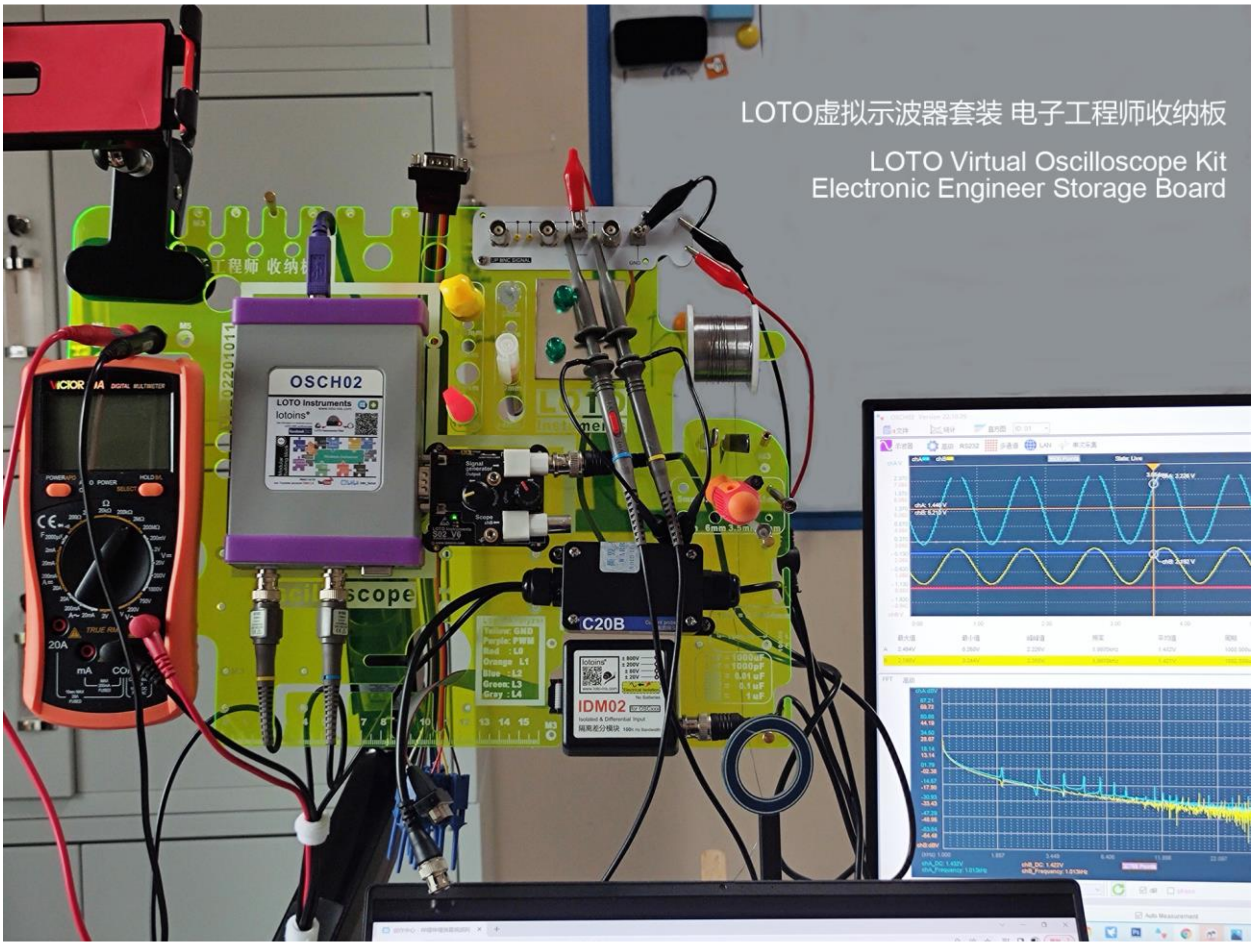


LOTO PC Oscilloscope

Software User's Manual

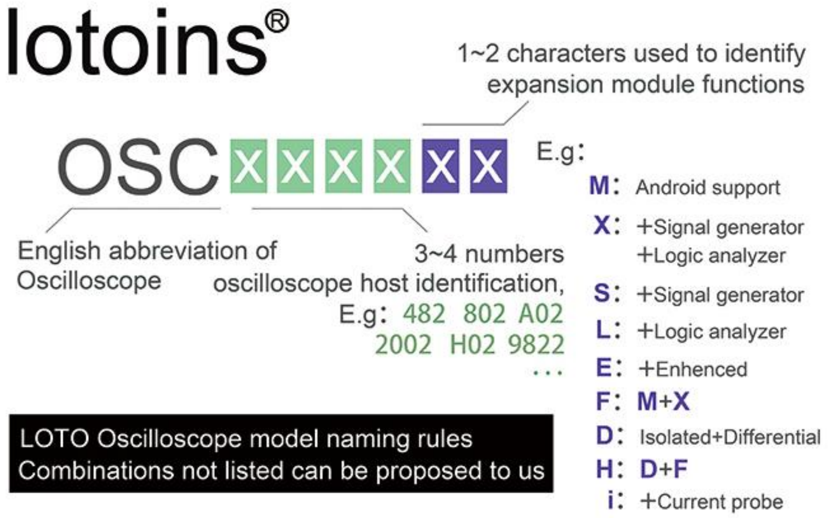
Contents

1 Welcome

Thanks for choosing LOTO USB oscilloscope solutions. LOTO Instruments is committed to the development of high-performance software for virtual instrumentation products. It has been committed to the research and development of virtual instruments for many years, improving the cost, the functional architecture of traditional instruments and providing cost-effective products. Welcome to contact us (173393190@qq.com) for any suggestion, customization development or business cooperation.

2 Software Version

This manual describes the common software features of all models.



The hardware specifications of different models can be found in their respective data sheets. This manual is based on the software version in the list below.

Model		Software version	Release date
OSC482	OSC482	version 23.06.01	2023/06/01
	OSC482S		
	OSC482L		
	OSC482X		
	OSC482F		
	OSC482H		
	OSC482D		
	OSC482M		
OSC802	OSC802	version 23.07.08	2023/07/08
	OSC802D		
OSCA02	OSCA02	version 23.07.08	2023/07/08

	OSCA02D OSCA02L OSCA02M OSCA02S OSCA02X OSCA02F OSCA02H		
OSC2002	OSC2002 OSC2002E OSC2002L OSC2002D OSC2002S OSC2002X OSC2002F OSC2002H OSC2002M	version 23.07.08	2023/07/08
OSC980	OSC980 OSC984	version 22.11.17	2022/11/17
SIG851		version 1.0.1	2020/12/10
SIG852		version 1.0.2	2022/01/09
OSCH02	OSCH02 OSCH02L OSCH02D OSCH02S OSCH02X OSCH02F OSCH02H	version 23.07.08	2023/07/08

3 Software and driver down load

To make sure LOTO users could have the best experiences and better products, we keep updating oscilloscope software-correcting bugs, adding practical features, and improving software operations.

The latest version of software &driver &documents will be updated to the link:

http://www.loto-osc.com

We can download the corresponding software from google drive:

https://drive.google.com/open?id=1Py0I-_Uud0JqKU4wiCyk8YOeHiVqj_y-

No matter when you download, you will download the latest version. There is no fixed schedule of updates, we usually release a new version in every one or two months.

bilibili loto_lucus
YouTube TAO LV
LOTO USB oscilloscopes
LOTO Instruments Files

You can also contact us via Facebook public group: LOTO USB oscilloscopes

LOTO USB oscilloscopes

4 Safety Warning

△1 **Make sure that the input of the device does not exceed the voltage range.**

△2 **Make sure that the black clip of the probe is well grounded and equipotential to the PC power supply during measurement.**

△3 **Do not connect the probe to the mains voltage socket (110V / 220V).**

WARNING!	Damages caused by violation of this safety warning will not be covered by the warranty.

△1 note: the oscilloscope device has an input range of ±5v and it should never be exposed to higher voltages although it has an over-voltage protection circuit which can protect the device from voltages even ten times the maximum allowed.

To avoid the risk of damage, it is required to select an input probe with the attenuation factor according the nature of the voltage to be measured to ensure that the device is used within the voltage range.

The device is provided with a 1X/10X probe: when the 1X position is selected, the voltage input range is ± 5v. When the 10X position is selected, the voltage input range is ±50v. If an optional 100X probe is used, the voltage input range is ±500v. Optional isolated differential module can measure ±800v.

△2 note: the virtual oscilloscope device takes power directly from the USB port of the PC, so its ground is connected to the PC ground as also the black clip of the probe. If the PC is powered by the power supply from the mains voltage socket, then the PC should be connected to the ground through a 3-hole power socket. In case the device is connected to a PAD or laptop and these are powered just by a battery, then the ground will be suspended. But in most cases, the device ground, PC ground and probe black clips are all connected to the ground. If the black clip is used to connect a non-earth potential in the circuit under test, it is equivalent to short-circuiting the circuit under test with the black clip directly to the ground.

△3 note: as described in △1, measuring 220V mains voltage is equivalent to measuring a voltage signal with a peak-to-peak value of about 611V. In this case it would be necessary to use a high source voltage probe with attenuation 100:1.

In addition, as described in △2, when the probe is connected directly to 220VAC, it is equivalent to passing 220VAC L or N through the oscilloscope's probe GND line and from this to connect it to the oscilloscope's internal GND and to the AC input protection ground.

In severe cases, the probe or oscilloscope may be burned, and since the oscilloscope itself (such as the metal parts of the case) is electrically connected to the probe GND, it could be dangerous for the human body. For this to measure mains, use a differential isolating probe specifically designed for a high source voltage.

The recommended practice is to use our associated isolation differential module IDM01 to measure the high voltage with uncertain grounding.

---

5 Minimum System Requirements

Operating system	Microsoft Windows XP, Win 7, Win 8, Win10. Supports both 32 and 64-bit systems.
CPU	2.0GHz or above
RAM	1.0GB or above
Software package	.net framework 2.0
Monitor Screen	60 Hz , >1024*768 resolution
Ports	USB 1.1 compliant port minimum. USB 2.0 compliant port recommended. Must be connected direct to the port or a powered USB hub. Will not works on a passive hub.

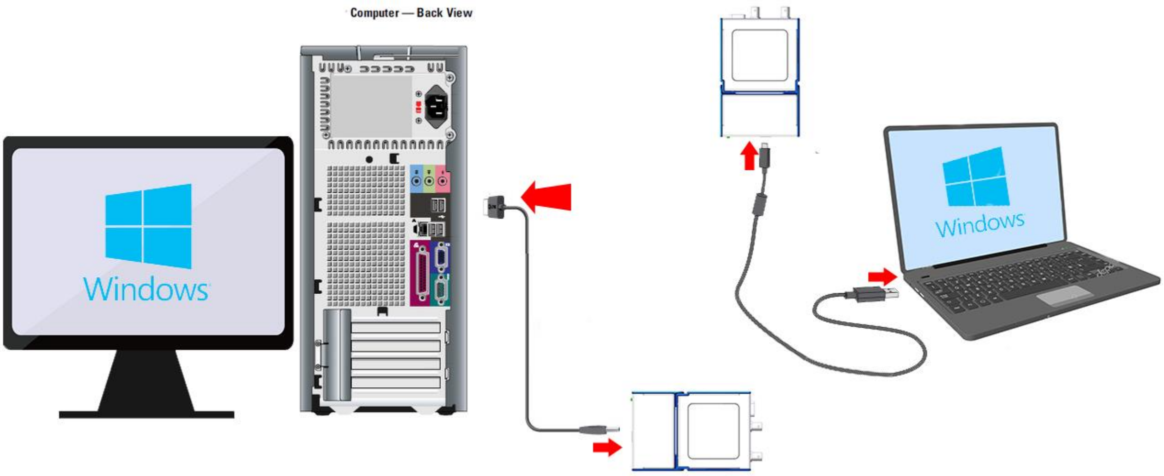
Warning

The software application requires Microsoft .NET framework 2.0. This component is an integral part of Windows systems since Win 7, so there is no need to install it on these systems, however on Windows XP system it may be required to install this component to use the software. The software installation process will determine if this component is required and will download and install it if necessary.

