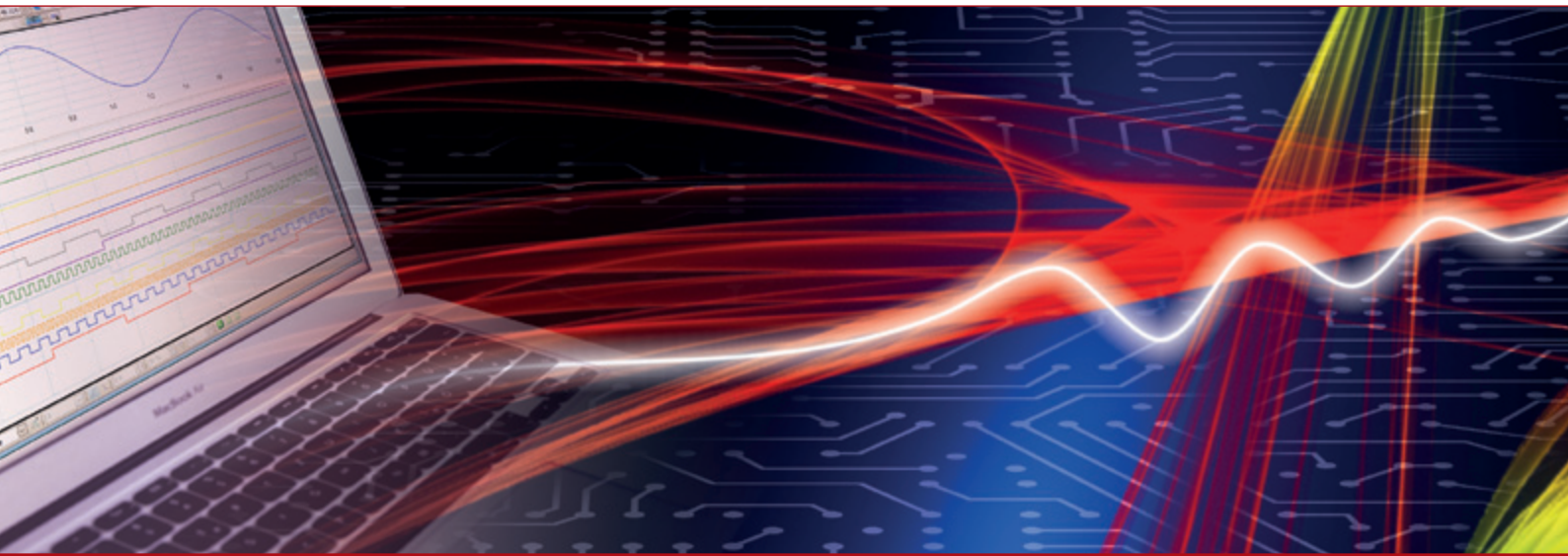


Product Datasheet - Technical Specifications



More information in our Web-Shop at ► www.meilhaus.com

Your contact

Technical and commercial sales, price information,
quotations, demo/test equipment, consulting:

Tel.: **+49 - (0)81 41 - 52 71-0**

FAX: **+49 - (0)81 41 - 52 71-129**

E-Mail: sales@meilhaus.com

Meilhaus Electronic GmbH
Am Sonnenlicht 2
82239 Alling/Germany

Tel. **+49 - (0)81 41 - 52 71-0**
Fax **+49 - (0)81 41 - 52 71-129**
E-Mail sales@meilhaus.com

Mentioned company and product names may be registered trademarks of the respective companies. Errors and omissions excepted. © Meilhaus Electronic.

www.meilhaus.com

PeakTech®

Unser Wert ist messbar...



PeakTech® I286, I326, I331

User manual
PC Oscilloscope

1. Safety instructions for operating the appliance

This product complies with the requirements of the following European Union Directives for CE conformity: 2014/30/EU (Electromagnetic Compatibility), 2014/35/EU (Low Voltage), 2011/65/EU (RoHS) with the Commission's amendment of Directive (EU) 2015/863. Overvoltage category II; pollution degree 2.

To ensure the operational safety of the unit and to avoid serious injuries due to current or voltage surges or short circuits, it is essential to observe the following safety instructions when operating the unit.

Damage caused by non-observance of these instructions is excluded from claims of any kind.

General:

- Read these operating instructions carefully and make them available to subsequent users.
- It is essential to observe the warning notices on the unit; do not cover or remove them.
- Pay attention to the use of the unit and use it only in its appropriate overvoltage category.
- Familiarise yourself with the functions of the meter and its accessories before taking the first measurement.
- Do not operate the meter unattended or secured against unauthorised access.
- Use the unit only for its intended purpose and pay particular attention to warnings on the unit and information on maximum input values.

Electrical safety:

- Voltages above 25 VAC or 60 VDC are generally considered dangerous voltage.
- Work on dangerous voltages only by or under the supervision of qualified personnel.
- When working on dangerous voltages, wear suitable protective equipment and observe the relevant safety rules.
- Do not exceed the maximum permissible input values under any circumstances (serious risk of injury and/or destruction of the unit).
- Pay special attention to the correct connection of the test leads depending on the measuring function to avoid a short circuit in the unit.
- Remove the test probes from the object to be measured before changing the measuring function.
- Never touch the bare test probes during measurement, only hold the test leads by the handle behind the finger guard.
- Discharge any capacitors present before measuring the circuit to be measured.
- Note the different properties of the galvanic isolation of the various interfaces and connections.

Measurement environment:

- Avoid any proximity to explosive and flammable substances, gases and dust. An electric spark could cause an explosion or deflagration - danger to life!
- Do not carry out measurements in corrosive environments, the unit could be damaged or contact points inside and outside the unit could corrode.

- Avoid working in environments with high interference frequencies, high-energy circuits or strong magnetic fields, as these can negatively affect the unit.
- Avoid storage and use in extremely cold, humid or hot environments, as well as long-term exposure to direct sunlight.
- Use units in humid or dusty environments only according to their IP protection class.
- If no IP protection class is specified, use the unit only in dust-free and dry indoor areas.
- When working in damp or outdoor areas, take special care to ensure that the handles of the test leads and test probes are completely dry.
- Before starting measurement operation, the unit should be stabilised to the ambient temperature (important when transporting from cold to warm rooms and vice versa).

Maintenance and care:

- Never operate the appliance if it is not completely closed.
- Before each use, check the unit and its accessories for damage to the insulation, cracks, kinks and breaks. If in doubt, do not take any measurements.
- Maintenance and repair work on the unit may only be carried out by qualified personnel.
- Clean the cabinet regularly with a damp cloth and a mild detergent. Do not use corrosive abrasive cleaners.
- Do not make any technical changes to the unit.

2. Safety symbols and terms

2.1 Safety symbols

You can find the following symbols in this operating manual or on the meter.



WARNING!

"Warning" indicates conditions and operating steps that pose a danger to the operator.



CAUTION!

"Caution" indicates conditions and operations that may cause damage to the product or other property.

Danger: High voltage



See operating instructions



Protective conductor terminal



Unit dimensions



Earth terminal (Earth)



This product can only be used in the specified applications. Read the following safety instructions before operating the unit.



Warning:

To avoid fire or electric shock, please use the correct power supply. Only use the power supply unit from the manufacturer.



Warning:

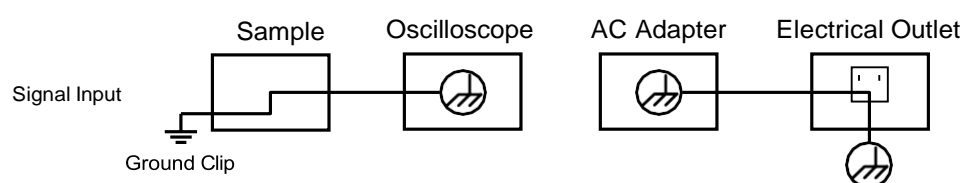
The channels of the oscilloscope are not electrically separated. The channels should apply common ground when measuring. To prevent short circuits, the ground contacts of the probes must not be connected to different non-isolated DC levels.



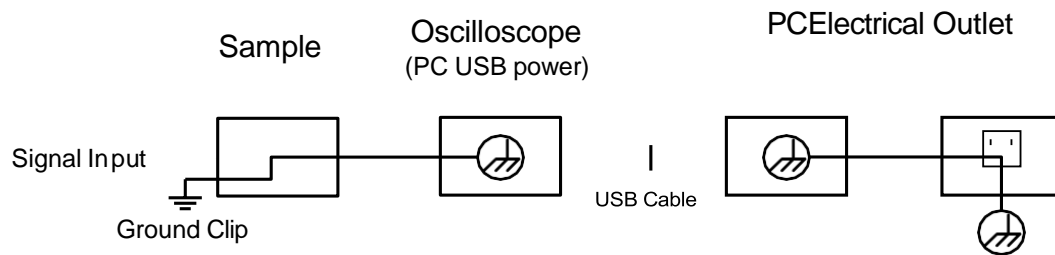
Warning:

The channels should use a common ground in measurement mode. To prevent short circuits, the ground contacts of the probes must not be connected to different non-insulated DC levels.

The oscilloscope earth wire connection device diagram:



The diagram of the earth wire connection setup when the oscilloscope is powered via the PC's USB port:



It is not allowed to measure from the mains while the oscilloscope is powered by the AC adapter or when the oscilloscope is powered from the USB port of the PC:

Warning:

If the oscilloscope is connected to an input signal greater than 42V peak (30Vrms) or circuits greater than 4800VA, please follow the instructions below to avoid fire or electric shock:

- Only use insulated probes and test leads.
- Check all accessories before use and replace if damaged. If in doubt, do not take any measurements.
- Remove the USB cable connecting the oscilloscope to the computer.
Never exceed the maximum specified input voltages. As the voltage is transmitted directly to the oscilloscope with the aid of the probe, the unit may be damaged or there is a risk of injury from electric shock.
- Do not use exposed BNC or banana plugs.
- Do not insert any metal objects into the connections.

To avoid fire or electric shock when a probe input is connected to more than 42V peak (30 Vrms) or to circuits of more than 4800VA:

- Use only insulated probes and adapters supplied with the meter or specified for the oscilloscope.
- Before use, check probes and accessories for mechanical damage and replace if damaged.
- Remove all probes and accessories from the oscilloscope when not in use.
- Do not apply voltages greater than 400 V difference to earth potential to the inputs when measuring in a CAT II environment.
- Do not apply voltages greater than 400 V difference between the isolated inputs when measuring in a CAT II environment.
- Do not apply input voltages above the oscilloscope's rating. Be careful when performing 1:1 tests, as the voltage is transmitted directly to the oscilloscope via the probe tip.
- Do not touch the exposed part of the BNC inputs.
- Do not insert any metal objects into the connections.
- Always use the oscilloscope in the manner specified.
- Rated voltages mentioned in the warning are the limits for "operating voltage". The specified V ACrms (50-60Hz) for AC sinusoidal applications and as V DC for DC applications. Overvoltage category II refers to local level, which becomes applicable for appliances and portable devices.

3. Device overview

3.1 P 1286



1. AC power adapter connection
2. USB Host Port: For WiFi Dongle (Optional)
3. USB port: For direct connection to the PC
4. LAN Port: Network port for integrating the oscilloscope into a network.
5. Probe compensation: test signal output (3.3/1KHz)
6. Multi port: trigger output; pass/fail output; EXT trigger input, function generator
7. Channel 2 Signal Input
8. Channel 1 Signal Input

Note: If you operate the oscilloscope in a network and thus connect the unit to a PC via the LAN port, it is necessary to supply the oscilloscope with power using the AC power adapter.

3.2 P 1326 / P 1331



1. AC power adapter connection
2. USB Host Port: For WiFi Dongle (Optional)
3. USB port: For direct connection to the PC
4. LAN Port: Network port for integrating the oscilloscope into a network.
5. Probe compensation: test signal output (3.3/1KHz)
6. Channel 4 Signal Input
7. Channel 3 Signal Input
8. Channel 2 Signal Input
9. Channel 1 Signal Input

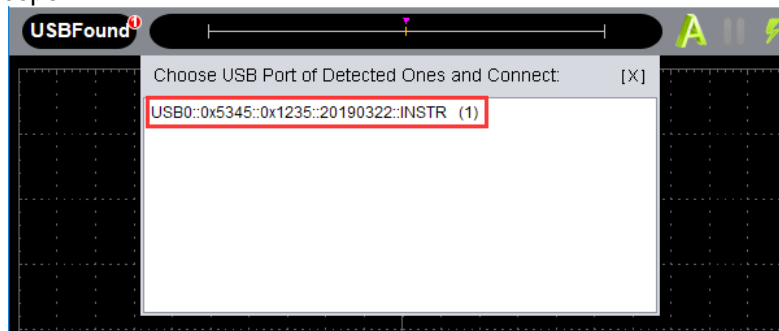
Note: If you operate the oscilloscope in a network and thus connect the unit to a PC via the LAN port, it is necessary to supply the oscilloscope with power using the AC power adapter.

4. Connection with the PC

To operate the PC oscilloscope, you need the latest version of the software and the latest driver. The software for the oscilloscope can be found on the CD included in the scope of delivery or on our homepage, PeakTech.de on the product page. The installation of the driver is explained in subitem **6 PeakTech® Oscilloscope Software Help.**

To install the software, open the installation file and execute it. The installation of the software is then carried out automatically.

After installation, open the software and connect the PC oscilloscope to the computer using the USB cable supplied. In the upper left corner of the software, you will now see that the software has found the oscilloscope.



Click on USBFound and then on the displayed device in the window that opens. The oscilloscope can also be connected to the PC via the LAN port. To connect via LAN, go to point **22. Using the LAN port.**

4.1 Conduct the General Examination


It is recommended that after receiving a new oscilloscope, a check of the instrument is carried out as follows:

1. Check whether the unit has been damaged during transport.
If you find that the cardboard packaging or the protective foam pads are badly damaged, save them until the entire unit and its accessories have passed the electrical and mechanical test.
2. Checking the accessories
Check probes and other accessories for possible damage or bare or bent cables and wires before putting them into operation. If in doubt, do not take any measurements and inform your dealer.
3. Checking the unit
If you notice any damage to the exterior of the unit or if the unit does not function properly or does not pass the performance test, please contact your dealer. If the unit has been damaged during transport, please keep the outer packaging.

4.2 Performing the functional test

Check the proper functioning of the meter as follows:


Do a quick functional test to check the normal operation of the unit. First connect the oscilloscope to the PC, then carry out the following steps:

1. In the software interface, click  to restore the factory settings. The default setting of the damping coefficient of the probe in the software is 10X.

2. Set the oscilloscope probe switch to 10X and connect the probe to CH1 channel. Connect the probe tip and earth terminal to the probe compensation connector on the oscilloscope.
3. Click on **A** at the top right of the software interface to start AUTO set mode. The square waveform of 1 kHz and 3.3VSS is displayed.
4. Check channel 2 by repeating steps 2 and step 3.

4.3 Performing the probe compensation

When you connect the probe to an input channel for the first time, you must adapt the probe to the input channel. An uncompensated or incorrectly compensated probe will result in measurement errors. Perform the probe compensation as follows:

Click on  (main menu) in the software interface. Select **channel** and set the **divider ratio to x10**. Set the switch on the probe to 10x and connect the probe to channel 1 of the oscilloscope. If you use a probe with a gripper tip, make sure that the gripper tip keeps close contact with the probe.

Connect the probe tip to the signal output of the probe compensator and connect the reference terminal to the ground terminal of the probe compensator, then click **A** (Auto-Set) on the top right of the software interface.

Check the displayed waveforms and adjust the probe until correct compensation is achieved (see **fig. 8** and **9**).