6 Using OSCxxx for the first time

We designed this virtual oscilloscope to be as simple as possible. Even connecting it for the first-time can be quick by following the steps below.

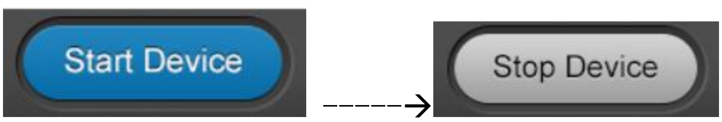
(1) : Connect the virtual oscilloscope to the PC via the supplied USB cable. You should use only the supplied USB cable, or use a cable of a better quality. Poor quality cables may cause communication problems or may become unusable. If you are using the device on a desktop PC, be sure to connect it to the USB port on the back of the chassis. The front ports may also be unstable due to power supply issues. If you use a laptop, it is easy to connect to the USB ports on the sides.



(2) : Install the driver. When the virtual oscilloscope is connected to the PC for the first time, in Win 8.1 and Win10/11 systems the device driver will be automatically recognized and installed. In Win XP and Win 7 32-bit operating systems the device will be recognized, and the system will require the driver installation. The steps for installing the driver are different according the operating system. See Sections 8.1 and 8.2 for more information on the specific installation process.

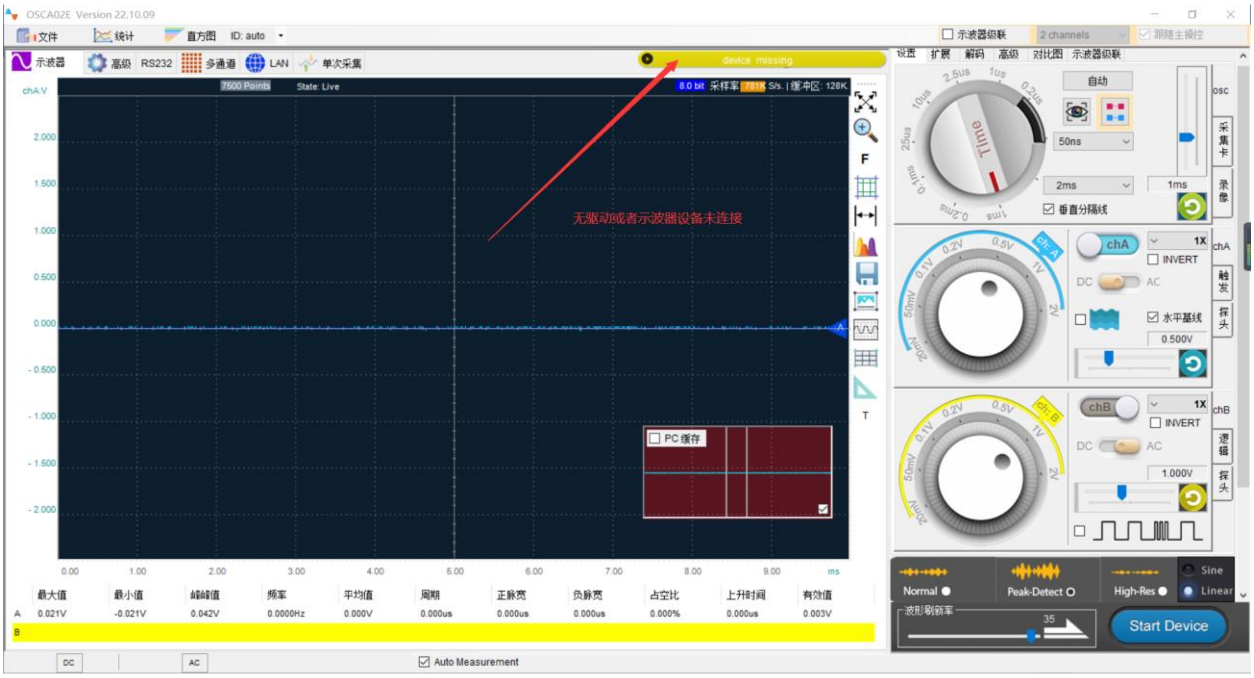
Microsoft Windows xp Windows 7 Windows 8 Windows 10

(3): Open the virtual oscilloscope software, click the **Start Device** button in the lower right corner, then you will see that channel A has been turned on by default, with the zero-voltage baseline in the middle of the screen. Channel B can be enabled in its setting area. When the zero-voltage baseline is in the vertical center, this indicates that the virtual oscilloscope is operating normally.

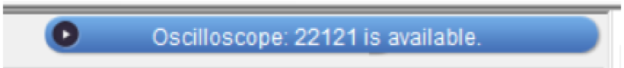


As shown in the figure above, after turning on all the oscilloscope channel switches, there are noise lines, which means that the virtual oscilloscope can work normally.

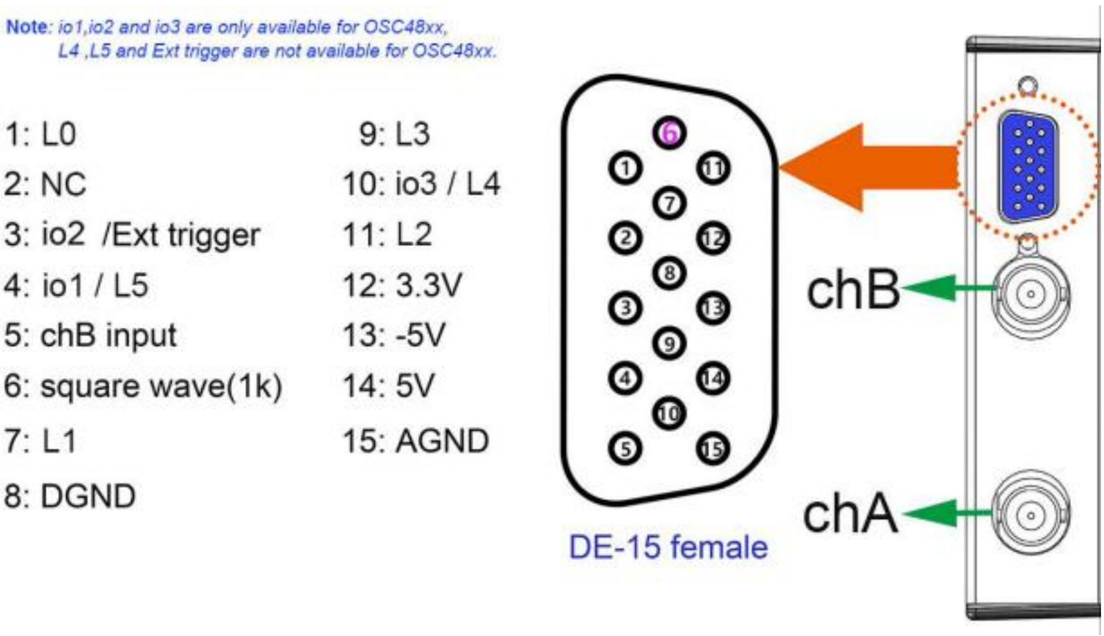
If the driver is not installed successfully, or the oscilloscope device is not connected to the computer, there will be a yellow warning bar above the LOTO oscilloscope software:



Normally it should be blue here:



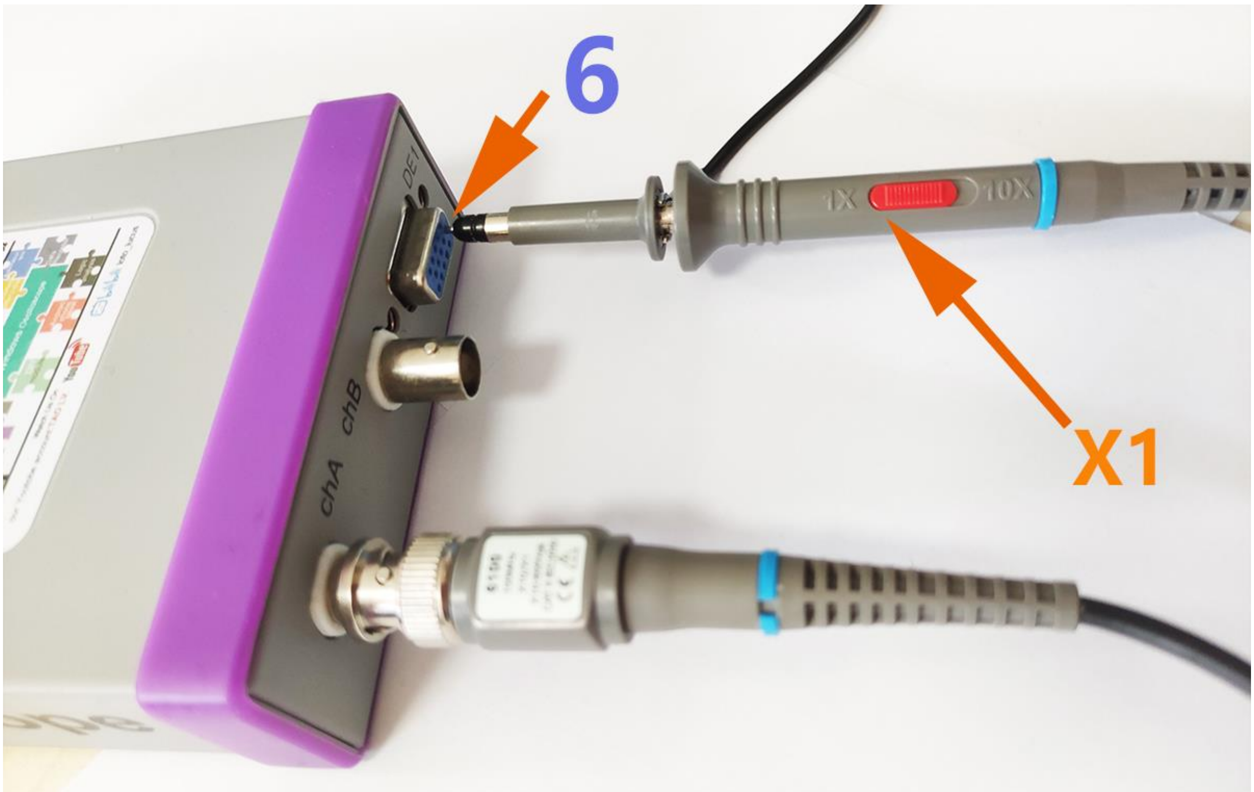
(4): Use the standard probe calibration signal to display your first waveform on the screen. Some kind of device has an extended metal contact next to the two BNC input connectors that output a square wave with amplitude of approximately 3.3V and a frequency of 1 KHz. The other kind of device has this square wave output in pin 6 of a DE-15 interface.



This signal allows a quick check whether the product is working normally when there is no other AC signal at hand. Set one probe with x1 selector; connect it to the oscilloscope channel A port from the BNC connector side and to the metal contact on the other side. Adjusts the time knob in the software to 1ms position. The square wave will appear on the screen.

Measuring the built-in square wave is an important way to check the quality of the oscilloscope. If you are not sure where the problem is in your actual application, please disconnect the oscilloscope from the target system as soon as possible, and use the oscilloscope to test the built-in square wave.

Wave to troubleshoot whether it is the problem of the oscilloscope. If it is normal for the oscilloscope to measure its own square wave, then the oscilloscope can basically conclude that there is no problem.



6.1 Video demonstration:

<LOTO USB Oscilloscopes Basics --- First Use Guidence>

https://www.youtube.com/watch?v=mUM9R6rnH90

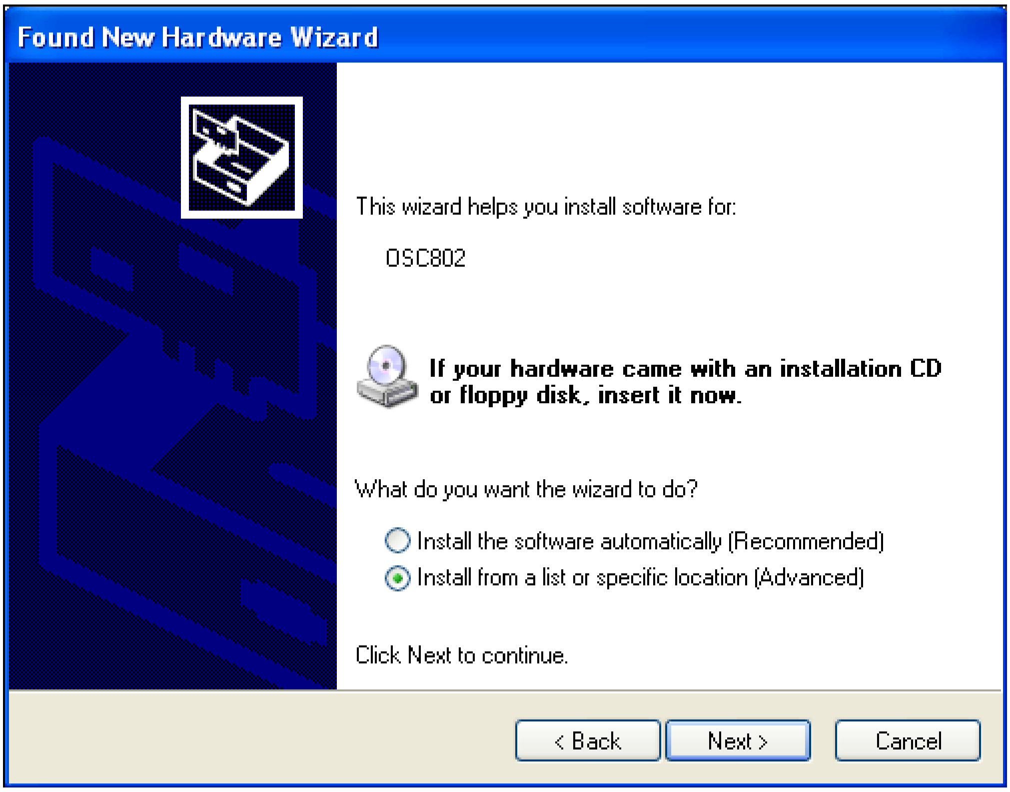
7 Drivers Installation

This chapter describes the detailed steps of installing drivers for this product under various types of Windows operating systems, which can cover the guidance most users need to install drivers. However, in some users' operating systems, there will be exceptional situations of driver conflict, which will lead to the failure of driver installation or the failure of normal opening of the device when the installation is successful. Users can refer to the detailed solutions for driver installation under exceptional circumstances listed in Appendix 1 of this manual.

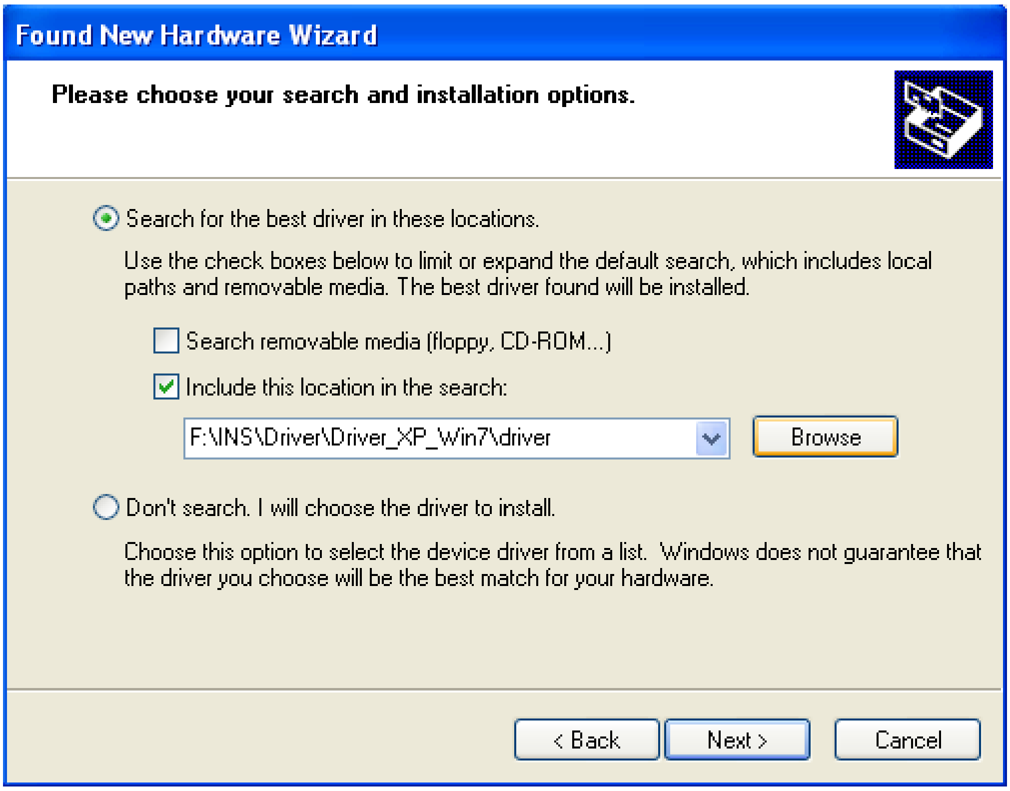
7.1 Driver installation on Windows XP

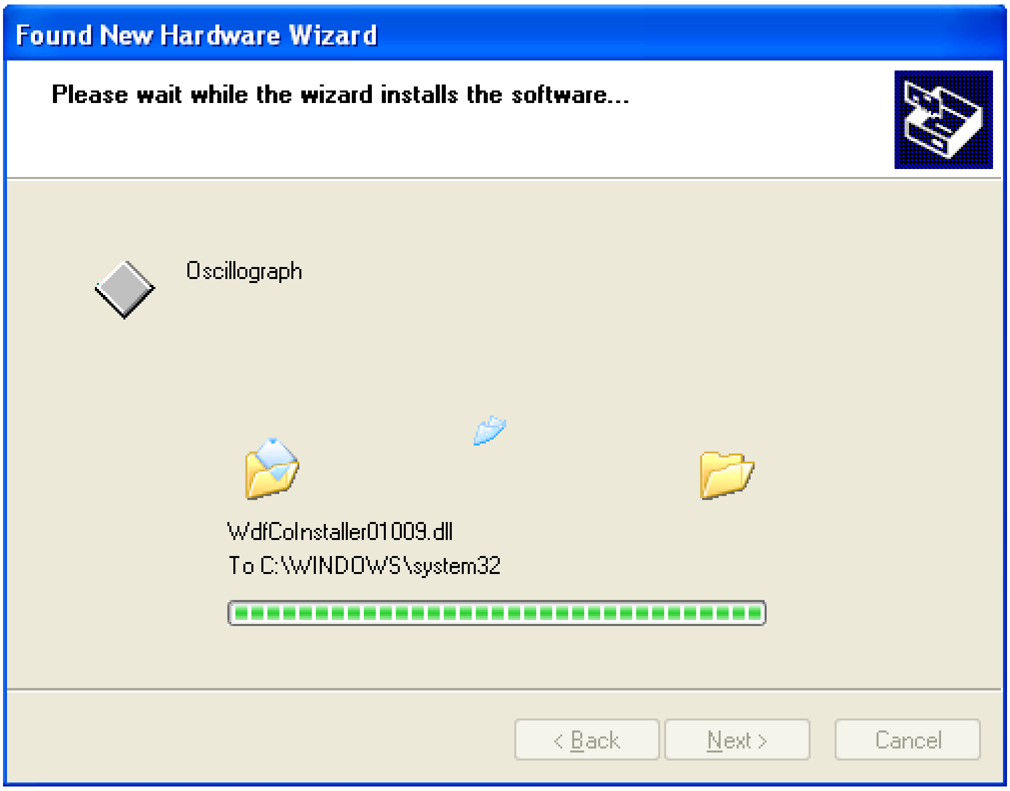
1: Select “Install from a list or specific...”, then click on “Next” :

> **Note:**
> For install the driver manually right clicks on **Computer** and select **Manage** so to enter in the **Device Manager**. In **Other devices** search the device with name **OSCxxx.** Then right click on it and select **Update Driver.**

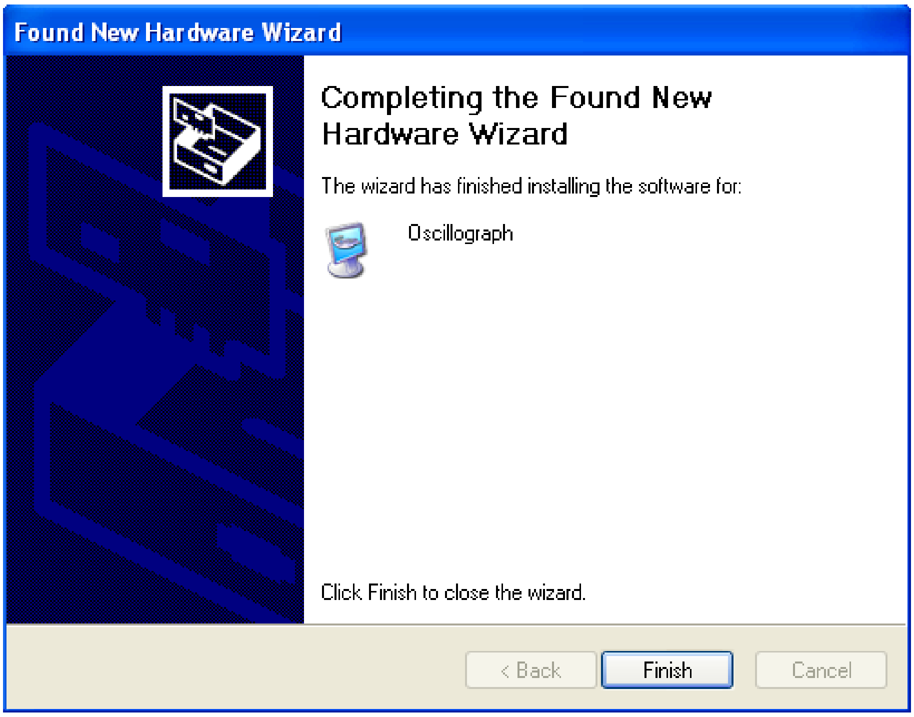


2: Click on **Browse**, specify the search path as "(software installation package path)*driver*". For example "Driver_XP_Win7\driver", then click **next**:





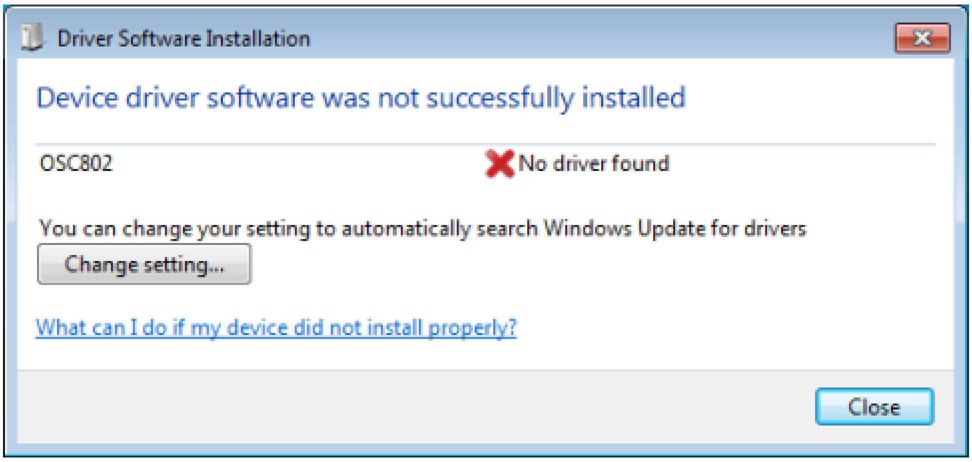
3: Once the installation is complete, you can see the installed driver in the **Device Manager**:



7.2 Driver installation on Windows 7 - 32 bit

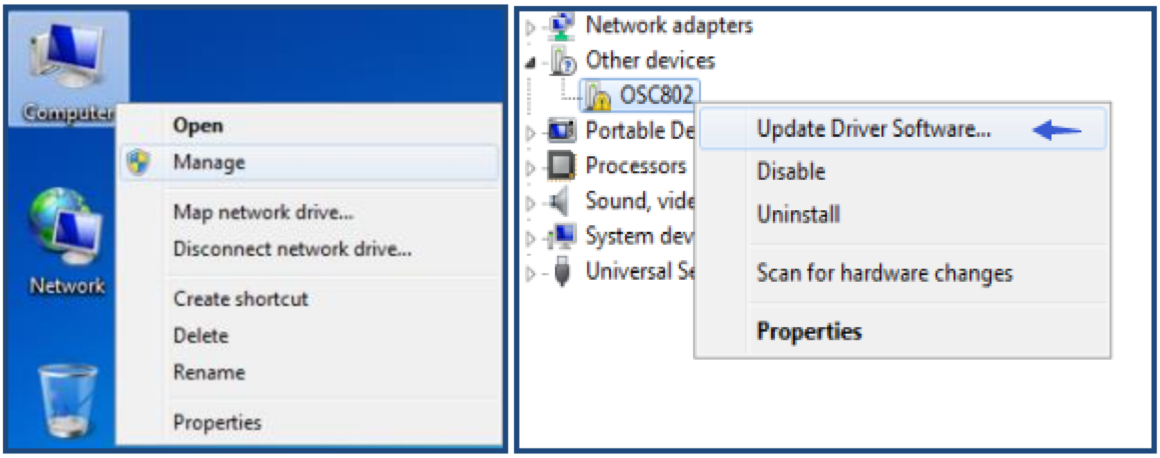
Warning:
Windows 7 – (32bit and 64 bit) cannot install the driver automatically. It is required to install the driver manually.