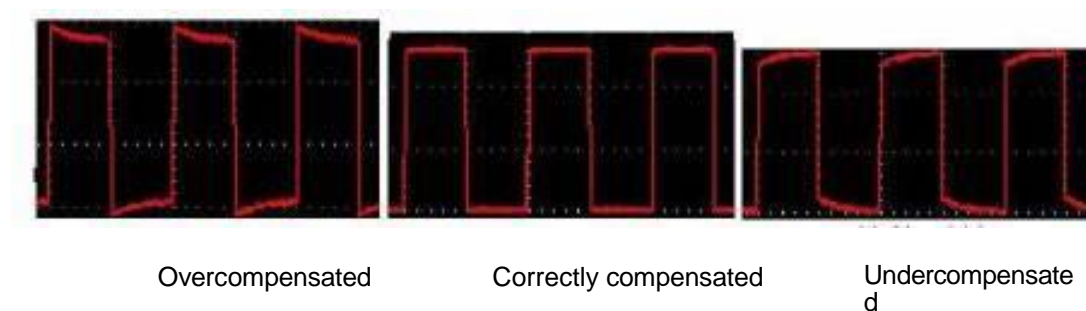


Fig. 8 Waveform representation of the probe compensation

Repeat the steps if necessary.

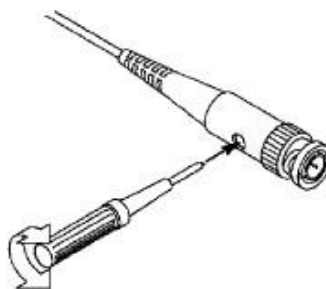


Fig. 9 Scanning head adjustment

4.4 Setting the probe damping factor

The probe has several probe damping factors that affect the vertical scaling factor of the oscilloscope.

If the set probe attenuation factor is to be changed or checked, press the function menu key for the respective channel and then the selection key corresponding to the probe until the correct value is displayed.

This setting remains valid until it is changed again.



Note: The damping factor of the probe in the menu is preset to 10X at the factory.

Make sure that the value set on the attenuation switch of the probe corresponds to the attenuation value set on the oscilloscope.

The values that can be set with the switch on the probe are 1 X and 10X (see Fig. 10).



Fig. 10 Damping switch



Note: When the attenuator switch is set to 1X, the probe limits the bandwidth of the oscilloscope to 5 MHz. You must set the switch to 10X if you want to use the entire bandwidth of the oscilloscope.



Warning:

To avoid electric shock, always keep your fingers behind the protective safety ring of the probe during measurement.

To protect yourself from electric shock, do not touch any metal parts of the probe tip when it is connected to the power supply.

Before taking any measurements, always connect the probe to the oscilloscope and connect the ground terminal to the earth potential.

5. General care and cleaning

General maintenance

To avoid damage to the instrument or probes, do not expose it to all sprays, liquids or solvents.

Cleaning

Check the unit and the probes as often as the operating conditions require. To clean the exterior of the unit, carry out the following steps:

1. Wipe off dust on the unit and the probes with a soft cloth.
2. Disconnect the USB cable before cleaning the oscilloscope. Clean the unit with a damp soft cloth (no dripping water). It is recommended to wipe the instrument with mild detergent or fresh water. To avoid damage to the instrument or the probes, do not use aggressive chemical cleaning agents.



Warning:

Before putting the unit back into operation, it is necessary to make sure that the unit is completely dry. This is the only way to avoid any electrical short-circuit or physical injury from the moisture.

6. PeakTech® Oscilloscope Software Help

Minimum system requirements

Processor: Pentium(R) 4/ 2.4 GHz

Memory: 1 GB Hard disk space: minimum
1 GB

Recommended system requirements

Processor: Pentium(R) Dual-Core 2.4
GHz Memory: 2 GB Hard disk space:
minimum 2 GB

Further requirements

Operating system: Windows XP to Windows 10

Ports: USB 2.0 or LAN

Resolution: 1024 x 768

PC software

Use the supplied USB cable to connect the oscilloscope to a PC and its USB ports.


Note: If you use a USB cable that is not supplied by us, you may experience some problems such as connection errors and signal interference.

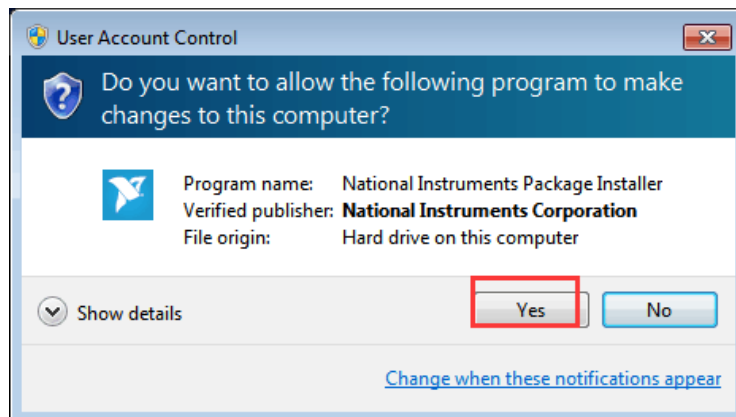
NI - VISA driver installation

To operate the PC oscilloscope with the PC, the driver installation of NI-VISA version 19.5 (or higher) is required. If a working NI-VISA is already installed on the PC, the following steps can be neglected.

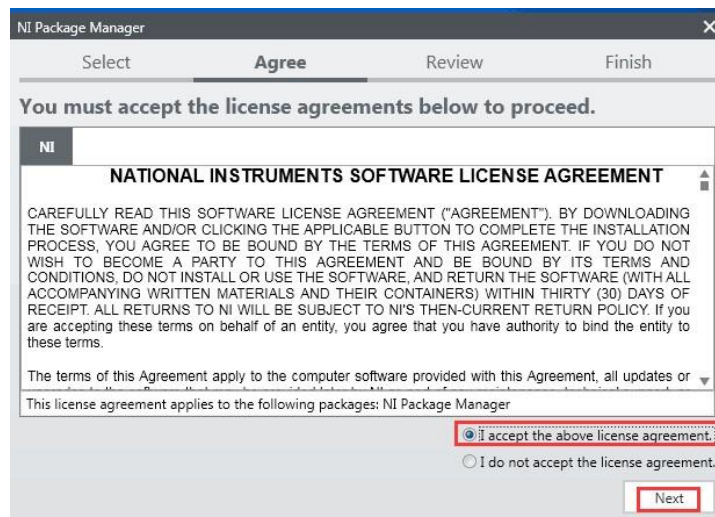
Download the NI-VISA 19.5 online repack file from the CD supplied to your PC and start the driver installation. If there is no Internet connection on the PC to be used, download the offline driver installation version 19.5 from the NI website, save the approx. 1 GB download pack to a USB stick and connect this to the PC to be used in order to carry out the driver installation there.

The installation procedure is as follows:

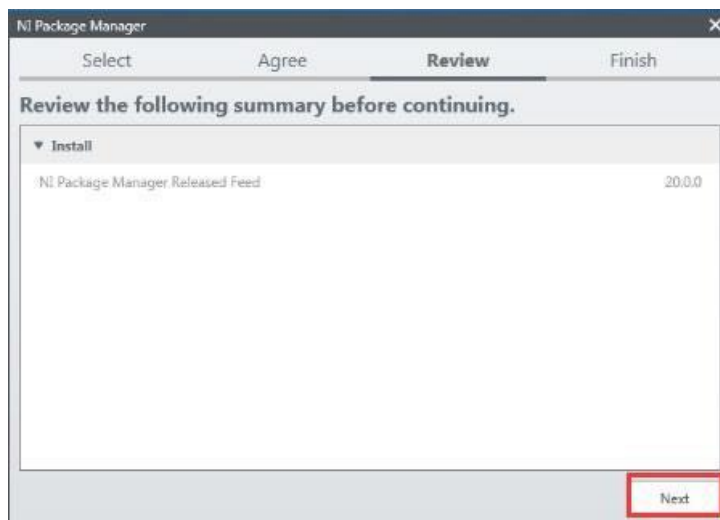
Start the installation file of the driver.  ni-visa_19.5_online_repack.exe



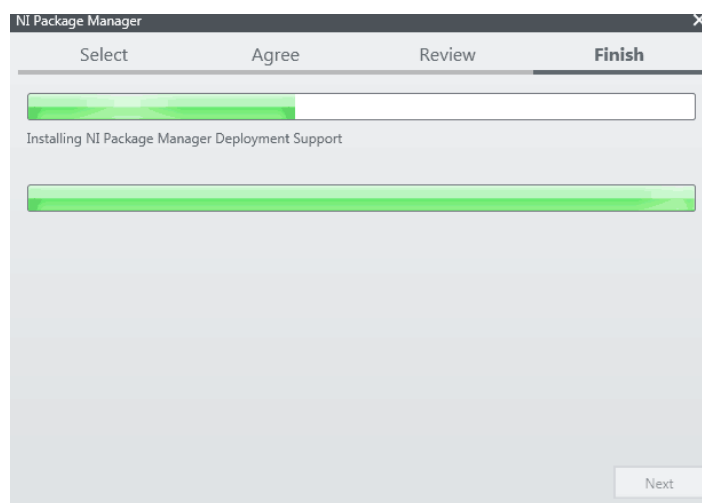
Click Yes here to start the installation.



In this step, accept the general terms and conditions by selecting the upper field and confirming with Next.



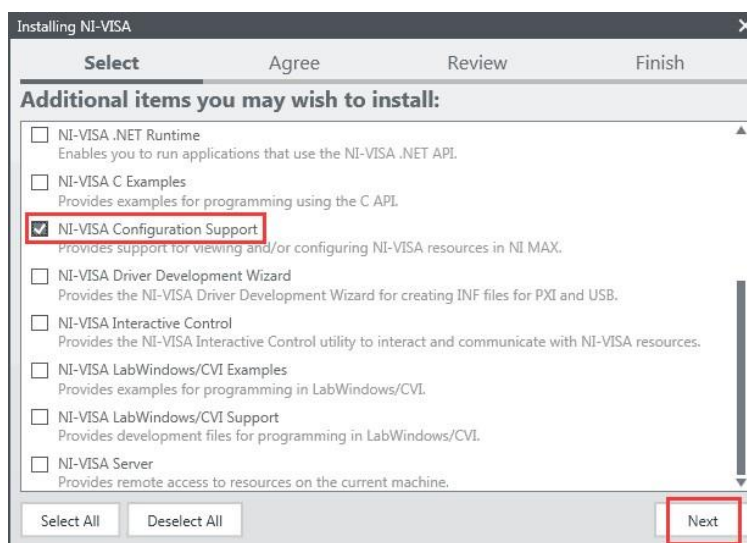
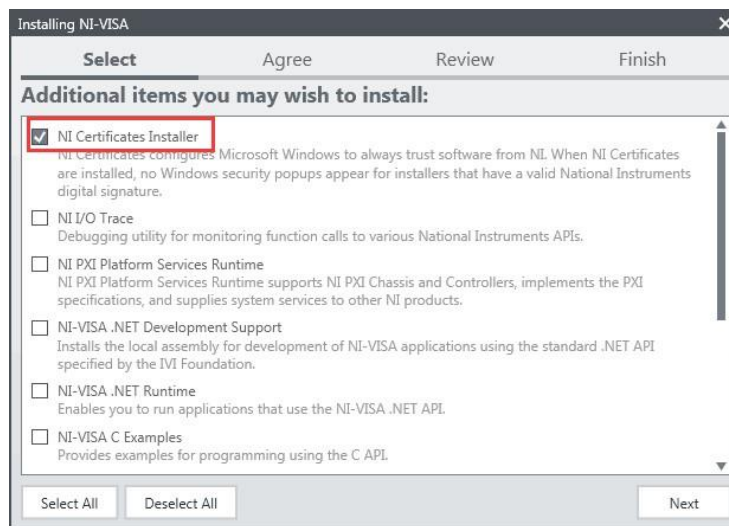
Click Next again in this window to carry out the installation.



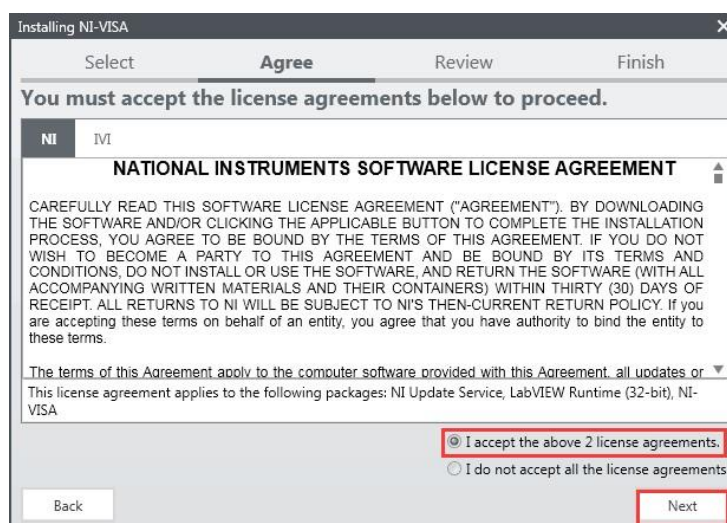
Now wait until the installation is complete.

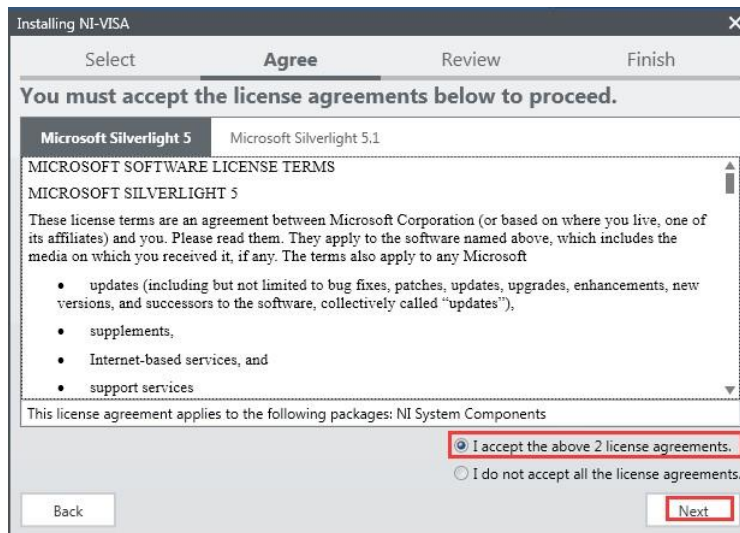


Next, the NI Package Manager opens, where you must select the required functions.

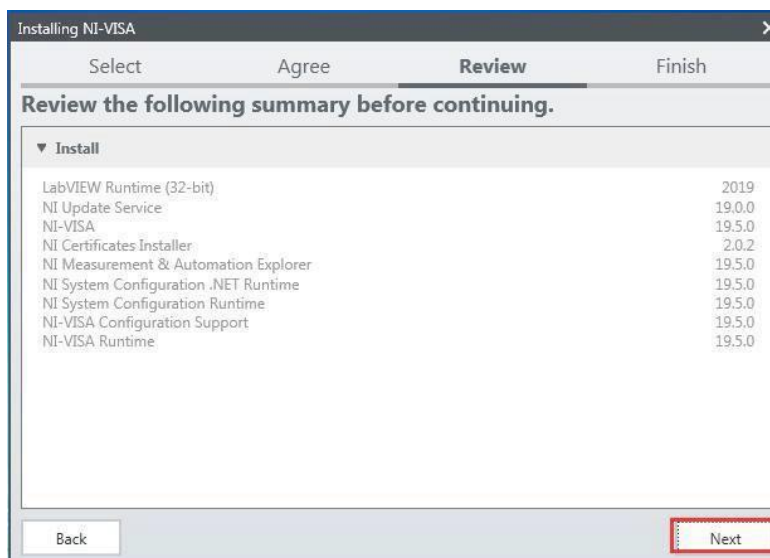


In this step, select only the two items NI Certificate Installer and NI-VISA Configuration Support.