Windows 7 will automatically search for the driver through Windows Update when the device is connected, but this automatic installation will fail since the driver is not present in the system:

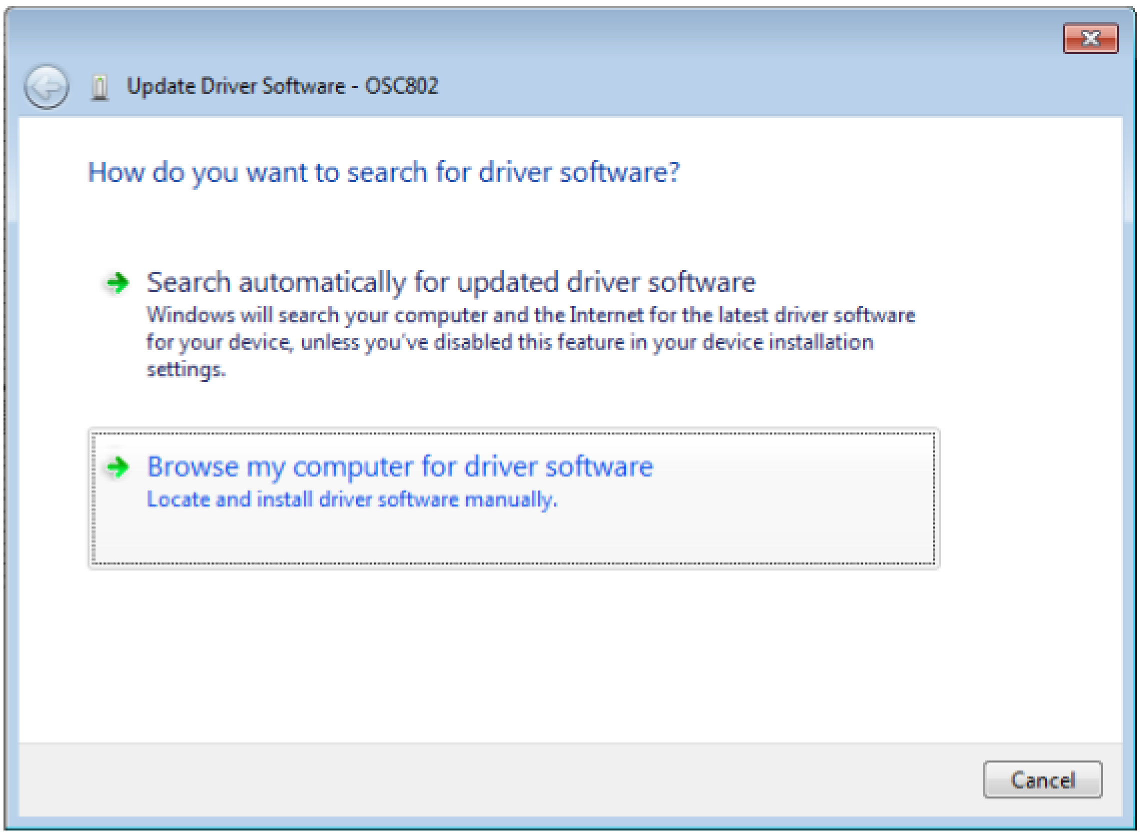


Manual installation:

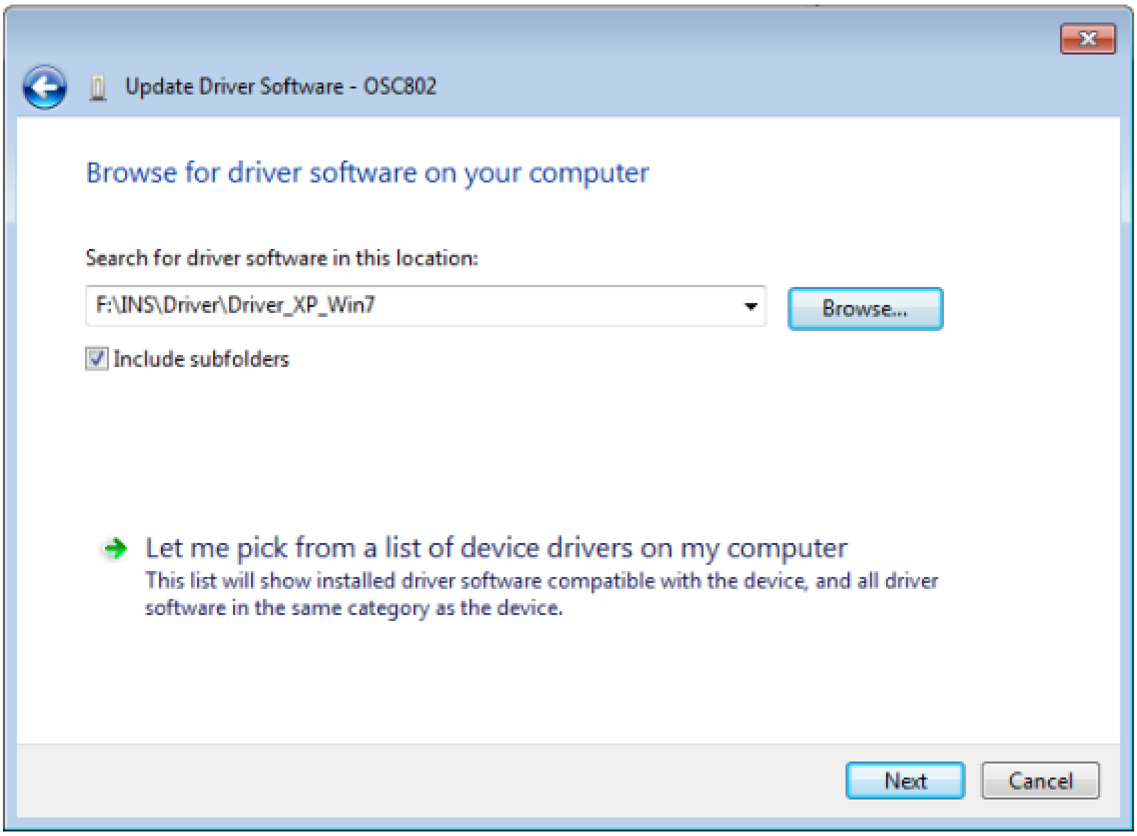
1. Right click on **Computer**, then select **Manage.** Enter in the **Device Manager**, search for the device with name **OSCxxx** with the yellow exclamation mark in **Other devices**, and then right click on it and select **Update the Driver Software**:



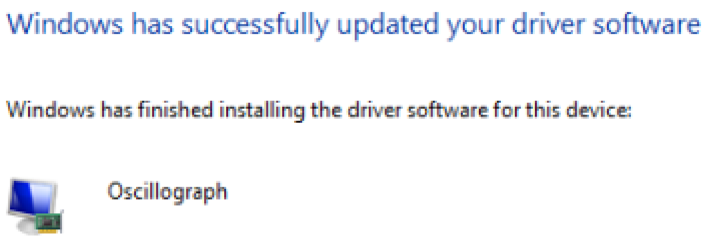
2. In the dialog box, select “Browse my computer for driver software”:



3. In the next dialog box specify the driver location as "*(Installation package path)\driver*" via the **Browse...** button. For example ":\ *Driver_XP_Win7\driver*":



4. Select "Install this driver software anyway":



7.3 Driver installation on Windows 7 - 64 bit

Manual installation

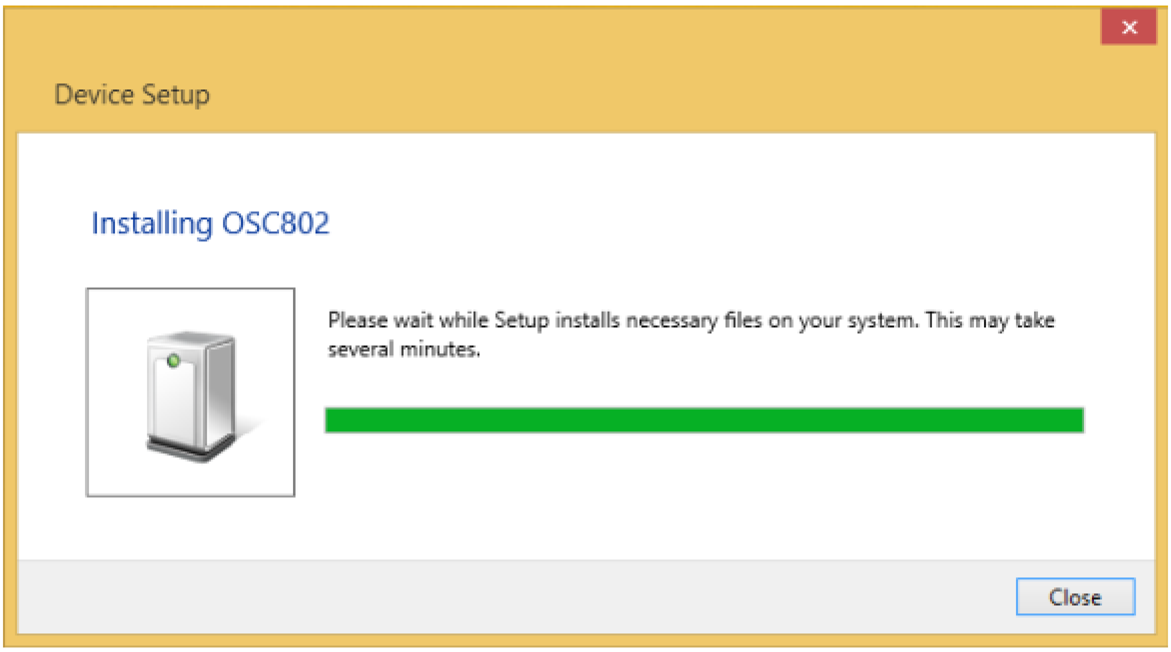
1. Right click on **Computer**, then select **Manage.** Enter in the **Device Manager**, search for the device with name **OSCxxx** with the yellow exclamation mark in **Other devices**, and then right click on it and select **Update the Driver Software**.

2. Use the **Browse...** button to specify the driver location as "*(Installation Package Path)\driver*". For example "*:\ Driver_Win7_64\driver*".

3. Other steps are the same as for the above windows 7 32-bit system.

7.4 Driver installation on Windows 8 and Windows 10

The system will automatically install the drivers for the device, and the user does not need to proceed with any manual installation. Some systems cannot recognize the oscilloscope device normally, please follow the steps in Appendix 1 to install the driver.



7.5 Driver installation issues solution

If the above steps still do not allow to install the driver properly, in Appendix I you can find a detailed solution.

8 Introduction to the software usage

This section describes the basic concepts and usage of the virtual oscilloscope software.

8.1 Some basic concepts on the PC virtual oscilloscope

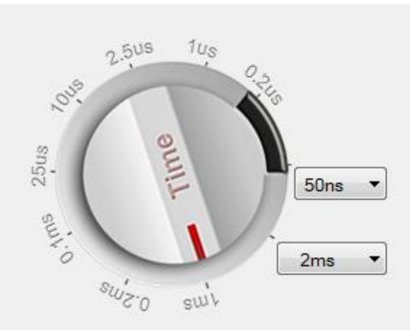
An oscilloscope is a measuring instrument that displays the relationship between voltage and time of an electric signal or a waveform.

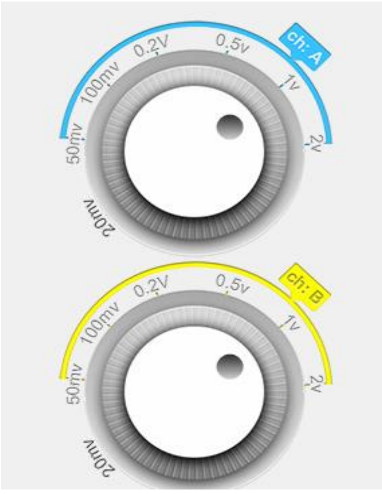
When displaying the voltage-time relationship, the abscissa represents the time value (from left to right for the direction of time growth), and the ordinate represents the voltage value (from bottom to top for lower voltages to higher voltages).

Unit (Time)	Description
s	Seconds
ms	Milliseconds, that is, one thousandth of a second (10^{-3} s)
us	Microsecond, one thousandth of a millisecond (10^{-6} s)
ns	Nanoseconds, one thousandth of a microsecond (10^{-9} s)
v	Voltage in Volts
mv	Millivolt, that is, one thousandth of a Volt (10^{-3} V)

Waveform plots: In the oscilloscope software the drawing area of the waveform is evenly divided with grids which serve for quickly measurement on the signal.

The oscilloscope software interface provides two types of knobs for setting the grid units, one for the time and one for the voltage.

	Time axis setting. The value set by the knob will set the time span for the horizontal grid divisions.

	Voltage axis setting (one for each channel). The value set by the knob will set the voltage span for the vertical grid divisions. The voltage span can be different for each channel

All the input channels of this oscilloscope are synchronized in time, so they share the same time axis and the same time settings from the corresponding knob.

The sampling rate from the Analog to Digital converter in the oscilloscope hardware vary automatically according the selected time scale so to make a better usage of the device internal memory, without losing resolution in the time analysis.

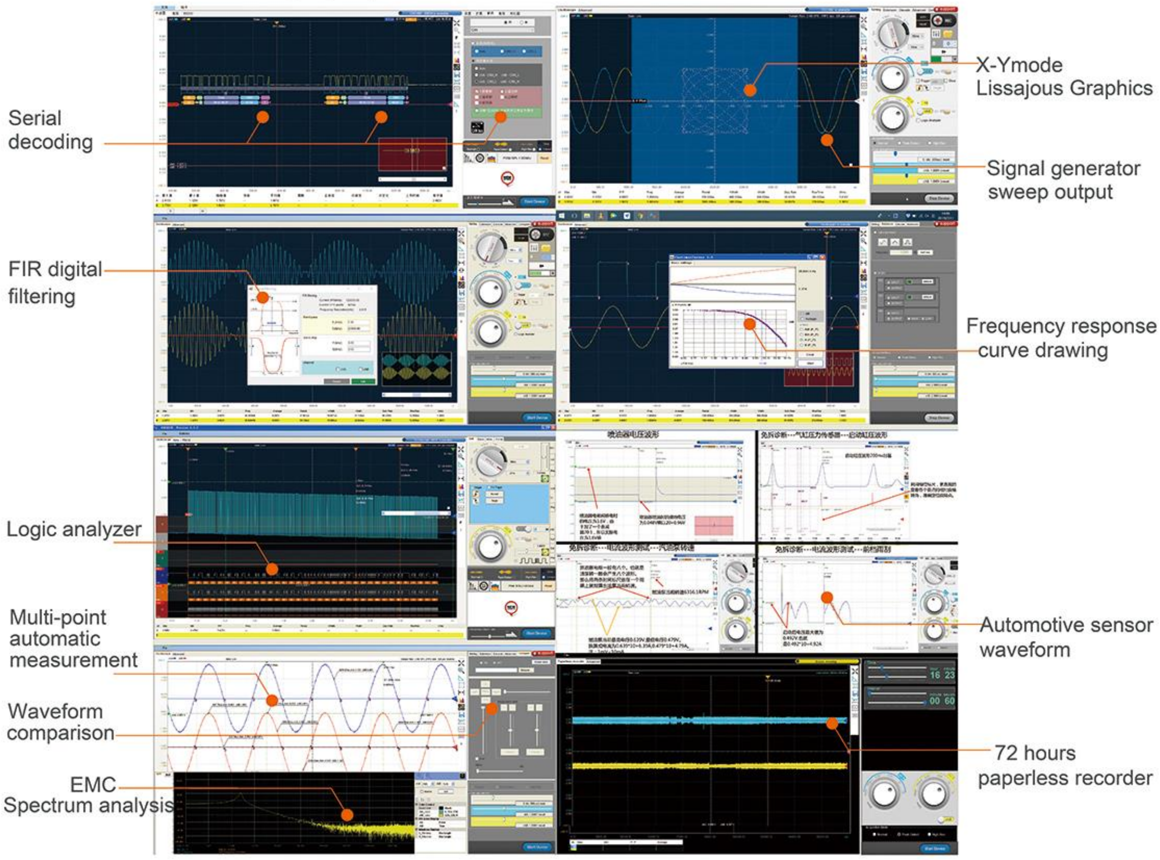
8.2 Introduction to the Virtual Oscilloscope Software

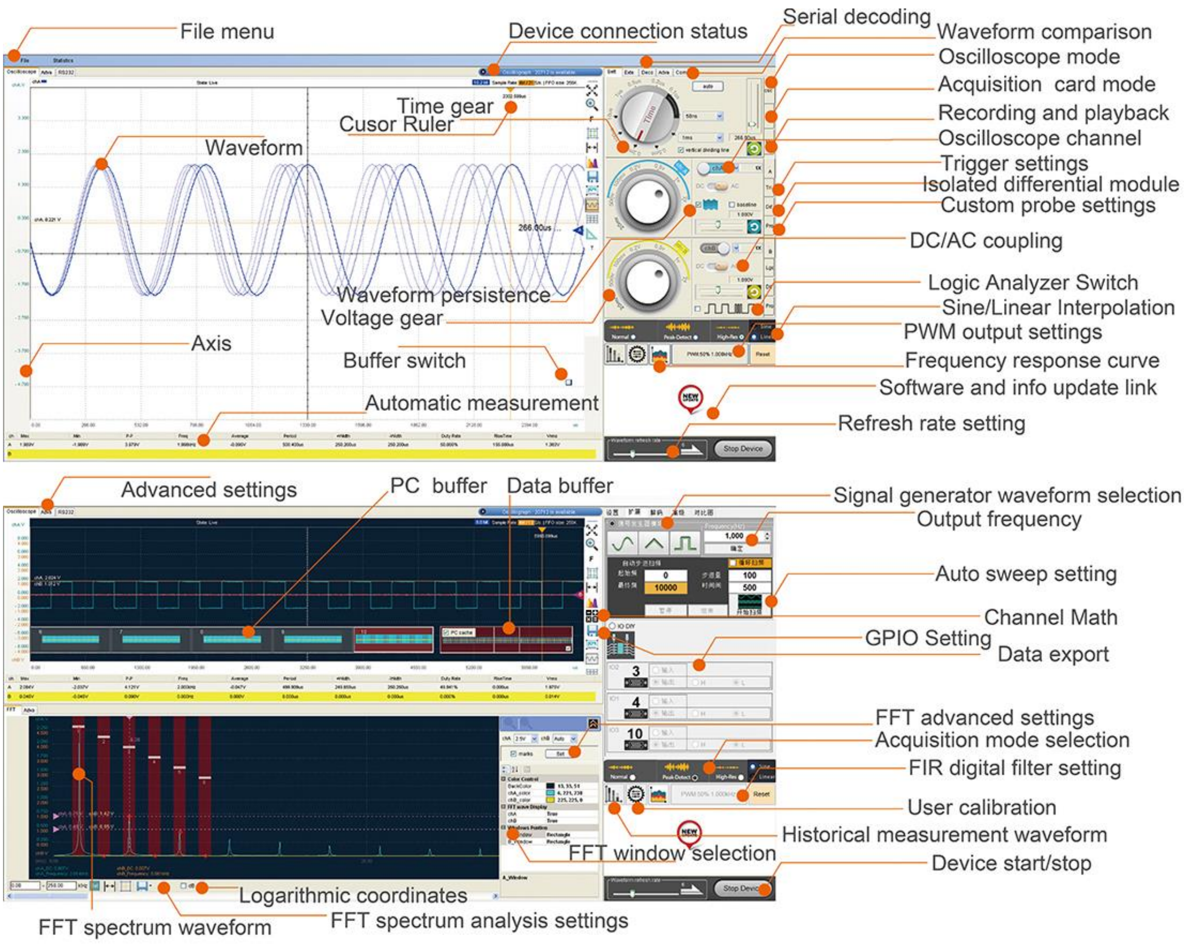
Double-click on the application icon to open it.

- If the application does not start probably it is due to an incorrect installation of the .NET Framework 2.0 under Win XP systems. In this case please refer to the relevant instruction in Chapter 8.

- If the application does not start under Win 7 / Win 8, it is possible that the issue is due to the system permission settings. In this case right-click on the application icon and select to **Run as administrator**.

Interface display of part of the Windows software



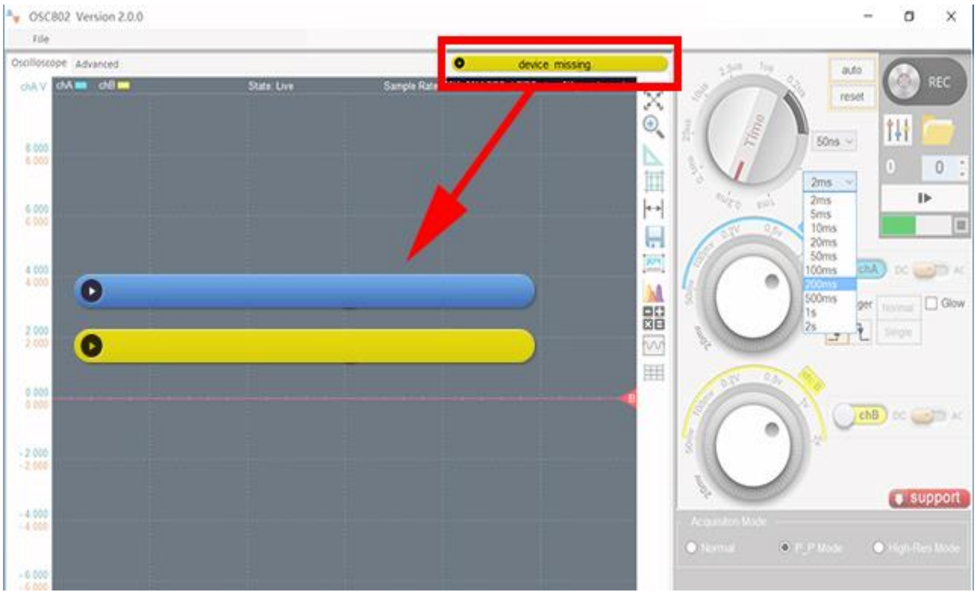


8.2.1 Device monitoring

After the software interface is opened, the OSCxxx device status will be monitored in real time. When the device is connected on the USB port of the PC, the software interface will display a message on blue background.

If the device gets disconnected physically or logically, the software interface will display a message on yellow background.

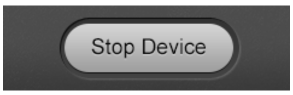
If the software always displays a yellow bar after plugging in the device, it means that the connection is abnormal or the oscilloscope driver is not installed.