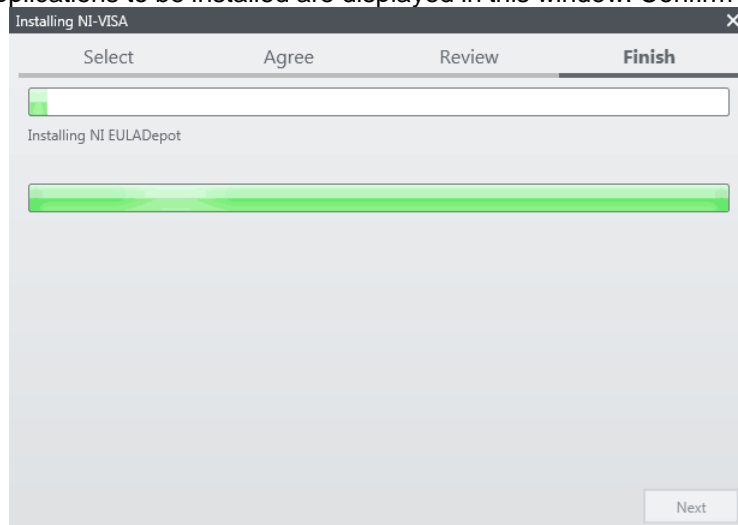




Confirm the terms and conditions again (this window may appear up to 2 times because the Certificate Installer and the Configuration Support have been selected).



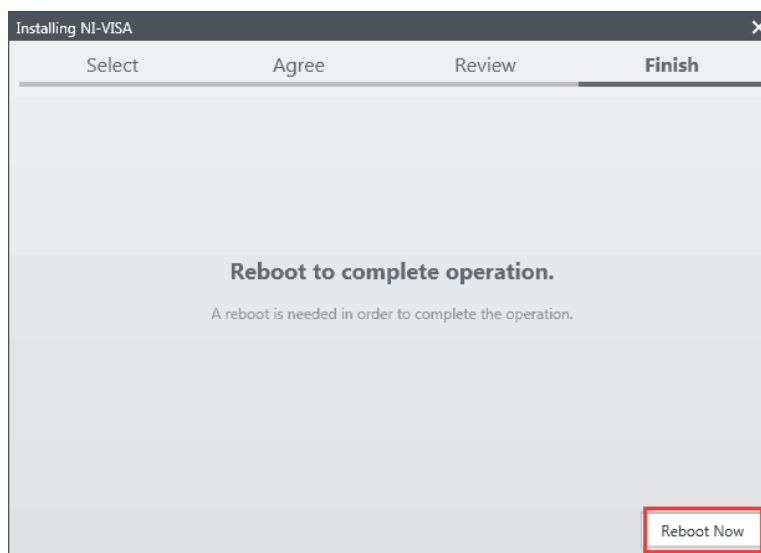
The applications to be installed are displayed in this window. Confirm with Next.



Now the last selected functions are installed. When this installation is finished, confirm again with Next.

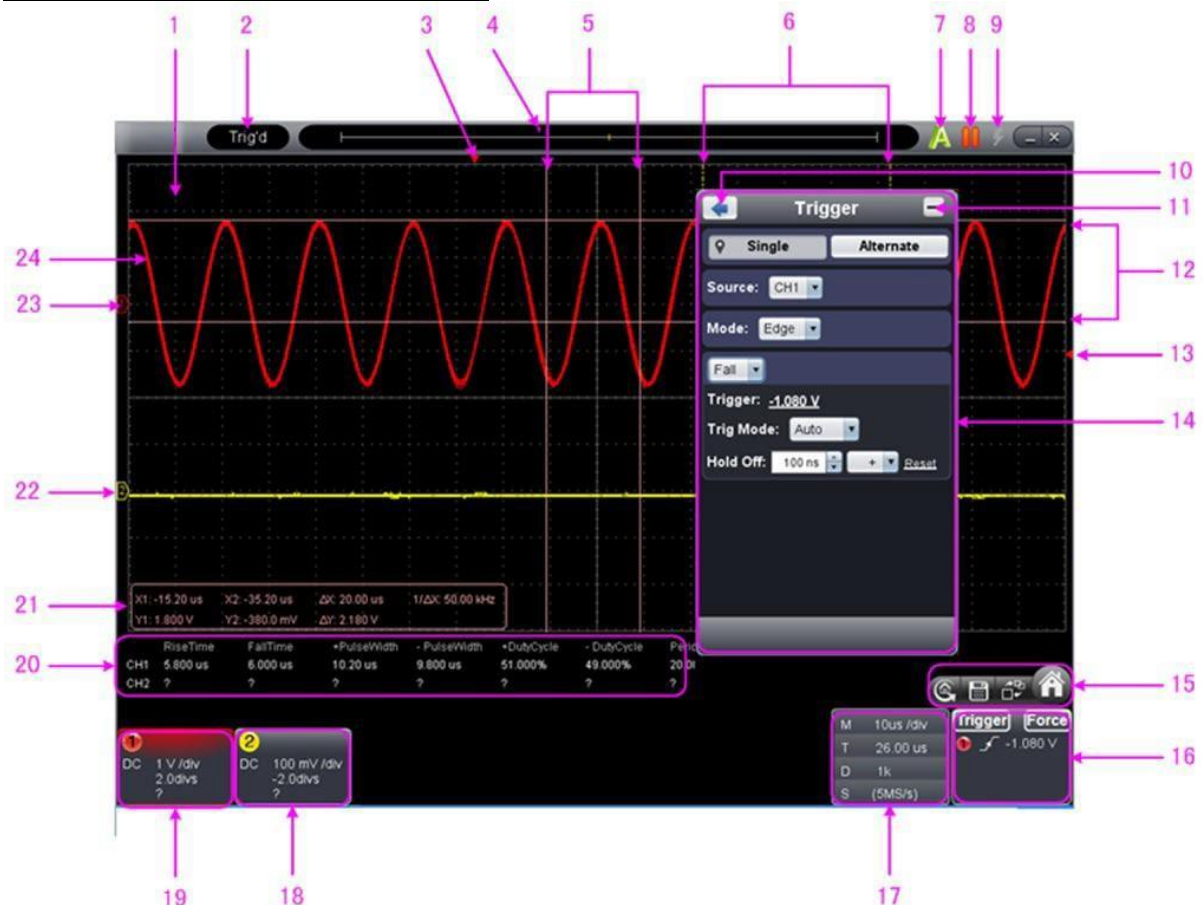




After the installation is complete, this window appears, which allows National Instruments to obtain general information about the operation and for error predictions in the event of a fault. You can answer no to this as it will not affect the operation of the driver.




Finally, the request to restart the system (the PC) appears.


7. Introduction to the user interface



1. Waveform Display range
2. Display status, click to "Disconnect", "Install USB Driver" or "Connect LAN".
3. The red mark shows the horizontal position of the trigger.
4. The mark shows the trigger position of the internal memory.
5. Time measurement with the help of the cursor
6. The two yellow lines show the size of the extended display window
7. Auto set, note [21. Using executive buttons](#)
8. Run/Stop, note [21. Using executive buttons](#)
9. Single trigger, note [21. Using executive buttons](#)
10. Back to the main menu
11. Hide menu
12. Voltage measurement with the help of the cursor
13. The red marking shows the position of the trigger level for channel 1 (yellow for channel 2). It can be moved up and down.
14. Function menu: Click  to hide the menu
15.  Shortcut key for resetting to factory settings

 Shortcut key for exporting waveforms

 Switch between the three view / one view mode. In Three View display mode, the upper left window becomes XP mode, the upper right FFT window is

 Showing and hiding the main menu

16. Trigger window, see [8.4. Setting the trigger system](#)
17. Sample - and period window, see [8.3. Setting the horizontal system](#)
- 18./19. Display window of channels 1 and 2, see [8.2. Setting the vertical system](#)

20. Display of the measured type and the value of the corresponding channel, see [10. To perform an automatic measurement](#)
21. Cursor measurement window, see [12. Measurements with the cursor](#)
- 22./The yellow line shows the reference point to ground (zero position) of the CH2 waveform. If the line is not shown, it means that this channel is switched off. (Red line is for CH1).
24. the waveform of CH1

Keyboard shortcuts

F5:	Run/Stop
Ctrl + Enter:	Autoset
Q:	The voltage division of channel 1 is reduced by one step.
A:	The voltage division of channel 1 increases by one step.
W:	The voltage division of channel 2 decreases by one step.
S:	The voltage division of channel 2 increases by one step.
E:	The voltage division of channel 3 decreases by one step.
D:	The voltage division of channel 3 increases by one step.
R:	The voltage division of channel 4 decreases by one step (only P 1326 and P 1331).
F:	The voltage division of channel 4 increases by one step.
←:	Time base decreases by one step
→:	Time base increases by one step
F1:	Open the help

Status information guide

Auto	Automatic trigger mode
Ready	Ready for a trigger
TRIG'D	Has triggered
Scan	Slow Scan
Stop	Data acquisition stopped
Error	An error has occurred
ReSyncing	Synchronise with the oscilloscope again
AutoSet	In Process of self-registration

The status when the oscilloscope is not connected:

Offline	No USB connection with the oscilloscope.
USBFound	Available USB devices found
USBDrvEr	Error during USB driver installation
MachineNotSupport	Cannot recognise device

The status when the oscilloscope is connected:



Linking	Connection with oscilloscope active
Connect	Connection successful
Match	Matching to oscilloscope
Syncing	Synchronise settings

8. Operation

8.1 Setting the probe damping factor

The probe has several probe damping factors that affect the vertical scaling factor of the oscilloscope.

In the oscilloscope menu you can change or check the probe damping coefficient:

- (1) Click on  to access the main menu and select "Channel".
- (2) Set ProbeRate"  to the correct value according to the probe.

This setting is valid until it is changed again.



Hint:

The damping factor of the probe in the menu is preset to 10X at the factory.

Make sure that the value set on the attenuation switch of the probe corresponds to the attenuation value set on the oscilloscope.



Fig. 10 Damping switch



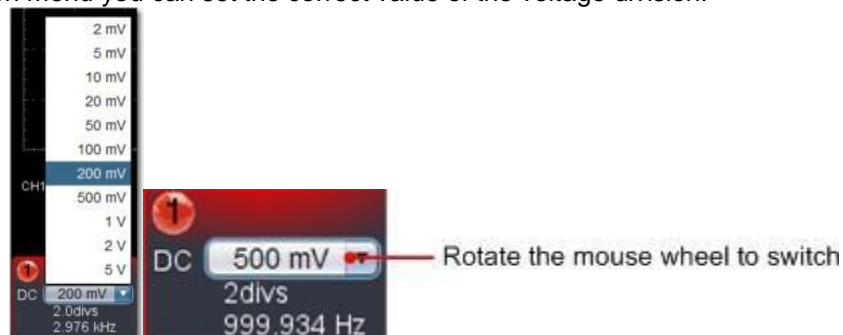
Note: When the attenuator switch is set to 1X, the probe limits the bandwidth of the oscilloscope to 5 MHz. You must set the switch to 10X if you want to use the entire bandwidth of the oscilloscope.

8.2 Adjustment of the vertical system

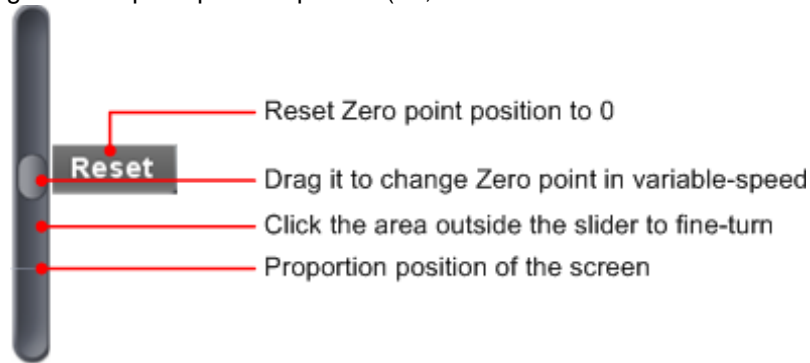
You can set the corresponding parameters of the vertical system in the channel menu (18/19 in [6.Introduction to the user interface](#)).



In the drop-down menu you can set the correct value of the voltage division.



You can adjust the zero position by using the control bar on the vertical display position of the signal. You can also drag the zero point position pointer (22, 23 in [6. Introduction to the user interface](#)).



Drag the slider up to increase, down to decrease, the further from the centre, the higher the change in speed.

Keyboard - Shortcuts

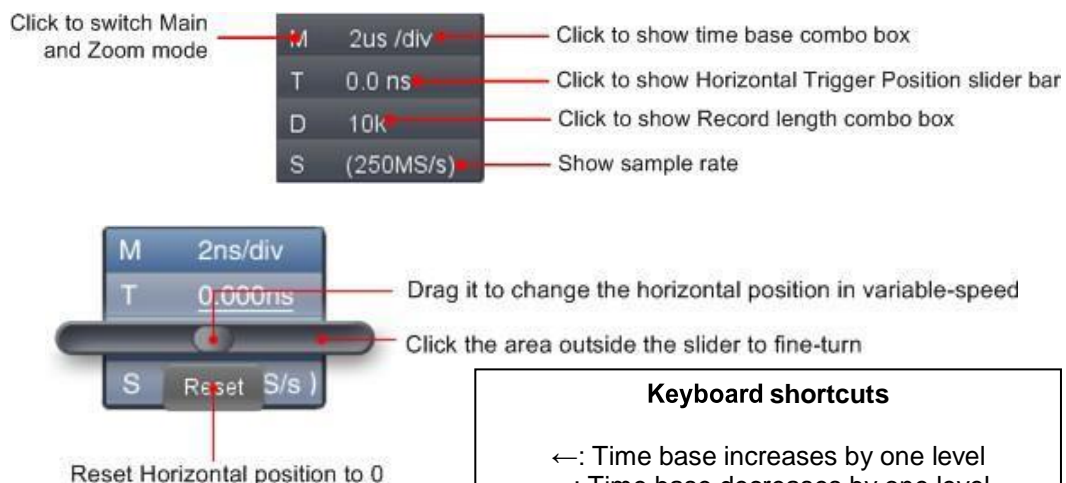
Q:	The voltage division of channel 1 decreases by one level
A:	The voltage division of channel 1 increases by one level
W:	The voltage division of channel 2 decreases by one level
S:	The voltage division of channel 2 increases by one level
E:	The voltage division of channel 3 decreases by one step (only P 1326 and P 1331)
D:	The voltage division of channel 3 increases by one step (only P 1326 and P 1331)
R:	The voltage division of channel 4 decreases by one step (only P 1326 and P 1331)
F:	The voltage division of channel 4 increases by one step (only P 1326 and P 1331)

Cymometer

This is a 6-digit Cymometer. The Cymometer can measure frequencies from 2Hz to full bandwidth. However, it can only measure the frequency accurately if the measured channel has a trigger signal and is in **edge mode**. In **single trigger mode**, it is a single-channel Cymometer and can only measure the frequency of the triggered channel. In **ALT trigger mode**, it is a two-channel cymal and can measure the frequency of two channels.

8.3 Adjustment of the horizontal system

You can set the corresponding parameters of the horizontal system in the acquisition and time base window (17 in [8. Introduction to the user interface](#)).