8.2.2 Start and stop

After the driver has been successfully installed, if the oscilloscope is connected when the software is opened, the software automatically turns on the device and starts the real-time acquisition. If the device is connected before opening the software, it is required to start the device manually by clicking on the **Start Device** button in the lower right corner of the software or by pressing the **Enter key** （**shortcut key** (Enter) - start/stop） on the keyboard.

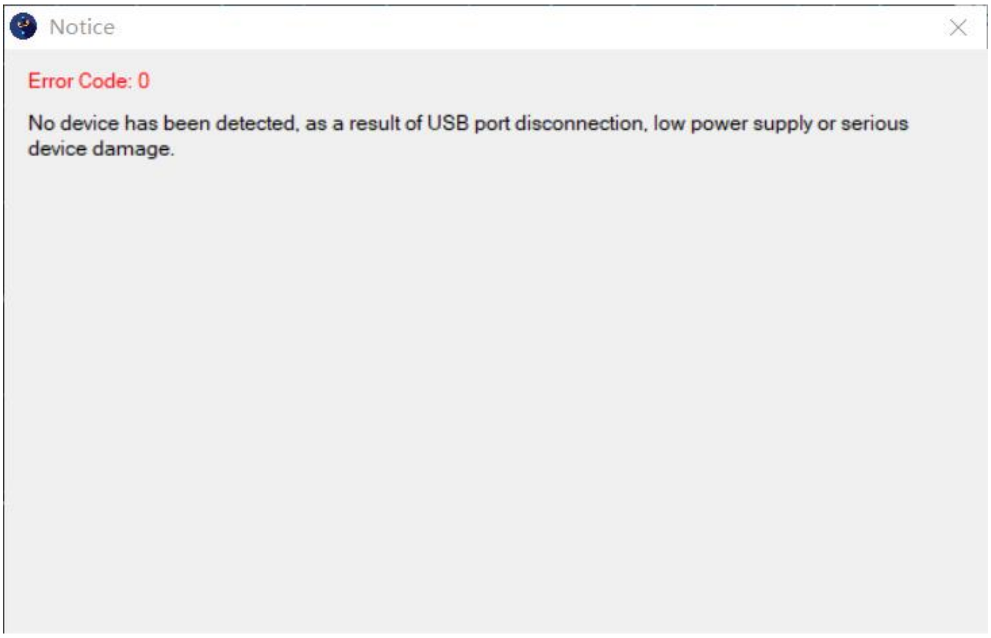
After the device is started the **Start Device** button becomes the **Stop Device** button which allows to stop the acquisition at any time. The acquisition can be stopped also by pressing the **Space Bar** on the keyboard or when the USB cable is unplugged during the operations.



The start/stop button can only control the device real-time acquisition, but it is not used for play/stop operations on past recorded data. In this case there are specific control buttons. For details, please refer to Chapter 9.2.8.

8.2.2.1 start errors

Pressing the boot device button usually encounters two error messages: 0 and 1.



If the error code 0 is reported, the possible reasons are that the USB cable is damaged/device is damaged, the driver is not installed, the device is not connected, or the software and the oscilloscope model do not correspond.

If Error code2 is reported, the possible reason is that the USB power supply is insufficient, or the driver of the oscilloscope device conflicts with other drivers, or the oscilloscope is set as a slave device when it is cascaded.

> **Note**
> When the device is not used it is recommended to close the software and then to unplug the device's USB cable.

8.2.3 Waveform display

8.2.3.1 Waveform curves

The waveforms are drawn on the plot area using different colors to distinguish them. The default background color of the plot area is dark blue. The waveform for channel A (chA) and channel B (chB) are drawn respectively in cyan and yellow colors:
.

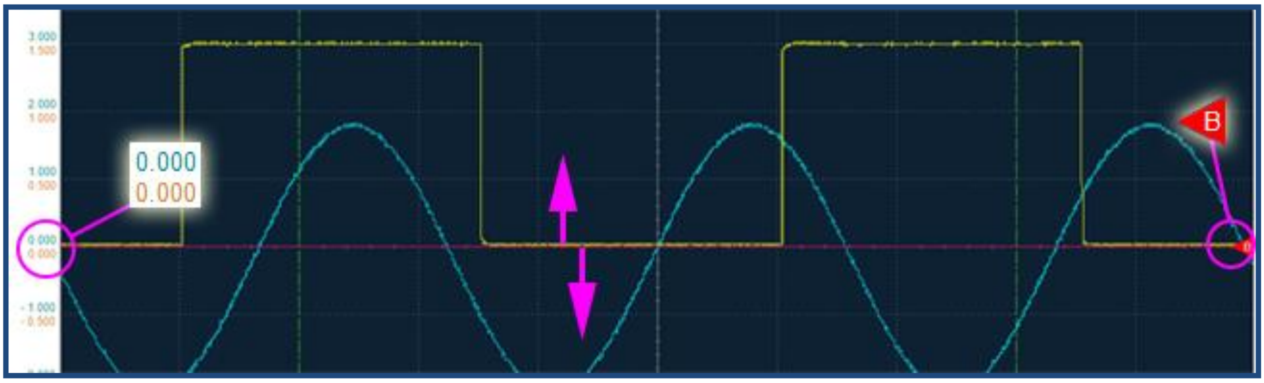
8.2.3.2 Zero-voltage baseline

The channel's waveforms are drawn according their zero-voltage baselines. These lines represent 0 Volts; positive voltages are drawn above while negative voltages are drawn below the baselines. The zero-voltage baseline position of each channel is indicated by a horizontal line and a triangle on the side which can be dragged with the mouse to move the baseline. A number inside the triangle indicates the corresponding channel number. By default, the zero-voltage baseline is set in the middle of the screen for all channels. Channel A (chA) waveform and the channel B (chB) waveform uses respectively blue and red colors for the zero-voltage baselines, as: 

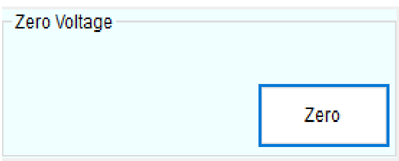
, 

. When the mouse is placed on the triangle of the zero-voltage baseline and the left button is clicked, you can drag up and down the line to change its position.

The zero-voltage baseline for each channel is identified by a triangle on the right side of the waveform display area. Channel A baseline is in blue while channel B baseline is in red. You can use the mouse to slide up and down these triangles to adjust the levels for the zero-voltage baselines. The grid values on the left axis will change accordingly. By default, the zero-voltage baselines for channel A and channel B are in the middle of the display area.

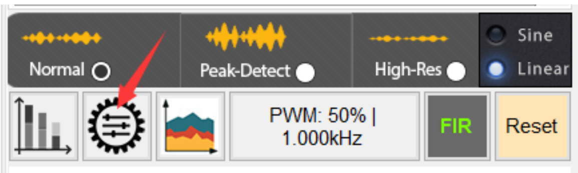


The user can self-calibrate the zero voltage level using this function. Zero level calibration is required since different environment conditions (as temperature) or aging of the hardware components. When calibrating the zero position, please be careful not to input any signal, just let the channel input hang.



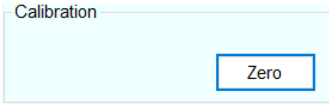
8.2.3.3 Self-calibration function

The above chapters have explained the accuracy of the equipment caused by factors such as ambient temperature and component aging. Our products also support Calibration correction function for zero position and voltage amplitude. Customers can calibrate the oscilloscope with their own standard source.



Click  will pop up the calibration window. Before calibrating, there are a few precautions:

1. Before calibrating the zero position, you need to turn on both channels and adjust the time base gear to the ms level, such as 0.5ms,1ms etc. The current zero position calibration does not support permanent storage after the user self-calibration. If you need to calibrate zero, you need to perform calibration when opening the device software. Note that when calibrating the zero position, do not connect any signal to the probe of the oscilloscope and keep it away from any interference.

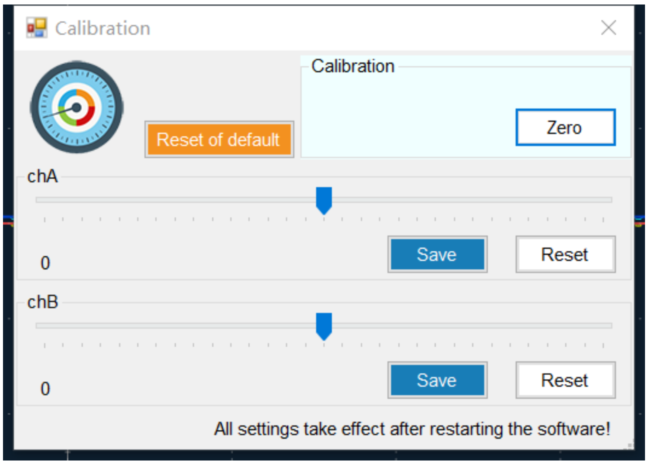


2. When performing voltage calibration, first adjust the voltage knob of the channel to be calibrated to the voltage range that needs to be calibrated, and then Click the calibration button to call up the calibration window, select the channel voltage slider to calibrate, after calibration, click the save button.

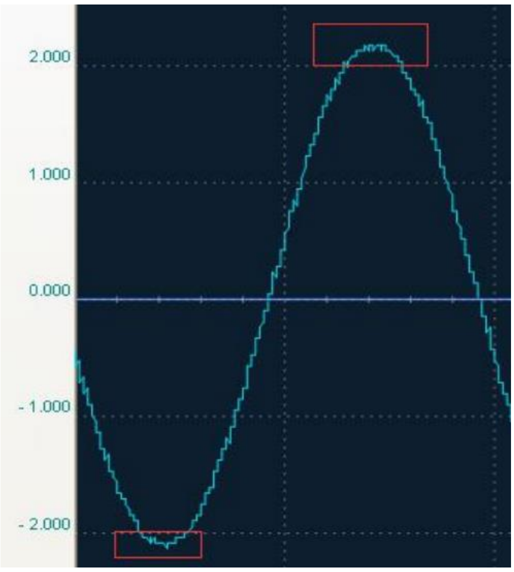
For example: We feel that the 1V voltage scale measurement value of channel chA is not accurate enough. The calibration steps are as follows:

a. Set the voltage wheel gear of channel chA to 1V;

b. Click the button  and the calibration window will pop up;

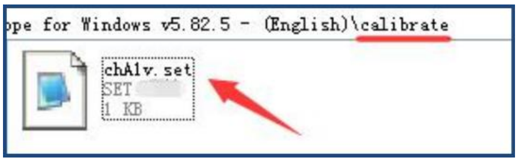


c. Connect the probe of channel chA to a signal source that we consider to be standard, and set the signal source to output a standard sine Wave, the amplitude is for example ±2V. false. Assume. The display of the oscilloscope is shown in the figure below: From the figure, we can see the waveform and peak Both the trough and the trough exceed the voltage grid line of ±2V.



d. Adjust the voltage calibration slider of chA to adjust the waveform to the ±2V position, and click the Save button to save.

e. At this time, there will be one more folder (calibrate) in the root directory of the oscilloscope software, and one more file in the folder (chA1v.set).



f. For other voltage range calibration steps, follow the steps described in a ~ e. After the calibration is completed, you need to close the oscilloscope software,restart the software, then the user calibration parameters of the voltage range will take effect.

If you want to restore the factory default calibration parameters of all voltage ranges, click Reset of default button, and restart the oscilloscope software.