See also [19. How to enlarge the waveform \(zooming\)](#)

8.4 Setting the trigger system

The trigger determines when the software starts to acquire the measurement data and display the waveform. Once the trigger is set correctly, an unstable display can be converted into a meaningful waveform.

When the software starts collecting measurement data and has collected enough data, a waveform is displayed to the left of the trigger point.

The data acquisition also continues to work during the waiting time until the next trigger point. Once a trigger point is detected, there is enough continuous data to display the waveform on the right side of the trigger point.

Click on  to access the main menu and select "Trigger".

You can also click on the  button to enter the trigger menu.



8.5 Trigger control

The oscilloscope offers two trigger types: Single trigger and alternate trigger. Single trigger: Use one trigger level to capture stable waveforms with two channels simultaneously.

Alternate trigger: Trigger on non-synchronised signals.

The Single Trigger and Alternate Trigger menus are described below:

Single trigger

1. In the trigger menu, select the option "Single" (the selected function is highlighted).
2. Select source.
3. Select mode.

Single trigger has four modes:

Edge trigger, video trigger, slope trigger and pulse trigger.

Flank Trigger: Occurs when the trigger input through a voltage level with the specified edge.

Video trigger: Trigger on video fields or video lines for a standard video signal.

Slope trigger: The oscilloscope starts triggering according to the rise or fall speed of the signal.

Pulse trigger: Occurs with pulses of certain widths.

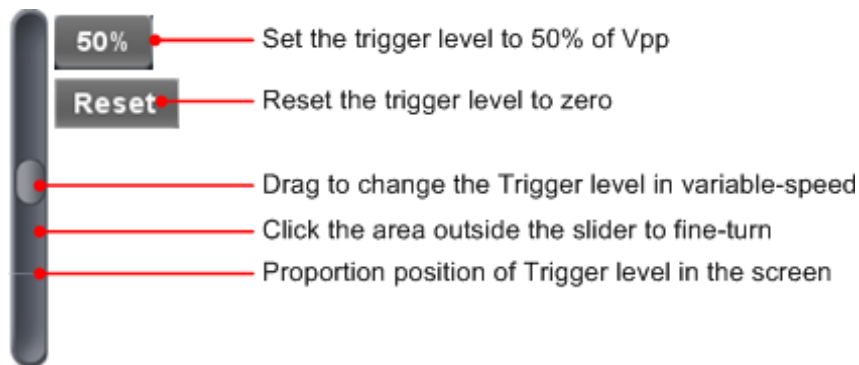
The four trigger modes in the single trigger menu in detail:

Edge

An edge trigger occurs at the trigger threshold of the input signal. Select the edge trigger mode to trigger on the rising or falling edge of the signal.

Settings in the Trigger menu:

1. Select "Rise" to trigger the rising edge Select "Fall" to trigger the falling edge
2. Click on the voltage value to open a slider with which you can set the trigger value. (See also [7. Introduction to the user interface.](#))



3. Select Trig Mode:
Auto: Acquisition of waveforms even if no trigger fires. Normal:
Acquisition of waveforms when the trigger fires.
Single: When the trigger fires, waveform capture stops.
- 4 SET Hold Off:
When the trigger fires, stop capturing the waveform (100 ns ~ 10 s).

Setting method:

"+", "++", "+++" is the position of the digit that will be changed.

"+" Stands for the last digit

"++" the middle digit

"+++" stands for the first digit.

If "+" is selected, click ▲, thus the last digit is increased by 1.

Click "Reset" to reset the "Holdoff Time" to the default value (100ns)

Settings in the trigger window:



Force: Press this button to create a trigger signal. This function is primarily used with the trigger modes "Normal" and "Single".

Video trigger

Select Mode as "Video" to trigger on video fields or video lines of NTSC, PAL or SECAM standard video signals.

1. Select video modulation: NTSC, PAL or SECAM.
2. Set the synchronous trigger:
Line (video line), Field (video field), Odd (odd video field), Even (even video field) or Line Number (in created video line).
When "Line Number" is selected, the line number can be set from 1 - 525 with the Up/Down buttons.
3. SET Hold Off. This setting allows waveforms to be captured even when no trigger occurs.

Slope trigger

Slope mode lets the oscilloscope trigger on the rising/falling edge of a signal within a specified time period.

1. Setting the SLOPE conditions.
2. Setting the slope time.
3. Setting the high level and the low level.
4. Setting the trigger mode and the HOLD-OFF.
5. Setting the Slew rate = (High Level - Low Level) / Settings

Pulse trigger


A pulse trigger occurs according to the pulse width. Deviating signals can be detected by adjusting the pulse width condition.

1. Setting the pulse width conditions and the pulse duration.
2. Setting the trigger level
3. Setting the trigger mode and HOLD-OFF.

Alternating Trigger (ALT)

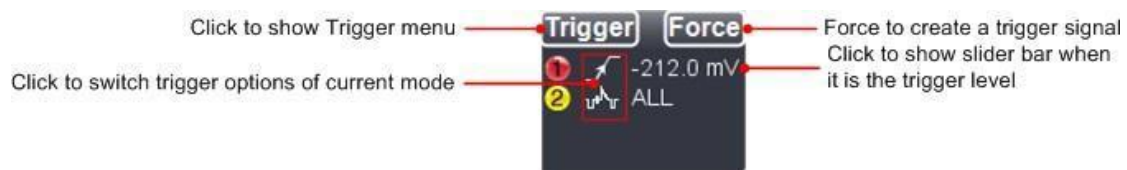
In alternating trigger mode, the trigger signal comes from two vertical channels. This mode is used to observe two independent signals. You can select different trigger modes for different channels. The following options are available: Edge, Video, Pulse or Slope.

Settings in the Trigger menu:











1. In the trigger menu, select the option "Alternate" (the selected button is marked ).
2. Select the trigger source.
3. Select trigger mode.

Note: In alternating trigger, only one channel is displayed in video mode at the highest. You cannot use video mode on both channels at the same time.








Settings in the trigger window:



Introduction of the symbols in the trigger window

	Rising edge		Falling edge
	Synchronous trigger in video line		Synchronous trigger in video field
	Synchronous trigger in odd video		Synchronous trigger in the straight video field
	Rising in Slope		Falling in Slope
	+ PulseWidth		-PulseWidth

9. Input channel settings

Click on  to open the main menu and select "Channel". You can also click  or  in the lower left corner of the window or  in 4-channel mode (only P1325 /1330)



Switching a channel on and off

Click "CH1" or "CH2" and place a tick in front of "ON" to switch the channel on. Click to remove the tick to switch the channel off.

Inverting a waveform

Invert waveform means to rotate the displayed signal by 180° with respect to the earth potential. Click "CH1" or "CH2" and place a tick in front of "Invert" to invert the signal or remove the tick to display the waveform in the normal state.

Set channel coupling

DC: Allows AC and DC components, in the input signal, to pass. AC: Blocks the DC component in the input signal. Ground: Input signal is interrupted

9.1 Adjusting the probe damping factor



For correct measurement results, the settings of the damping factor in the operating menu of the channel should always correspond to those of the probe (see Performing probe compensation). If the damping factor of the probe is 1:1, the setting for the input channel should also be X1.

Some functions can be selected in Window Channel:



10. To perform automatic measurements

Click  to display the main menu and select "Measurements".

The  "Automatic Measurements" function is located on this button. A total of twenty different types of measurements can be selected. A maximum of 8 types of measurements can be displayed simultaneously on the lower left corner of the screen. The oscilloscope provides 20 parameters for automatic measurement, including Vpp, Vmax, Vmin, Vtop, Vbase, Vamp, Vavg, Vrms, Overshoot, Preshoot, Freq, Period,  RiseTime, Falltime, Delay A→B, Delay A→B, +Width, -Width, +Duty

Show all:

Select the channel next to "Show all" and all possible measured values will be displayed.

10.1 Adding an automatic measurement:

Check the selected channel and the type of measurements.

The measurement results are displayed in the lower left corner of the window. You can add a maximum of 8 measurement types for each channel. When 8 measurement types are exceeded, the first selection is removed again.

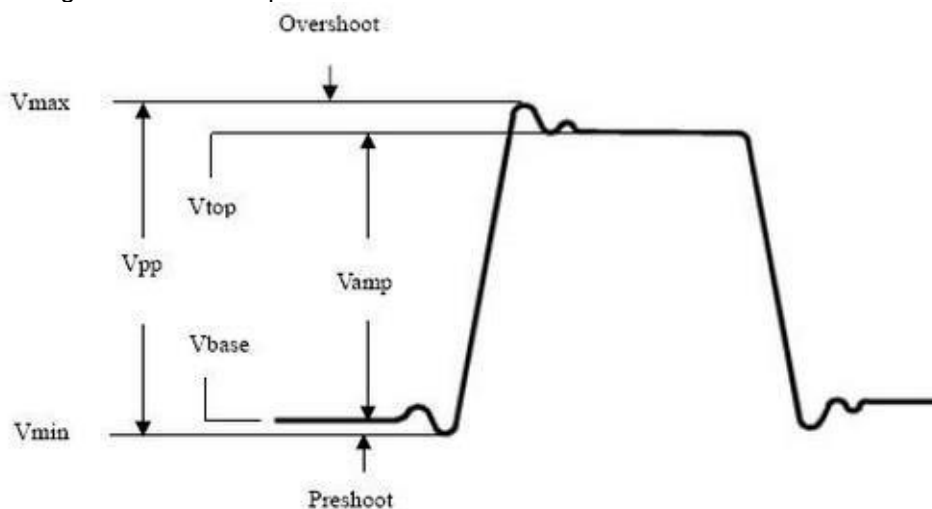
The measured values of the two channels can be displayed simultaneously.

10.2 Removing an automatic measurement:

Remove the tick in front of the type of measurement you want to remove. Click on "Remove all" to remove all measurements.

10.3 The automatic measurement of voltage parameters:

The **PeakTech**[®] oscilloscope provides automatic voltage measurements including Vpp, Vmax, Vmin, Vavg, Vamp, Vrms, Vtop, Vbase, Overshoot and Preshoot. The illustration below shows a pulse with some of the voltage measurement points.

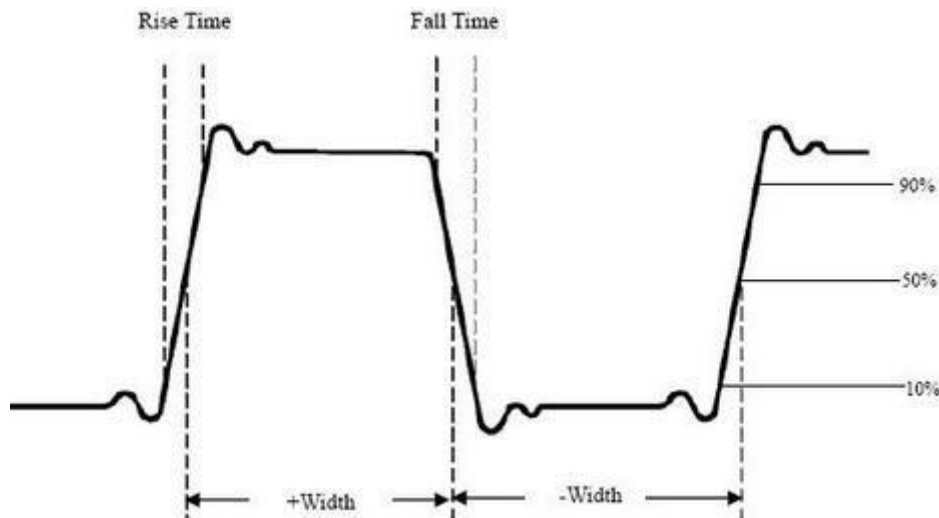


Vpp:	Peak - peak tension.
Vmax:	Maximum amplitude. The highest positive peak voltage measured over the entire curve.
Vmin:	Minimum amplitude. The highest negative peak voltage measured over the entire curve.
Vamp:	Voltage between Vtop and Vbase of a curve.
Vtop:	Flat-top voltage of the curve, useful for square/pulse signals.
	Vbase: Flat-base voltage of the curve, useful for square/pulse signals.
	Overshoot: (Overshoot) Defined as $(V_{max} - V_{top}) / V_{amp}$, useful for square wave and pulse signals.
Preshoot:	Defined as $(V_{min} - V_{base}) / V_{amp}$, useful for square and pulse signals.
Average:	The arithmetic mean over the entire curve.
Vrms:	The true rms voltage over the entire curve.

10.4 Automatic measurement of the time parameters

The oscilloscope provides automatic measurements of time parameters including Frequency, Period, Rise Time, Fall Time, +Width, -Width, Delay 1→2, Delay 1→2, +Duty and -Duty.

The illustration shows a pulse with some time measurement points.



Rise Time: The time taken for the leading edge of the first pulse in the curve to rise from 10% to 90% of its amplitude.

Fall Time: (Fall Time) The time taken for the leading edge of the first pulse in the curve to fall from 90% to 10% of its amplitude.

+Width: The width of the first positive pulse at the 50% amplitude point.

-Width: The width of the first negative pulse at the 50% amplitude point. Delay


1→2: The delay between the two channels at the rising edge. Delay 1→2:

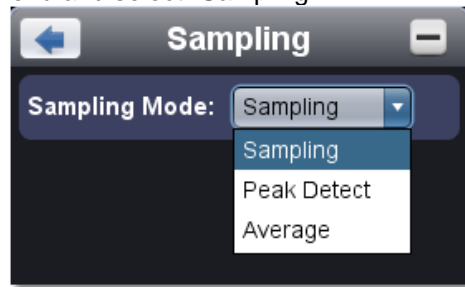
The delay between the two channels at the falling edge.

+Duty: +Duty ratio, defined as +Width/Period.

-Duty: - duty cycle, defined as -width/period.

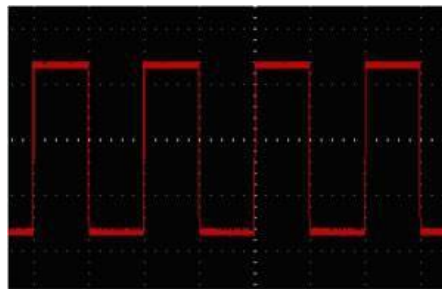
11. Setting up the scanning function

Click  to display the main menu and select "Sampling".

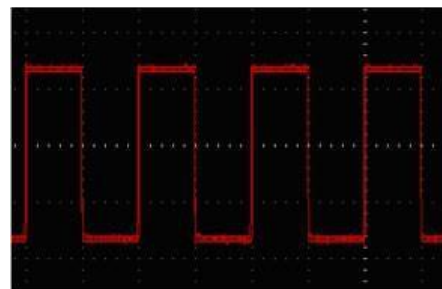


Sampling mode	Description
Sampling	Normal sampling mode
Peak value	To detect maximum and minimum samples. The highest and lowest points of adjacent intervals are found. This function is used to detect the degree of interference and the possibility of reducing the interference.
Average	Used to reduce random noise of any kind with an optional number of averages. (1 - 128)

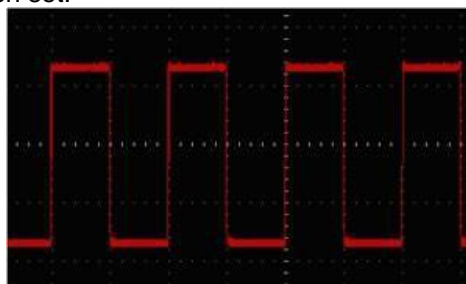
Normal sampling mode in which no burr could be detected.




Peak detection mode under which the burrs on the falling edge of the square wave signal and a strong noise were detected.



The displayed waveform after the noise has been removed using the average mode, where the average number of 16 has been set.