8.2.4 Waveform display toolbar

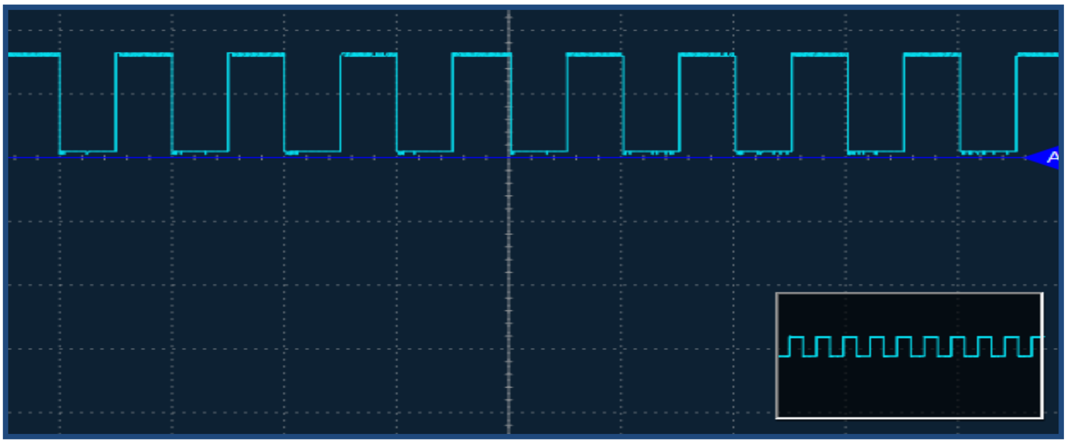
The display control functions can be accessed using the waveform display toolbar located on the right side of the waveform display area. When the mouse is placed over one of the buttons of this toolbar, a tooltip will show the name of the function corresponding to the button.

8.2.4.1 Default Display button

When  button is clicked, the waveform display area is restored to the default state, which is the original state according the knobs settings and without any zoom.

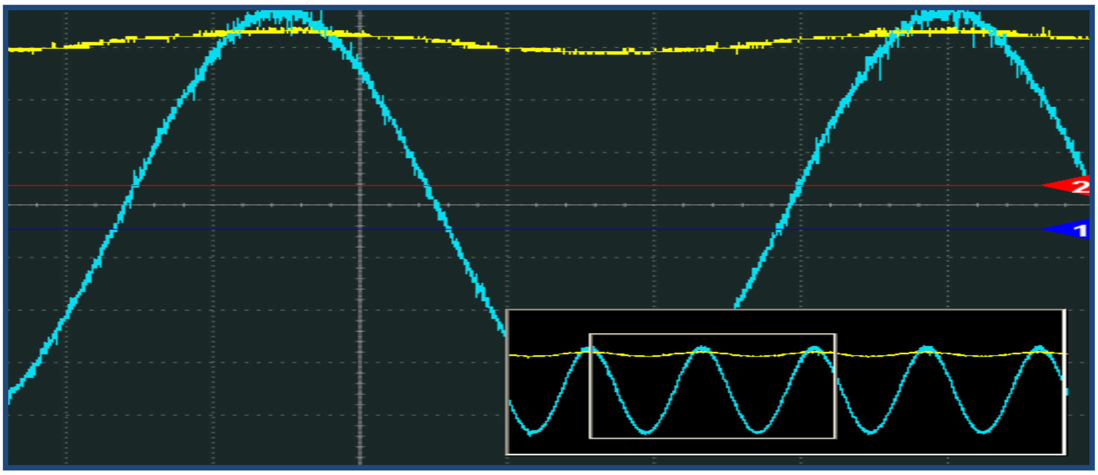
8.2.4.2 Zoom In button

When  button is clicked, the mouse cursor will become a cross and will allow to select an area for zooming. Press on the left button of the mouse to start the zoom area selection until the area of interest is no completely selected. Releasing the left button of the mouse, the selected area will be enlarged to the size of the full plot area.

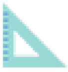


This operation can be repeated also in already zoomed areas. When this function is used a thumbnail appears in the lower right corner of the drawing area to allow to find the area currently selected and its position respect the full waveform area. This function will enlarge the horizontal and vertical coordinates simultaneously. If you want to zoom only according the horizontal axis and do not change the zoom level of the voltage axis, you can use the **X Axis Zoom** button, which will be introduced later.

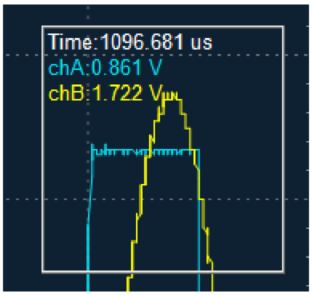
The **Default Display** button mentioned above allows to end the zoom-in state and to return to the original view state.



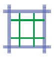
8.2.4.3 Ruler Measurement button

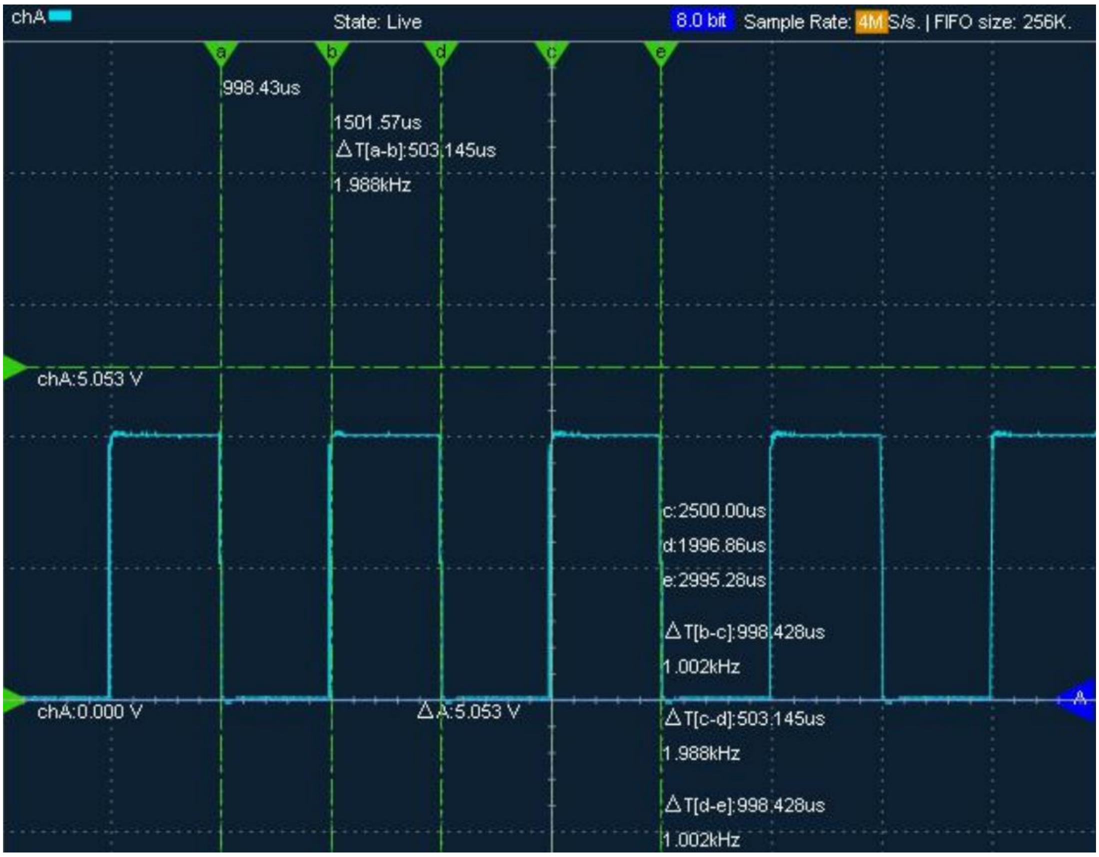
When  button is clicked, the mouse cursor will become a cross, allowing to select an area. Pressing the left button of the mouse the selection starts until the mouse button is not released. After the area has been selected the voltage and time span of the selected area will be displayed. Just one area can be selected with this tool so new selections overwrite the previous ones.

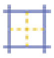
In white it is reported the time span. In the same color as each channel waveform is reported the voltage span selected of the corresponding channel. This measurement is convenient and mainly used for calculating time and voltage intervals.

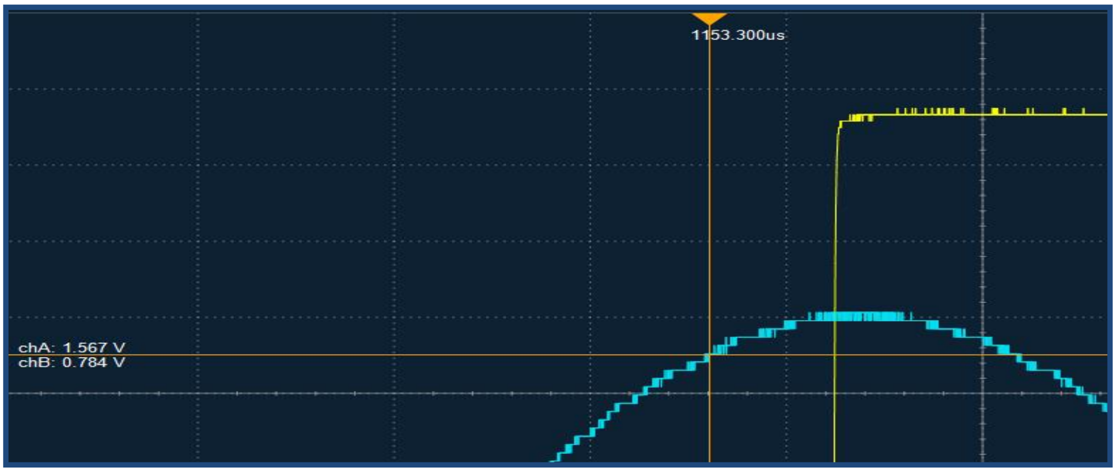


8.2.4.4 Markings / Cursor Measurement button

Clicking on  button, two green horizontal lines and vertical lines will appear in the display area. Dragging with the mouse the green triangle on the side of these marking lines, it is possible to adjust the position of the markings for data measurement.

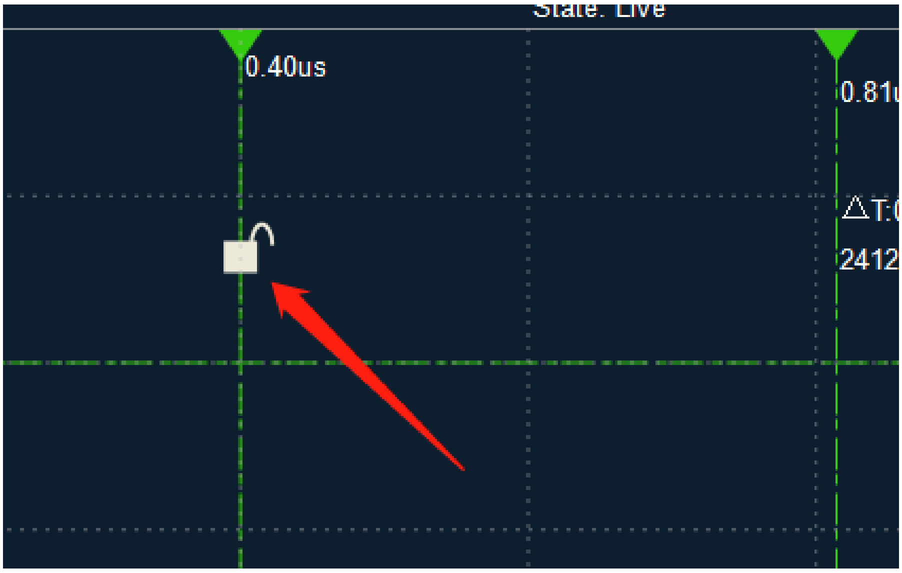


Clicking on  button, the marking measurement will become a cursor measurement. With this measurement an orange cross will appear in the drawing area following the mouse position and displaying the horizontal and vertical coordinate data of the cursor position.