12. Measurements with the Cursor

Click  to display the main menu and select "Cursor".




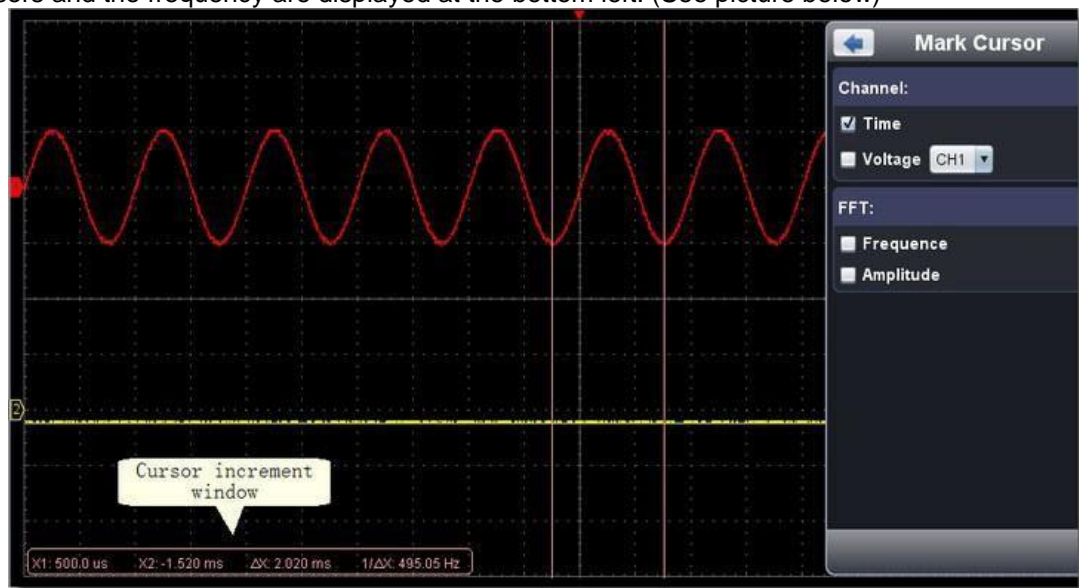
12.1 Normal mode

9. Select source:
Select the channel to be measured by the cursor. CH1 or CH2
10. Select the type of measurement:
Select either time measurement, voltage measurement or both.

12.2 Time measurement with the cursor:


Click on the "Time" option, two bright red lines will then appear along the vertical direction of the screen representing Cursor 1 and Cursor 2.

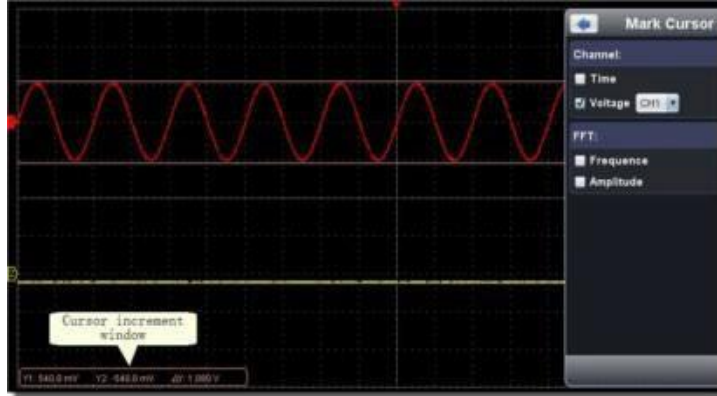
Place the mouse pointer over cursor 1 or cursor 2 and drag the mouse pointer to  after the change and set the positions of cursor 1 and cursor 2 based on the waveform to be measured. In the cursor measurement window, the current time of the two cursors, the absolute time difference of the two cursors and the frequency are displayed at the bottom left. (See picture below)



12.3 Voltage cursor measurement:

Click on the "Tension" option, two bright red lines will then be displayed along the horizontal direction of the screen, representing Cursor 1 and Cursor 2.

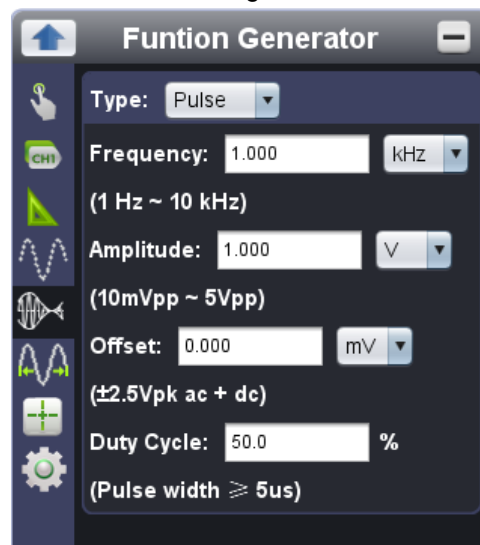
Place the mouse pointer over cursor 1 or cursor 2 and drag the mouse pointer to  after the change and set the position of cursor 1 and cursor 2 based on the waveform to be measured. In the cursor measurement window, the current position of the two cursors, the absolute voltage amplitude difference of the two cursors and the frequency are displayed at the bottom left. (See picture below)



13. Using the function generator (only P 1286)

The P 1286 has an integrated 5 MHz function generator. With this it is possible to generate 4 different waveforms. These are sine, square, ramp and pulse and are output through the Multi Channel.


To set the function generator, select the following function in the software:



In this window you can set the type of waveform, the level of the frequency to be output, the amplitude, the offset and the duty cycle.


Note: Symmetrical option is only available in the Ramp wave type; Duty Cycle option is only available in the Pulse wave type.

14. Setting the display system

Click  to display the main menu and select "Display".



Display type

Click on the button of the desired display type (the selected button gets a mark ).

Vector:

The space between the adjacent sample points in the display is filled with the vector shape.

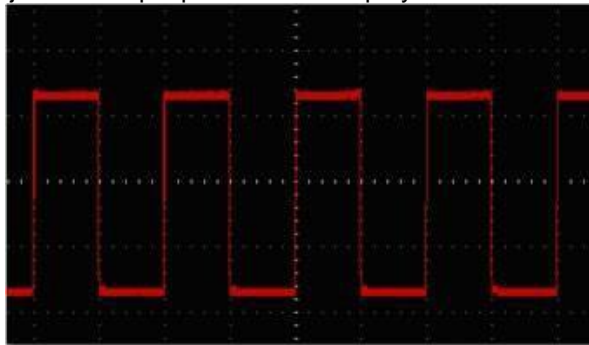


Fig. Display as vector form

Points:

Only the sampling points are displayed.

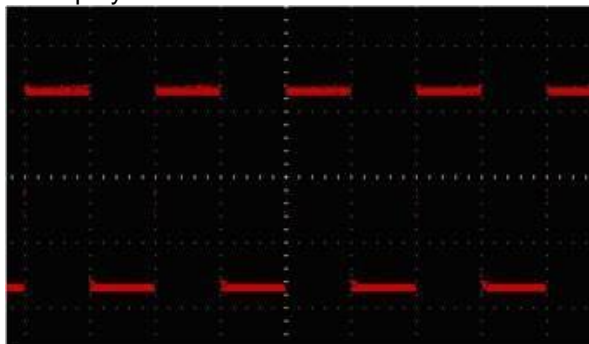


Fig. Display as point form

14.1 XY mode

If "XY mode" is selected, the user interface is switched to triple scale mode. Select the first channel and the second channel. In the XY window, the first channel is displayed in the horizontal axis and the second in the vertical axis.

Note: XY mode supports only 1K memory. The memory is automatically set with 1K.

14.2 Afterglow

With the Persist function, you can simulate the afterglow effect of a tube oscilloscope: the stored original data is faded, the new data is displayed in vivid colour.

Different afterglow times can be selected: Off, 0.5 second, 1 second, 2 seconds, 5 seconds and infinite.

If the "infinite" option is selected, the measuring points are saved until the value is changed. Click on the "Clear" button to clear the afterglow indicator.

Hint:

When the time base, voltage division, deep memory is changed, or the channel is turned off/on, the persistence will automatically be reset and recording of the waveform will be updated.

14.3 Raster Brightness


Drag the sliders to adjust the brightness of the grid in the waveform display area.

15. Application of the mathematics function

The mathematical measurement function shows the results of additions, multiplications, divisions and subtractions applied to channel 1 and channel 2 as well as the FFT operation of channel 1 and channel 2 respectively.

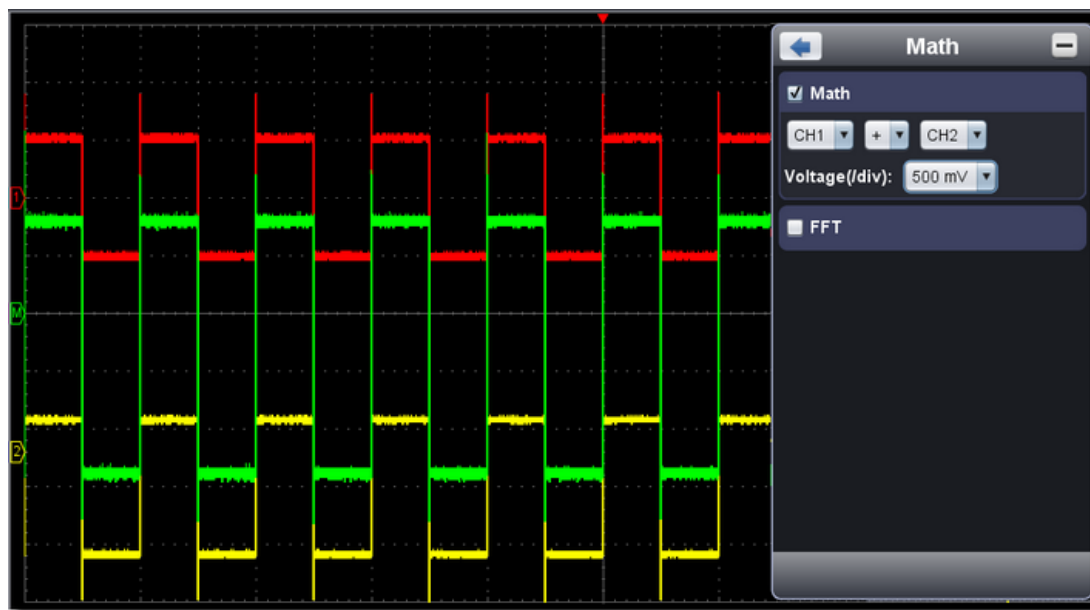


1. Switch on CH1 and CH2.

2. Click  to display the main menu and select "Maths Funct.".

3. Put a tick next to "Maths funct.".

Select the factors and the calculation function. Select the voltage division (M). The software converts the waveforms according to the two factors to calculate the selected voltage division (M). The green calculated curve (M) is displayed on the screen.




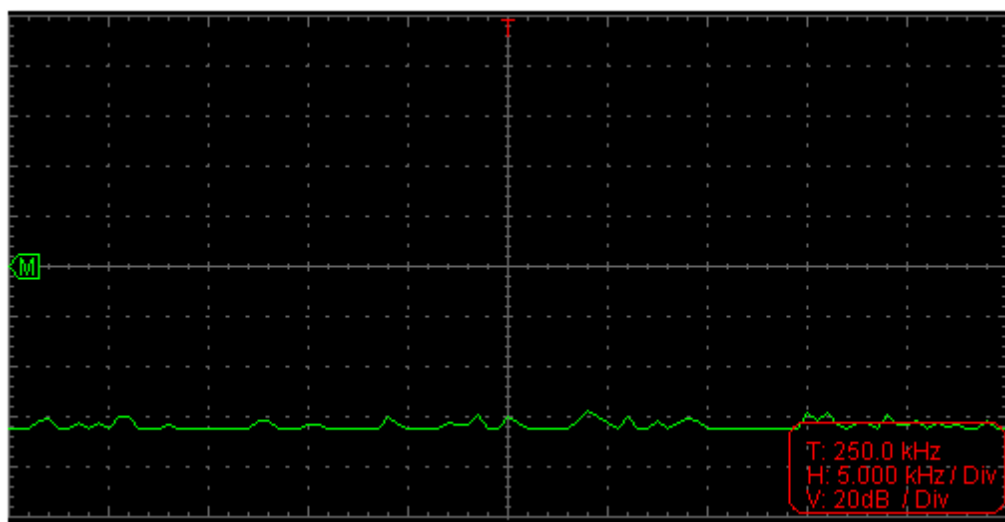
16. Using the FFT function

An FFT analysis converts a signal into its frequency components, which the oscilloscope uses to graphically display the frequency range of a signal in addition to the standard time range. You can compare these frequencies with known system frequencies such as system clocks, oscillators or power supplies.

The FFT function of this oscilloscope can convert 2048 points of the time domain signal into its frequency components. The final frequency contains 1024 points from 0Hz to the Nyquist frequency.



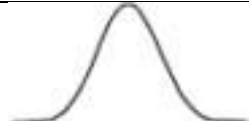

To use the FFT function, proceed as follows, for example:

1. Click  to display the main menu and select "Maths Funct.".
2. Place a tick in front of "FFT". The display window changes to the triple scale view.
3. Make the desired settings under "Channel", "Window", "Format", "Scale" and "Frequency Basis"
4. You can drag the signal curve anywhere in the FFT window to move the signal up, down and sideways.

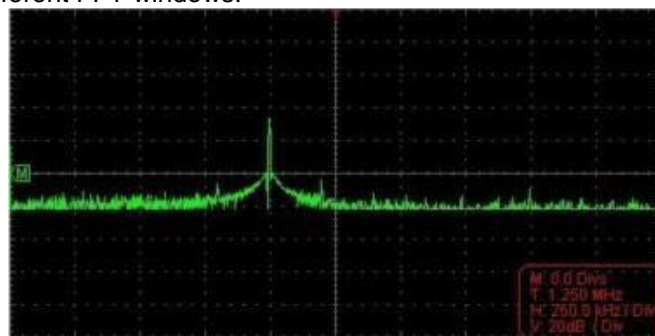


16.1 Selecting an FFT window

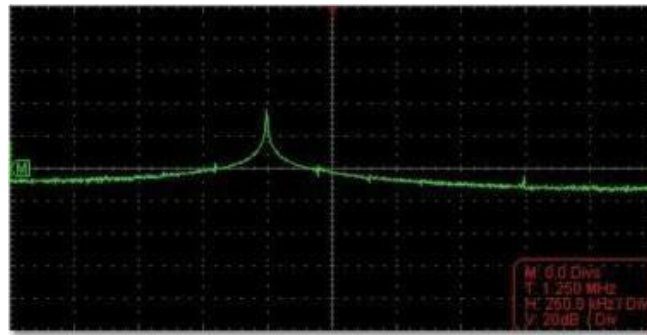
There are four FFT windows. Each window makes trade-offs between frequency resolution and amplitude accuracy. Choose the window based on what you want to measure and the characteristics of your source signal. The following table will help you choose the best window:

Type	Characteristics	Window
Hamming	This is a very good window for frequency resolution with slightly better amplitude accuracy than the rectangular window. It has a slightly better frequency resolution than the Hanning window. Use the Hamming window for measuring sine, periodic and narrowband noise. Best suited for transients or peaks, at where the signal levels before and after the event differ significantly.	
Rectangle	This window is best for frequency resolutions, but is the worst for accurately measuring the amplitude of these frequencies. It is the best window for measuring the frequency spectrum of non-repetitive signals and measuring frequency components near DC. Use the square wave window for measuring transients or peaks where the signal level before and after the event is almost the same. Also usable for sine waves with equal amplitude and with fixed frequencies as well as for broadband noise with a relatively slowly varying spectrum.	
Blackman	This is the best window for measuring the amplitude of frequencies, but offers the poorest frequency resolution. Use the Blackman-Harris window for single frequency signals and finding higher order harmonics.	
Hanning	This window is well suited for measuring amplitude accuracy, but less so for frequency resolutions. Use the Hanning window for measuring sine, periodic and narrowband noise. Best suited for transients or peaks, at where the signal levels before and after the event differ significantly.	

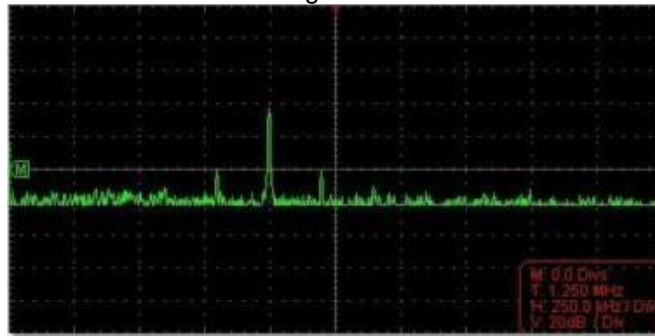
In the pictures below are examples of the measurement of a sine wave with a frequency of 1 kHz under the selection of four different FFT windows:



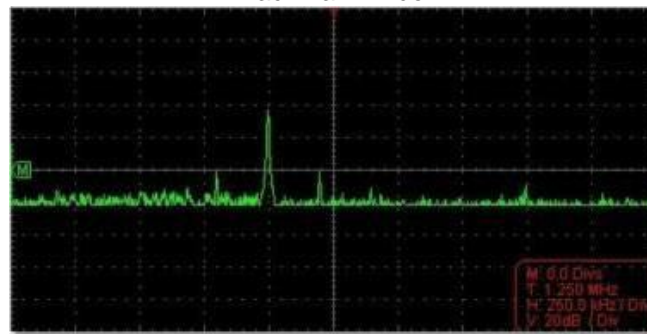
Hamming windows



Rectangle - Window



Blackman window



Hanning windows

Quick tips

- * If desired, use the zoom function to enlarge the FFT curve.
- * Use the dBV RMS scale for a detailed view of multiple frequencies, even if they have different amplitudes. Use the linear RMS scale to compare all frequencies in an overall view.
- * Signals that contain a DC component or offset can lead to incorrect FFT signal amplitude values. To minimise the DC component of the source signal, select AC coupling.
- * To reduce noise and aliasing in repetitive or single measurement waveforms, set the oscilloscope's acquisition mode to average.

Nyquist frequency:


The highest frequency that an oscilloscope that digitises in real time can measure is half the sampling rate and is called the Nyquist frequency. If not enough sampling points are acquired and the frequency is higher than the Nyquist frequency, the phenomenon of the "false waveform" occurs. Therefore, pay more attention to the relationship between the sampled and measured frequency.

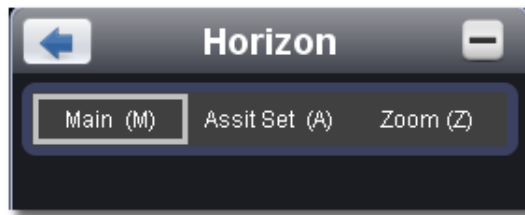
Hint:

In FFT mode, only settings for coupling, voltage division and zero setting can be made in the channel window; no other settings are possible.

If you want to apply other menus, first deactivate "FFT".

17. How to enlarge a waveform (zoom)

Click  to display the main menu and select "Horizon".



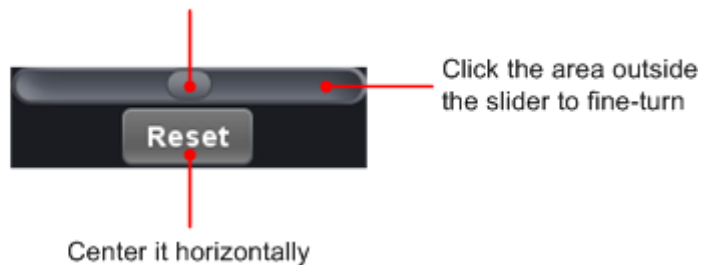
17.1 Main Time Base (Main)

The horizontal main time base setting is made to display the waveform.

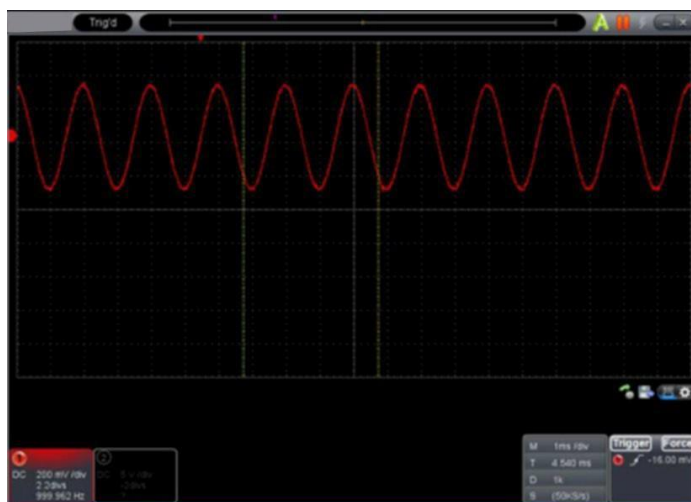
17.2 Define Window Area (Assist Set)

A window area is defined by two cursors, which expands to the full screen size. Select a value from the drop-down menu W to set the size of the window area. Click on the slider to move the set window area to the left or right.

Drag it to change the horizontal position in variable-speed

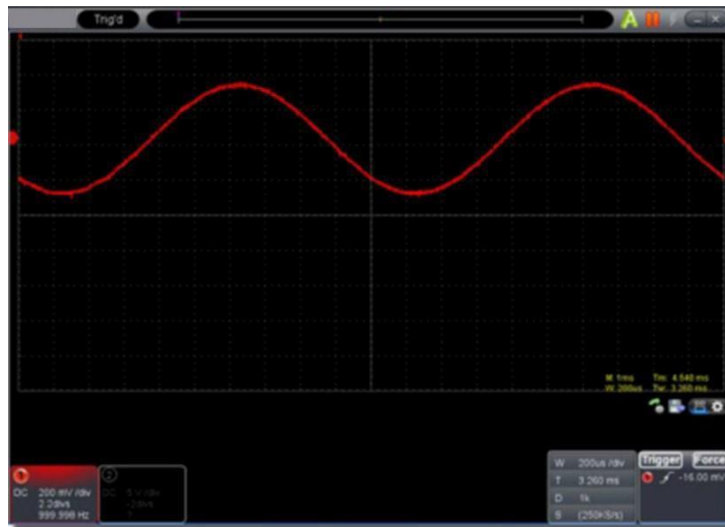


Note: In Assist Set mode, the time base and horizontal trigger cannot be set.

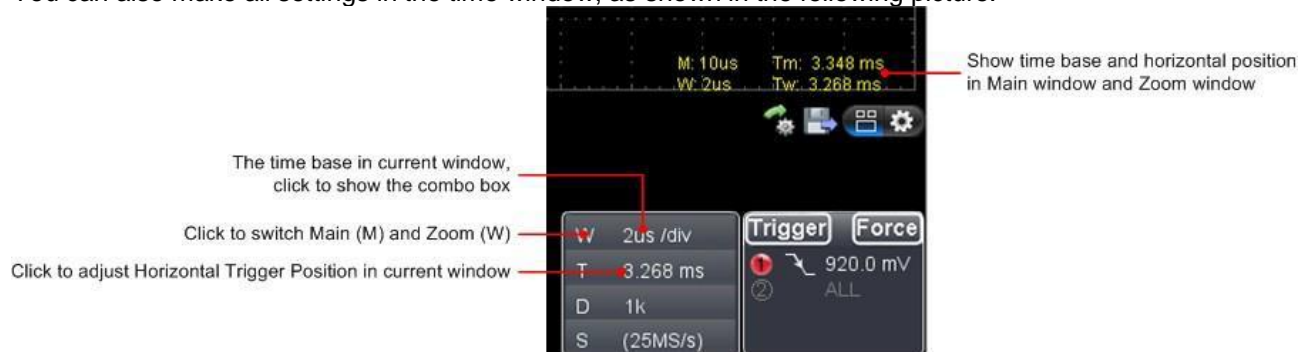


17.3 Window enlargement

Click on "Zoom", the window area defined by two cursors is expanded to the full screen size. You can adjust the time base W and the horizontal trigger position T in the Zoom menu. You can also adjust the red pointer on the horizontal trigger position ([6. Introduction to the user interface](#)).




You can also make all settings in the time window, as shown in the following picture:



18. Pass/Fail

The **pass/fail function** monitors deviations of signals and outputs pass/fail signals as a result of comparison with the input signal, which is in a predefined mask.


Click  to display the main menu and select "Pass/Fail".



18.1 Pass/Fail check

The pass/fail check detects whether the input signal is within the limits of the rule. If it exceeds the rule limits, it does not pass the check and is classified as a "fail"; if it is within the rule limits, it is allowed as a "pass". It can also output fail or pass signals via an integrated and configurable output port.

To perform a pass/fail check:

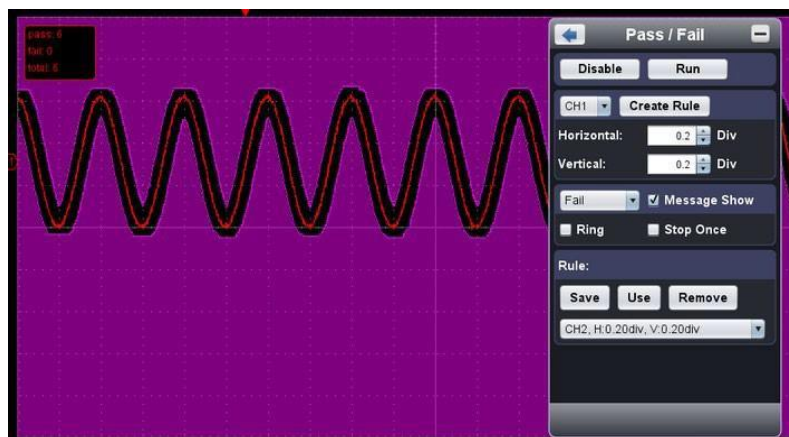
1. Select the input channel: CH 1, CH 2 or Math.
2. Set the horizontal and vertical tolerance. You can use the button  or enter the value directly.
3. Click on the "Create" button.
4. Set the output type: Select "Pass" or "Fail". Check the options "Ring", "Show Message" and "Stop Once" if needed.

Ring: A tone sounds when the created rule has been fulfilled.

Stop Once: Stops as soon as the created rule is fulfilled.

Message count: Notification of the count is displayed in the upper left corner of the screen.

5. Switch on: Click on "Activate".
6. Start test: Click "Go".
7. Stop test: Click "Stop".



18.2 Save created rule


Save: Saves the currently created rule.

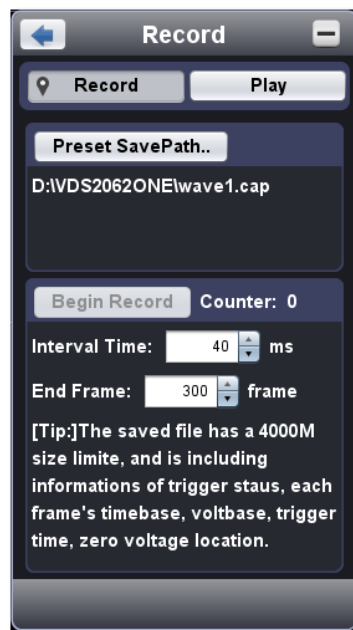
Apply: Select a created rule from the drop-down menu. Delete: Remove / delete a created rule.

19. Saving and loading waveforms

The waveform recording function can record input signal waveforms.

You can adjust the time interval between recorded images and get better examination results through the playback and storage function. The maximum value of a saved file size is 4000 M. The original trigger status, time base, voltage division, horizontal trigger position and zero position of each frame during recording are saved.

Click  to display the main menu and select "Recording".



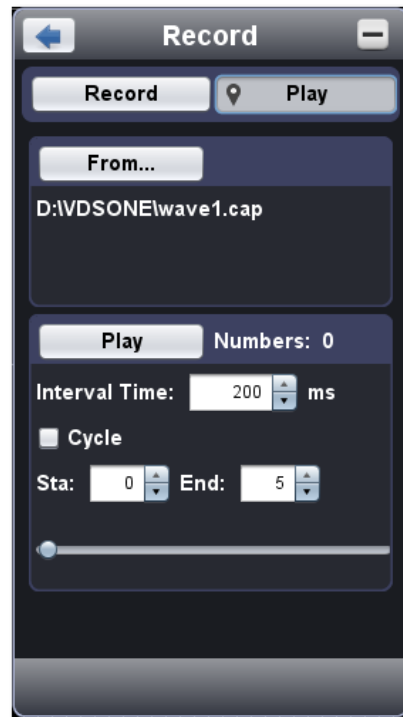
19.1 Recording

1. Select the "Record" button.
2. Click on "Save as..." to enter the storage location. Waveform files have the extension ". cap".
3. Set the "Interval Time" and "End Frame". The interval time refers to the distance between the recorded images. The interval time ranges from 0 to 100,000ms and in steps of 10ms.
4. Set the waveform to run status.
5. Click on "Begin Record". Counter starts counting the frame number.
6. Click on "End Record" to stop the recording or wait until the counter has reached the value of the end frame.

Hint:

1. Waveforms of the two channels can be recorded simultaneously.
2. You can switch the channels on and off during recording. Only channels that are switched on can be recorded. If a channel is switched off during recording, there will be no recording of that channel when the end frame is reached.

19.2 Playback of the recorded waveform




1. Select the "Play" button.
2. Click "From..." to open the waveform file you want to play back
3. Set the start frame "Start" and the end frame "End".
4. Set the interval time of the playback.
5. Click "Repeat" to select a playback loop. If "Repeat" is deactivated, the waveform is repeated only once.
6. Click "Play". "Number" shows the current frame number that is being played.
7. Click "Pause" to pause the playback.
8. Drag the slider to display the frame number you need.

Hint:

1. When the software is in RUN mode, the current data collection is stopped during playback.
2. If other menu items are selected during playback, playback is paused.

20. Setting up the software settings

Click  to display the main menu and select "Settings". Language
Select the national language.

Design

Select a black or blue software window design. Then "Restart" appears below "Design". After the software has been restarted, the window appears in the new design.

Open file

Select a saved BIN waveform file and open it.

Print view

Click here to open a print view of the waveform window. Menu selection in the print

view window:

File→Page Layout : Set the page margins Print: Open
print options Exit: Close print view

Display→Page orientation : Switch between landscape and portrait format Full
page: Display full page

Normal: display current selection User Scale...:

Set a zoom factor

Show waveform background colour: Mark is set and the background colour changes to the background colour set in the software window; click again to remove the mark and the background colour is white.

Set preview background colour: Click to display the colour selection dialogue box and select the colour of the preview page.

Save image

Click "Save Image" to save a screenshot as an image file (png, bmp or gif).

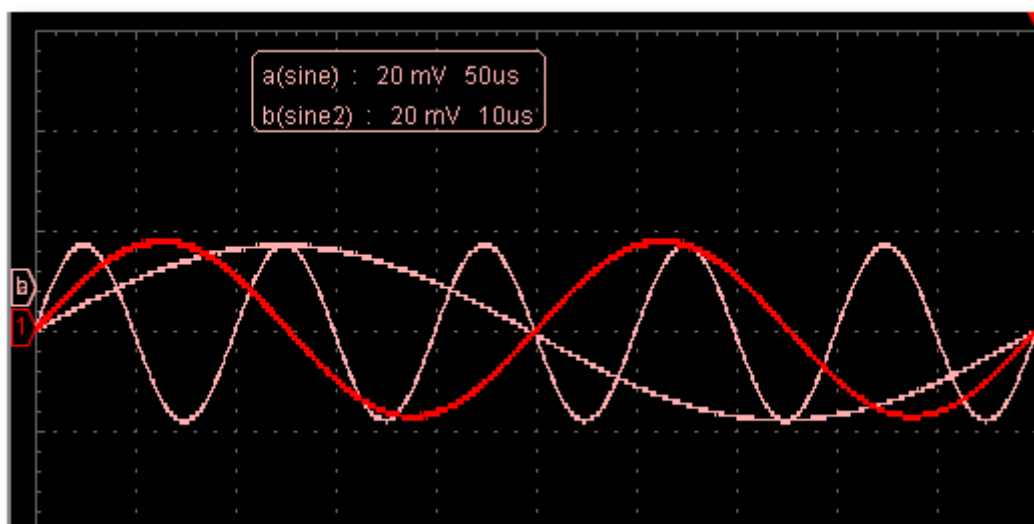
Save / Display

With this function, up to 8 reference waveforms can be stored. These waveforms can be displayed simultaneously with the current waveform. The recalled waveform cannot be adjusted or changed. The source can be CH1, CH2 or maths.

To save the waveform of CH1 as object "A" and then recall it, please proceed as described:

1. Select CH1 as the source.
2. Select "a" under "To Object".
3. Under "Name" enter an object name under which you want to save the waveform, e.g. "Sine". Click on "OK". The object is now saved / renamed as "a (sine)". This step can be skipped.
4. Click on "save".
5. Select "a (sine)" from the list, click "Show", the reference waveform is shown on the display. The object name and relevant information are also shown in the upper left corner of the screen. You can continue and map more reference waveforms on the screen.
Click "Remove All" to delete all reference waveforms from the screen.

If the selected object is not a saved waveform, "Not saved" is displayed.



20.1 Pause&Export

Export the waveform to a file in the specified format according to current recording length. You can choose between different file types: bin, txt, csv or xls.

You can also click on the  icon on the user interface to save it in any format (bin, txt, csv or xls) format.


20.2 Self Calibration (Self Cal)

With the help of self-calibration, the oscilloscope can quickly reach the optimum state to obtain accurate measurement results. You can perform this self-calibration at any time. Self-calibration must be performed when the ambient temperature changes by 5 °C or more.

Before performing a self-calibration, disconnect any probes or connecting leads from the inputs. When the unit is ready, click on "Self Cal".

20.3 Default (factory settings)

Click on "Default" to reset the unit to the factory settings.

You can also click on the  icon in the user interface to reset the unit to factory settings.

20.4 Help

Open the help document for this software. Alternatively, you can press the "F1" key.

21. Network

The oscilloscope can be connected to a PC via LAN port. For more information, see 23. Using the LAN port

Help window

Select a tab - "Channel", "Capture & Period" or "Trigger" to view an introduction to these functions. Click anywhere in the help window to display the next page.

If you do not want the help window to be displayed automatically every time you start the software, place a tick next to "Do not show again".

Info

Device version, serial number and PeakTech homepage are displayed.

Multi I/O

Control the function of the port 5 connector in the oscilloscope. Trigger

In: Input of the synchronous trigger signal

Trigger Out: Output of the synchronous trigger signal

Pass / Fail: high level if PASS; low level if FAIL

22. Using executive buttons

Performing keys are:

Auto Set  , Run/Stop  /  , Single Trigger  . See also [7.Introduction to the User Interface](#).

AutoSet:

This key is used to automatically set all the control values of the unit needed to generate a visible waveform. Press the **AUTOSET** key; the oscilloscope will then perform a quick automatic measurement of the signal.

The following table shows the parameter values of the **AUTOSET** function:

Parameter	Value
Acquisition Mode	Current
Vertical Coupling	DC
Vertical Scale	Adapt to corresponding pitch
Horizontal Level	Medium
Horizontal Scale	Adapt to corresponding pitch
Trigger Type	Current
Trigger Source	Show minimum number of channels
Trigger Coupling	Current
Trigger Slope	Current
Trigger Level	Average setting
Trigger Mode	Edge
Display format	YT

Run/Stop On - and off switching of the sampling of an input signal.

Single Trigger :You can start the trigger mode as "Single" directly, so when the trigger occurs, you get a waveform image and then the trigger stops

Keyboard Shortcuts

Ctrl + Enter: Auto set

F5: Run / Stop

23. Using the LAN connection

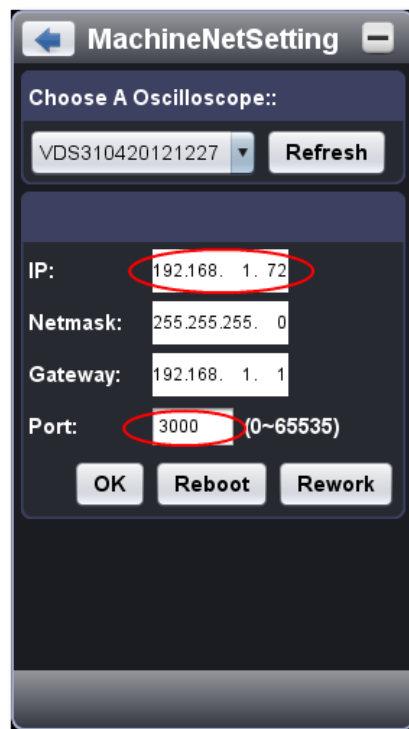
With the LAN connection, the oscilloscope can be connected to a PC directly or to a network via a router. These two types of connection are described below.

23.1 Direct connection with a LAN connection cable

1. Look at the network parameters of the computer
Then look at the IP address of the computer you want to connect the machine to. Suppose the IP address of the computer is 192.168.1.71.
2. Setting the network parameters of the oscilloscope
 - (1) Establish a connection via USB and call up the main menu: Use the supplied USB cable to connect the oscilloscope to a PC.


After the connection is successfully established, click on  to display the main menu and select "Settings", then click on "Network".

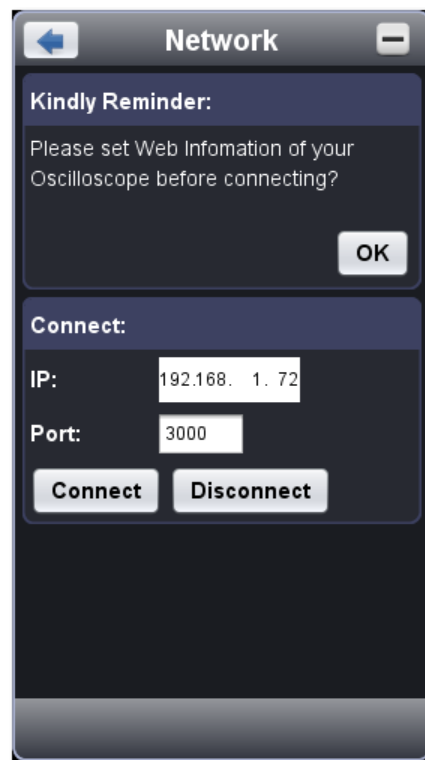
- (2) Setting the IP address and port of the oscilloscope:
In the network menu, click "OK" to enter device network settings. Select the oscilloscope to be connected from the list. If it is not listed, click "Refresh" to update the list. Enter the IP address of the oscilloscope. The first three bytes should be the same as the IP address of the computer (example: 192.168.1.71). The last byte should be different. Here, we set it to 192.168.1.72.
The range of port values is 0 ~ 4000, here we set it to 3000.



- (3) Click "Rework" to restart the oscilloscope.

23.2 Setting the network parameters in the software

- (1) Power supply of the oscilloscope:
Disconnect the USB cable from the computer. Connect the oscilloscope to the AC adapter.
- (2) Connect the LAN cable to the LAN interface of the oscilloscope; plug the other end into the LAN interface of the computer.
- (3) Set the parameters in the network menu:
Click  to display the main menu and select "Settings", click "Network". Set the IP address and port to the same value of the oscilloscope as in step 2.
- (4) Click on "Connect"



23.3 Connection to a network router

1. look at the network parameters of the computer

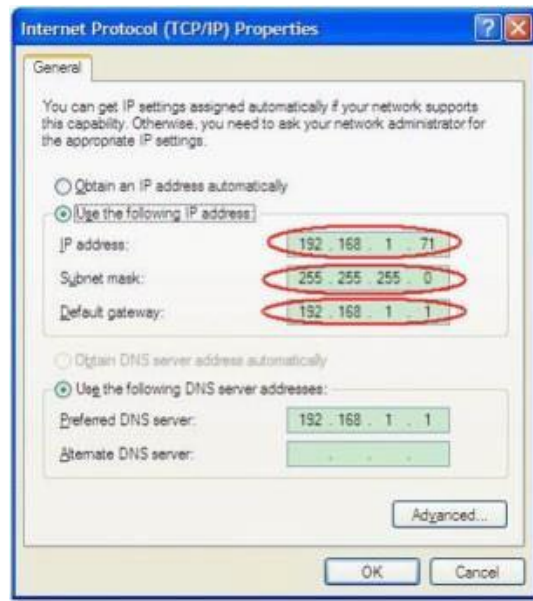
Then look at the IP address of the computer you want to connect the machine to. The default gateway and subnet mask should be set the same as the router.

Accepted:

IP address: 192.168.1.71

Subnet mask: 255.255.255.0


Default gateway: 192.168.1.1



23.4 Setting the network parameters of the oscilloscope

Establish a connection via USB and call up the main menu:

Use the supplied USB cable to connect the oscilloscope to a PC.

After the connection has been successfully established, click on  , in order to display the main menu and select "Settings", then click "Network".

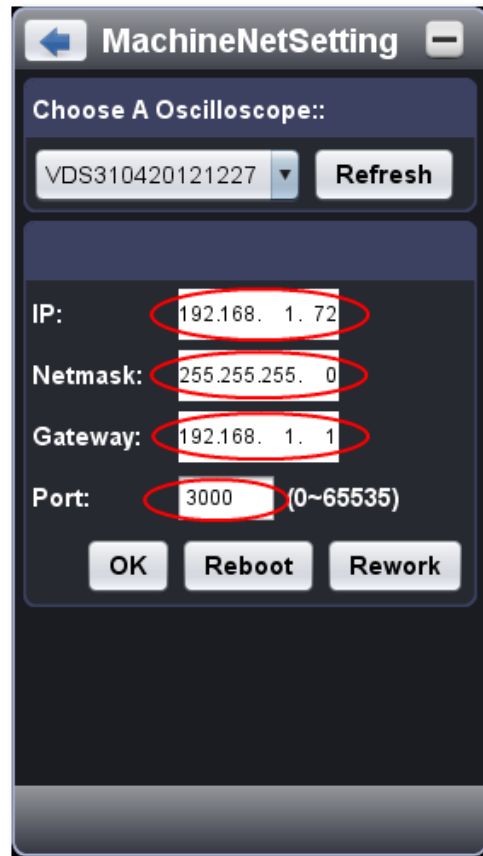
23.5 Setting the IP address and port of the oscilloscope:

In the network menu, click "OK" to enter device network settings.

Select the oscilloscope to be connected from the list. If it is not listed, click on "Refresh" to update the list. Enter the IP address of the oscilloscope. The first three bytes should be the same as the IP address of the computer (example: 192.168.1.71). The last byte should be different. Here we set it to 192.168.1.72

The range of port values is 0 ~ 4000, here we set it to 3000.


The default gateway and subnet mask should be set the same as the router.



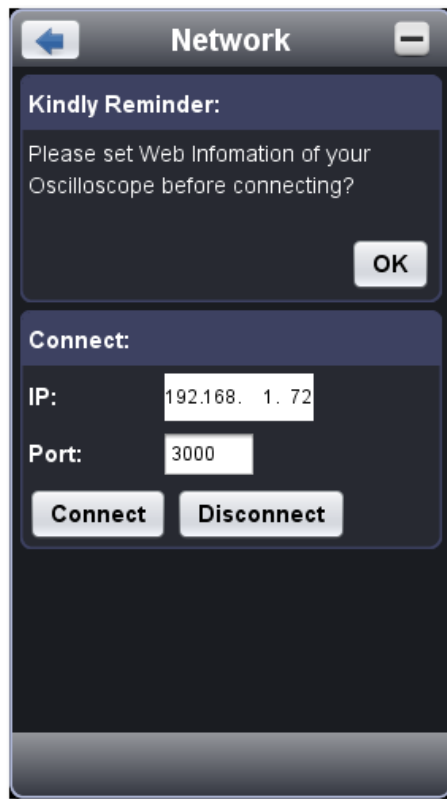
The screenshot shows a mobile application interface titled "MachineNetSetting". It features a section "Choose A Oscilloscope::" with a dropdown menu displaying "VDS310420121227" and a "Refresh" button. Below this, there are four input fields, each with a red circle highlighting the entered value: "IP:" with "192.168. 1. 72", "Netmask:" with "255.255.255. 0", "Gateway:" with "192.168. 1. 1", and "Port:" with "3000" (the range "0~65535" is shown to the right). At the bottom, there are three buttons: "OK", "Reboot", and "Rework".

23.6 Setting the network parameters in the software

- (1) Power supply of the oscilloscope:
Disconnect the USB cable from the computer. Connect the oscilloscope to the AC adapter.
- (2) Connecting to the router: Use a LAN cable (patch cable) to connect the oscilloscope to the router. The computer should also be connected to the router.
- (3) Set the parameters in the network menu:

Click , to display the main menu and select "Settings", click "Network". Set the IP address and port to the same value of the oscilloscope as in step 2.

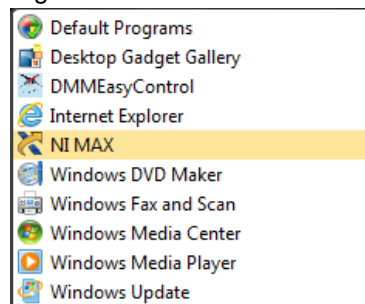
- (4) Click on "Connect"



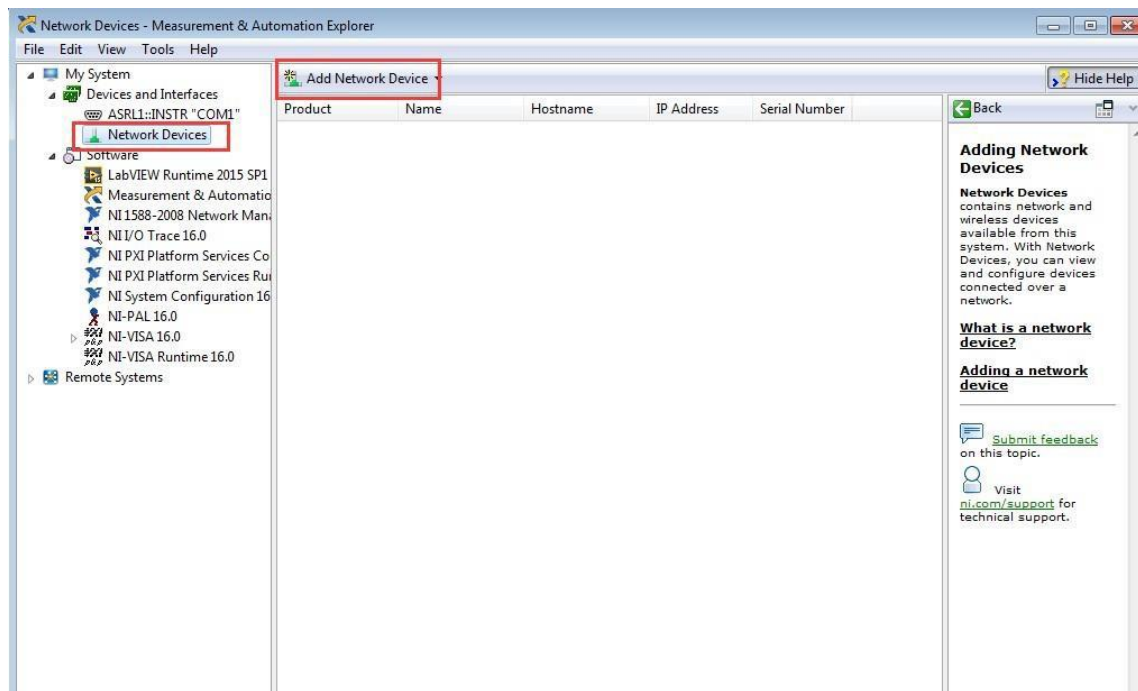
23.7 Connecting the unit to the network

If the oscilloscope does not automatically connect to the software via the LAN operating mode, create a new network device via the NI MAX Manager. To connect the device to the network, proceed as follows:

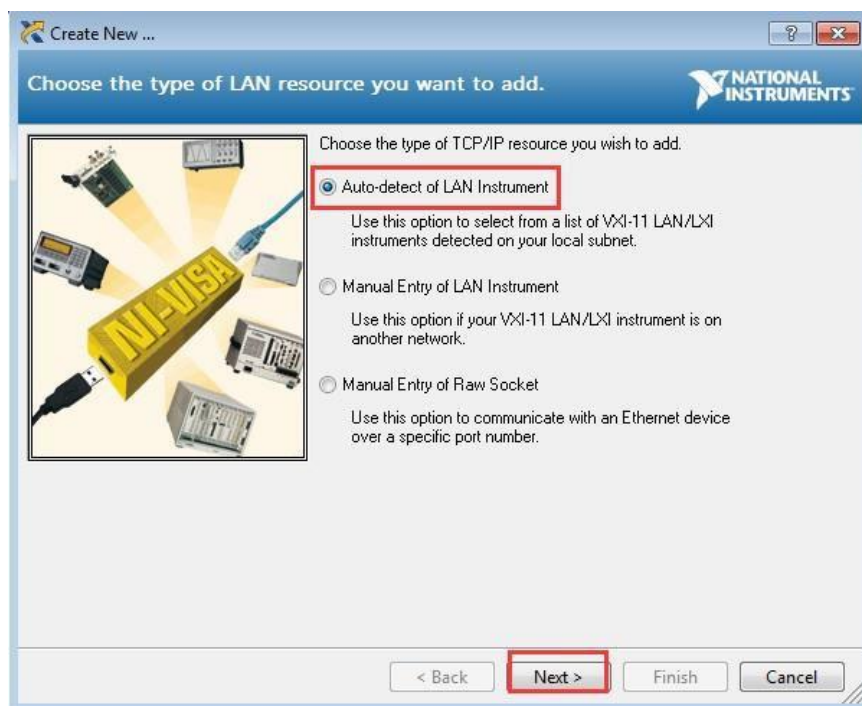
Locate and select the NI MAX Manager from the Windows Start menu.



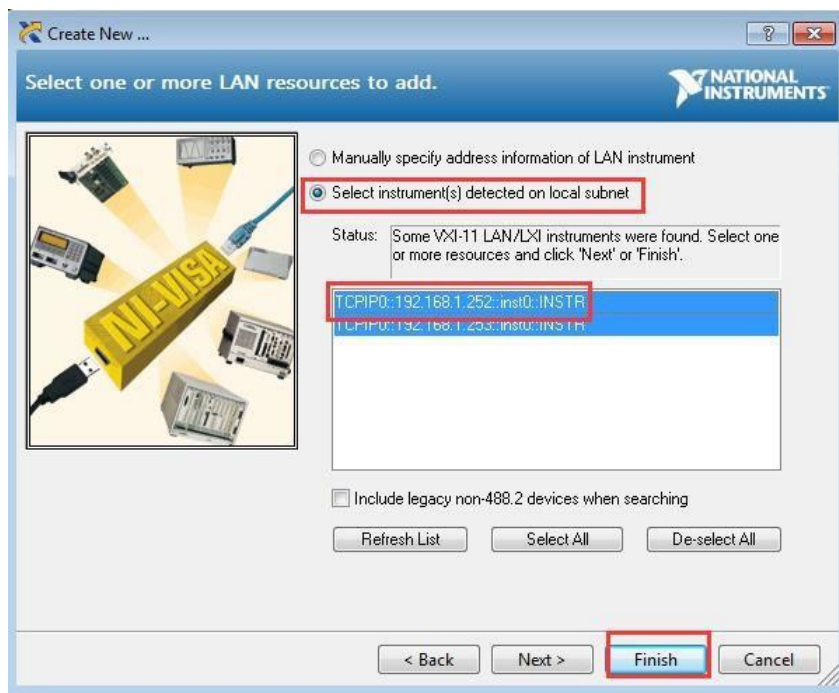
The NI MAX Manager opens. There, select the option Add Network Device as shown in the following picture:



Press the VISA TCP/IP Resource button and the interface for creating the network device opens.



Select the automatic search function to find the oscilloscope in the network. After this step, another window opens where you can select the oscilloscope.



After you have added the device manually, open the software again and the device will now be displayed under the menu item USBFound.

23.8 Connecting the unit via the WiFi function (hotspot, optional function)

To use the WiFi (WLAN) function of the oscilloscope, it is essential that the PC to be used supports WiFi communication. Furthermore, a WiFi USB stick is required, which is plugged into the WiFi interface (USB slot) of the oscilloscope.

For the WiFi connection function, press the Home button in the menu in the software and click on the Settings (Utility) icon and press the Network button.

Now select the WiFi setup.

In the WiFi setting interface, proceed as follows:

Select the AP connection type. Now enter the SSID and select the password. Enter the IP address as shown in the following picture and select the port (the port can be a value between 0 and 65536). Confirm the entries with Ok.



If automatic switching between WiFi and LAN connection is not possible on the target PC, deactivate LAN.

Check "Obtain IP address automatically" for the IP address of the target PC or use the same first 3 sections of the IP address as in point b. under i), e.g. 192.168.100.

To use the unit via WiFi, set up the NI MAX interface as shown in the example in the subitem 23.7 a.

Now the oscilloscope can be selected in the list of found devices in the main menu at the top left.

23.9 Connecting the unit via the WiFi function (router)

To use the WiFi (WLAN) function of the oscilloscope, it is essential that the PC to be used supports WiFi communication. Furthermore, a WiFi USB stick is required, which is plugged into the WiFi interface (USB slot) of the oscilloscope.

For the WiFi connection function, press the Home button in the menu in the software and click on the Settings (Utility) icon and press the Network button.

Now select the WiFi setup.

In the WiFi setting interface, proceed as follows:

Select the connection type STA. Now enter the SSID of the router and then enter the password for the router. Enter the IP address as shown in the following picture and select the port (the port can be a value between 0 and 65536). Confirm the entries with Ok.



The SSID and password in this case is an example of how the router is configured. After entering the information, confirm the entry with Ok. After confirming, you are automatically returned to the About menu of the WiFi settings. Go to the WiFi settings again and press the Refresh button to make sure that the entered data has been transmitted.

To use the unit via WiFi, set up the NI MAX interface as shown in the example in the subitem 23.7 a.

Now the oscilloscope can be selected in the list of found devices in the main menu at the top left.

24. Technical specifications

Unless otherwise stated, the technical data apply only to oscilloscopes with a set probe attenuation of 10X. The technical data only apply if the oscilloscope fulfils the following two conditions: at least

- * The unit should run continuously for 30 minutes.
- * Perform "self-calibration" if the operating temperature changes by up to or even more than 5°C.

Performance features		Comments		
Bandwidth		P 1286	100 MHz	
		P 1326	70 MHz	
		P 1331	100 MHz	
		P 1286	2	
Channels		P 1326 / 1331	4	
		Mode	Normal, Peak detection, Average	
Scanning	Sampling rate (real time)	P 1286	Dual CH	500 MSa/s
			Single CH	1 GSa/s
		P 1326	Four CH	250 MSa/s
			Dual CH	500 MSa/s
			Single CH	1 GSa/s
		P 1331	Four CH	250 MSa/s
			Dual CH	500 MSa/s
			Single CH	1 GSa/s
Entrance	Input coupling	DC, AC, Ground		
	Input impedance	1 MΩ ± 2% parallel with 15 pF ± 5 pF		
	Probe damping factor	1X, 10X, 100X, 1000X		
	Max. Input voltage	40 V (DC + AC peak-to-peak)		
	Channel Isolation	50Hz : 100 : 1 10 MHz : 40 : 1	400 V (peak-to-peak) (DC + AC peak-to-peak)	
	Bandwidth limit	20 MHz, or full bandwidth		
	Time delay between channels (typical)	150 ps		

Horizontal system	Measuring rates Range	P 1286	Dual CH	0.5 Sa/s ~ 500 MSa/s		
			Single CH	0.5 Sa/s ~1 GSa/s		
		P 1326	Four CH	0.5 Sa/s ~ 250 MSa/s		
			Dual CH	0.5 Sa/s ~ 500 MSa/s		
			Single CH	0.5 Sa/s ~1 GSa/s		
		P 1331	Four CH	0.5 Sa/s ~ 250 MSa/s		
			Dual CH	0.5 Sa/s ~ 500 MSa/s		
			Single CH	0.5 Sa/s ~1 GSa/s		
		Interpolation	(sin x)/x			
	Max memory length	P 1286	Dual CH	≤ Max. Sampling rate	10 M	
			Single CH			
			Single CH			
			Single CH			
P 1326		Four CH	≤ Max. Sampling rate	10 M		
		Dual CH				
		Single CH				
P 1331		Four CH	≤ Max. Sampling rate	10 M		
		Dual CH				
	Single CH					
Scan speed (Sa/div)	P 1286		2 ns/div ~ 100 s/div, step by 1 ~ 2 ~ 5			
	P 1286 / 1331		1 ns/div ~ 100 s/div, step by 1 ~ 2 ~ 5			
Sampling rate / time delay accuracy	± 25 ppm					
Interval (Δ T) Accuracy (DC ~ 100 MHz)	Single: ± (1 time interval + 25 ppm x measured value + 0.6 ns); average > 16: ± (1 time interval + 25 ppm x measured value + 0.4 ns)					

Vertical system	A/D - converter	P 1286		8 bits resolution (2 channels simultaneously)		
		P 1326 / P 1331		8 bits resolution (4 channels simultaneously)		
	Sensitivity	P 1286		2 mV/div ~ 5 V/div		
		P 1286 / P 1331		2 mV/div ~ 5 V/div		
	Displacement	P 1286 / P 1326 / P 1331		± 2 V (2 mV/div - 50 mV/div) ± 20 V (100 mV/div - 500 mV/div) ± 40 V (1 V/div - 5 V/div)		
	Analogue bandwidth	100 MHz (P1286, P 1331), 70 MHz (P 1326)				
	Single bandwidth	Full bandwidth				
	Lowest frequency	≥ 10 Hz (at input, AC coupling, -3 dB)				
	Rise time	P 1286 / 1331		≤ 3.5 ns (at the input, typical)		
		P 1326		≤ 5.0 ns (at the input, typical)		
	DC accuracy	± 3%				
	DC accuracy (average)	Average ≥ 16: ± (3% rgd + 0.05 div) for Δ V				
	Waveform inverted ON/OFF					

Measurements	Cursor		Δ V and Δ T between the cursors	
	Automatic measuring functions		Vpp, Vmax, Vmin, Vtop, Vbase, Vamp, Vavg, Vrms, Overshoot, Preshoot, Freq, Period, Rise Time, Fall Time, Delay A→B A→B +Width, -Width, +Duty, -Duty	
	Mathematical waveform		+, -, *, /, FFT	
	Lissajou's figure	Bandwidth	Full bandwidth	
Phases - Difference		± 3°		
Communication interfaces			P 1286 / 1326 / 1331	USB, LAN, USB Host (WiFi module optional)

Trigger:

Performance features		Comments	
Trigger level range	Internal	± 5 div from the screen centre	
Trigger level accuracy (typical)	Internal	± 0.3 div	
Displacement trigger		According to the memory length and time base	
Trigger Holdoff Range		100 ns ~ 10 s	
50% level setting (typical)		Input frequency ≥ 50 Hz	
Edge trigger		Slope	Ascending, Falling
Pulse trigger		Trigger condition	Positive pulse: >, <, = Negative pulse: >, <, =
		Pulse width range	30 ns ~ 10 s
Video Trigger		Modulation	NTSC, PAL and SECAM
		Line number range	1-525 (NTSC) and 1-625 (PAL/SECAM)
Slope Trigger		Trigger condition	Positive pulse: >, <, = Negative pulse: >, <, =
		Time setting	30 ns ~ 10 s

Function generator (P 1286)

Performance features	Comments
Resolution	0.1 Hz, or 5 digits RMS value
Stability	±50 ppm at 0 °C - (+40 °C)
Measured value Deviation	Up to ±50 ppm per year
Sinus	0.1 Hz - 5 MHz
Rectangle	0.1 Hz - 200 kHz (with a rise and fall time of <200ns)
Ramp	1 Hz - 10 kHz (with a rise - fall time ≥ 5µs)
Pulse	1 Hz - 10 kHz (with a pulse width of ≥ 5µs)

Amplitude (standard load at 50 Ω, only P 1286)

Output amplitude	10mVpp - 5Vpp (≤5MHz) @ High Z
Amplitude accuracy	± (1% of setting + 1 mVpp) (typical, 1kHz sinus, 0V offset)
Amplitude resolution	10 mVpp
DC Offset Range (AC+DC)	± (2.5 Vpk - amplitude Vpp/2)
DC Offset Accuracy	± (1 % of setting + 1 mV + amplitude Vpp x 0.5%)
DC Offset Resolution	10 mVpp

Communication / Interfaces

Interfaces	USB, USB Host (WiFi), LAN
Programming commands	SCPI

Power supply

Performance features	Comments	
Power supply	P 1286 / 1326 / 1331	5 - 15 V DC/ 1.2 A
Power consumption	P 1286 / 1326 / 1331	< 8 W

Environmental conditions

Temperature	Operating temperature: 0°C ~ 40°C Storage temperature: -20°C ~ 60°C
Relative humidity	≤ 90 %
Altitude a. s. l.	In operation: 3,000 m Not in operation: 15,000 m
Cooling	Natural convection

Mechanical specifications

Dimensions (W x H x D)	P 1286 / P 1326 / P 1331	120 x 18 x 190 mm
Weight		380 g

Appendix A:

Accessories included:

- Probe: 1.2 m, 1:1 (10:1) switchable
2 pc. P 1286
4 pcs P 1326/1331
- 1x CD (including software and operating instructions)
- 1x USB interface cable
- 1x protective cover
- 1x AC-DC mains adapter

All rights reserved, including those of translation, reprint and reproduction of this manual or parts thereof.

Reproductions of any kind (photocopy, microfilm or any other method) are only permitted with the written permission of the publisher.

Last version at time of printing. We reserve the right to make technical changes to the unit in the interests of progress.

We hereby confirm that all units meet the specifications stated in our documents and are delivered calibrated at the factory. A repetition of the calibration after 1 year is recommended.

© **PeakTech®** 12.2022 Po./Ehr./Lie.