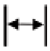


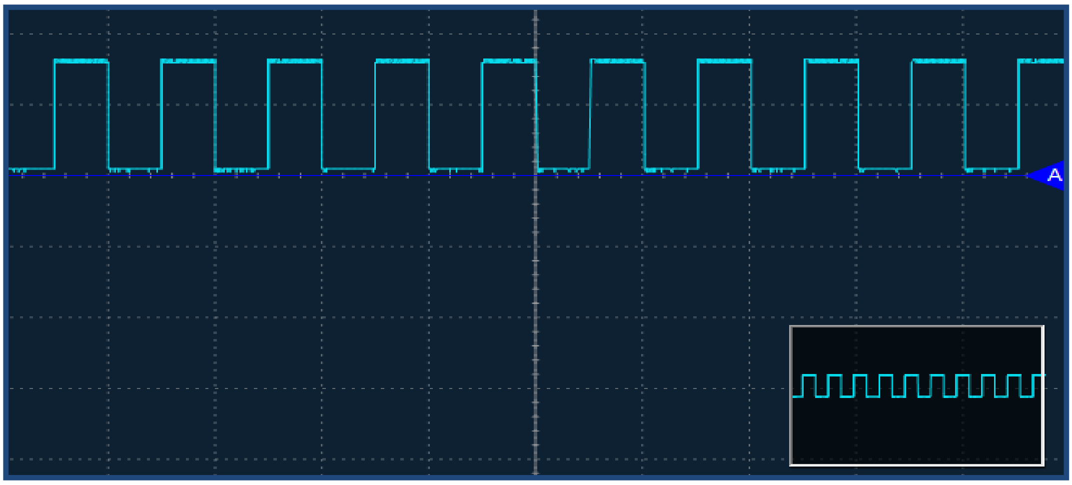
These two measurement methods are mutually exclusive so that only one kind can be chosen at time. The Marking Measurement is more suitable for waveforms. The Cursor Measurement is more suitable for punctual measurement on certain points of the waveform.

When paused, a lock icon will appear on the vertical marking. Click the lock icon to lock the vertical marking. After locking the vertical marking, you can use the paused buffer slider to move left and right to measure the data in the buffer.

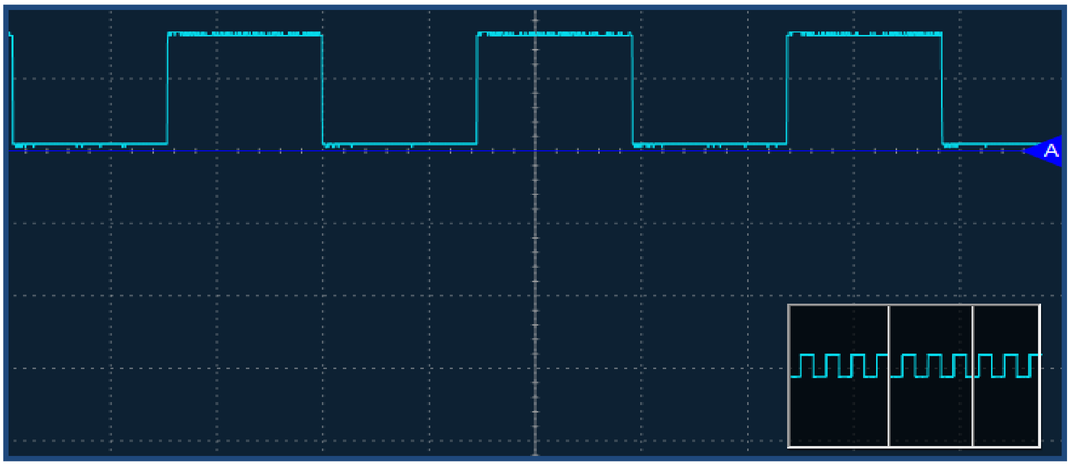


8.2.4.5 X Axis Zoom button

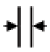
The  button zoom the display area along the X axis for a better observation of the period and frequency of the signal. When this button is clicked, a little window with the full waveform will appear in the lower right corner.



In the display area, select the area to be enlarged by dragging the mouse (the zooming operation is limited only to the X axis). The selected area will cover the entire display interface. In the little windows at the lower right corner, two white lines identify the current visualized area of the waveform.



8.2.4.6 Zoom Out button

The  button is enabled only when the waveform has been zoomed in. By clicking on it the visualized waveform can be continuously reduced by clicking on the visualized waveform until it is drawn fully on the screen.

8.2.4.7 Save button

The  button saves the data collected by the oscilloscope in the user's computer in plain text format (.txt). The destination path can be chosen during the process. The detailed function of this button is described in detail in 8.2.32.

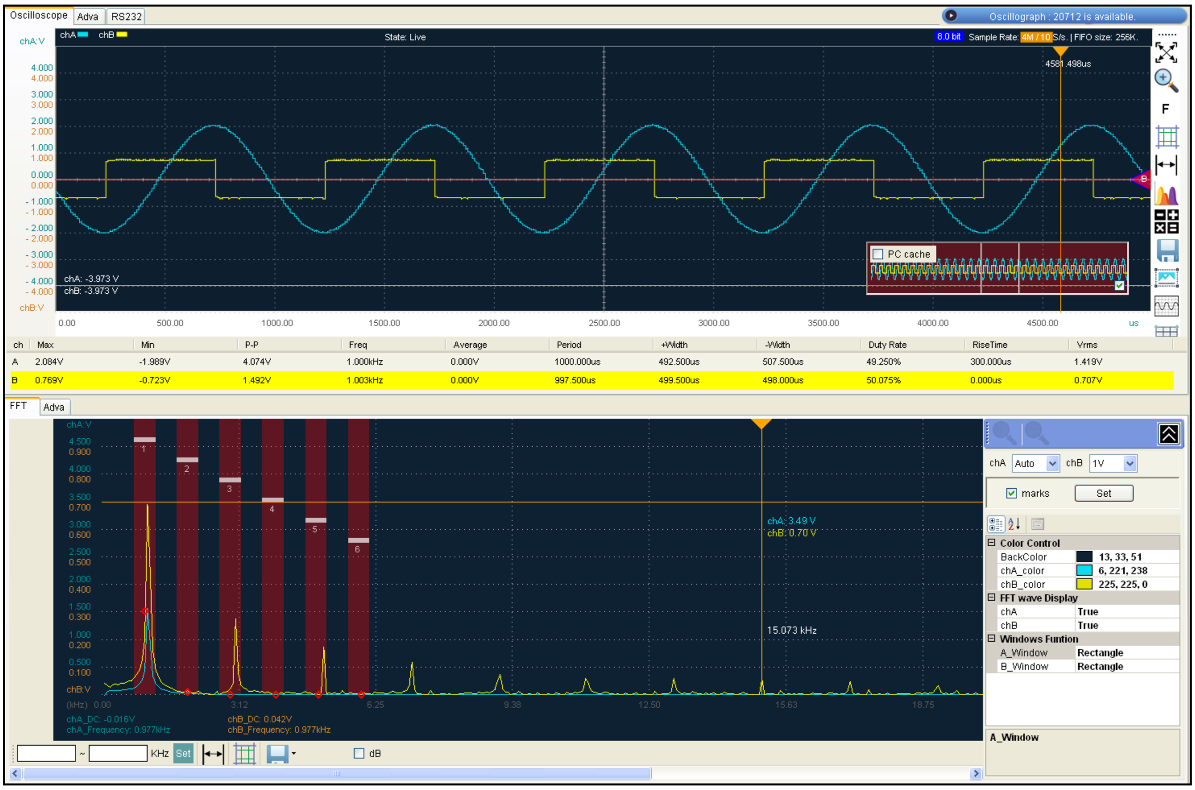
8.2.4.8 Screenshot button

The button takes a screenshot of the current screen and save it as a picture for later analysis and viewing. You can select the destination path, file name and the file format among the followings: .jpg, .bmp, and .gif (shortcut key (Ctrl + P) - screen-shots).

8.2.4.9 FFT button

The button shows the Fourier Transform of the original measured data coming from the enabled channels, to allow the frequency analysis of the signals.

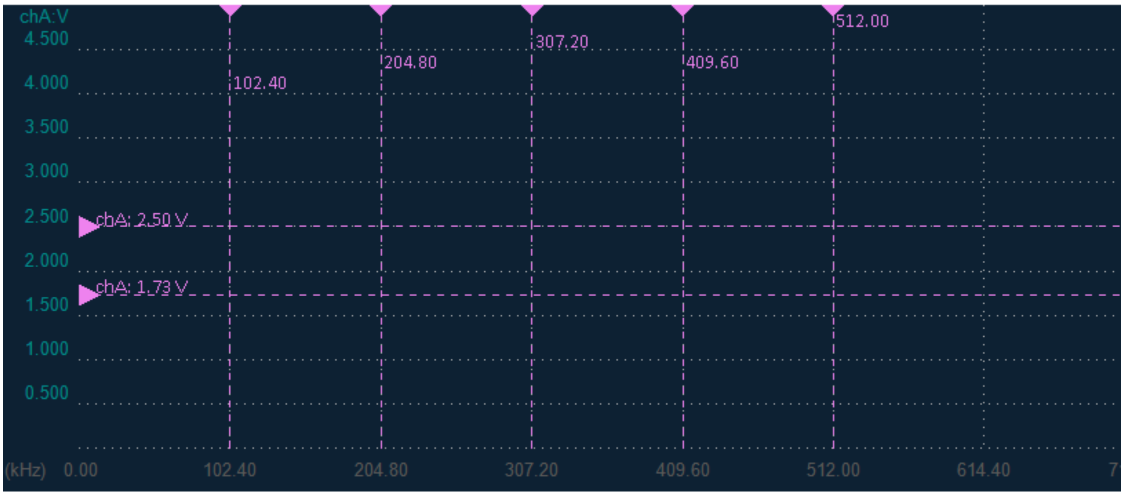
- Analyze the frequency domain of the signal and support dB display.
- Support mouse frame selection or input some frequency bands for display.
- In advanced settings, it supports specified fundamental and harmonic analysis, harmonic comparison analysis, and waveform distortion THD detection.
- Supports the setting of fundamental wave and harmonic rules, and alarms for waveforms that do not meet user-defined rules. The alarm supports software alarms and IO port alarms. Suitable for application scenarios such as audio analysis.

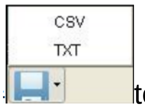


- Enter the frequency range in the text box and click the set button to specify the frequency range and select the frequency band to display the observation frequency.

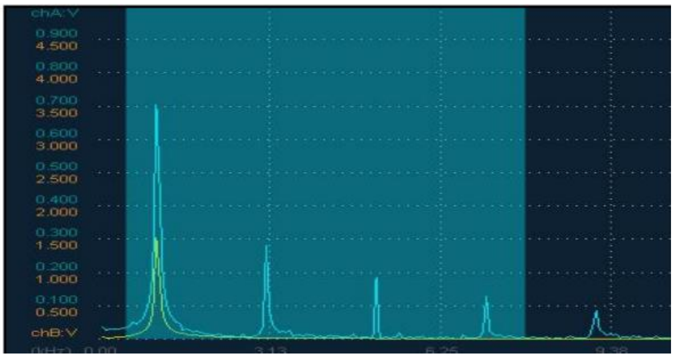
◆ Click the button  to open several frequency domain FFT measurement lines to facilitate the observation of the harmonic frequency value. Click again to restore and close the measurement line.

As shown in the figure: Some models such as: OSC980 have the measuring line function enabled by default.

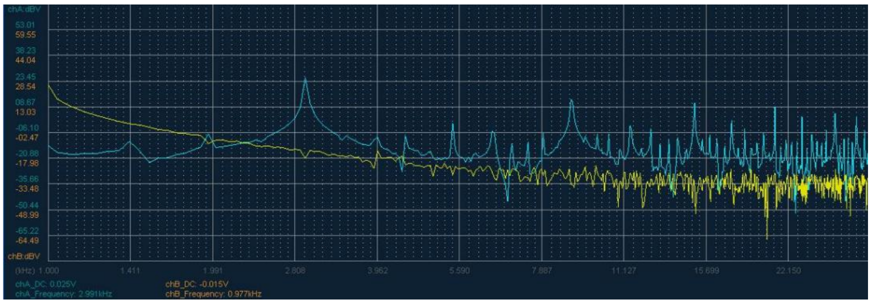


◆ Click the Save Export button  to export the FFT data (frequency and voltage value) as a file, supporting CSV and TXT formats.

◆ After clicking the button , you can use the left mouse button to select an area in the FFT waveform display area, and perform the mouse selection to observe and display. Click  to restore the FFT default display. Note: When using the dB mode to display FFT, you need to restore the FFT default display. The dB mode currently does not support the mouse to select the frequency band display.

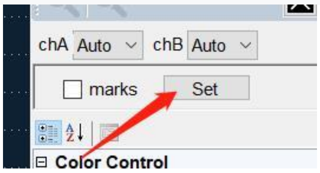


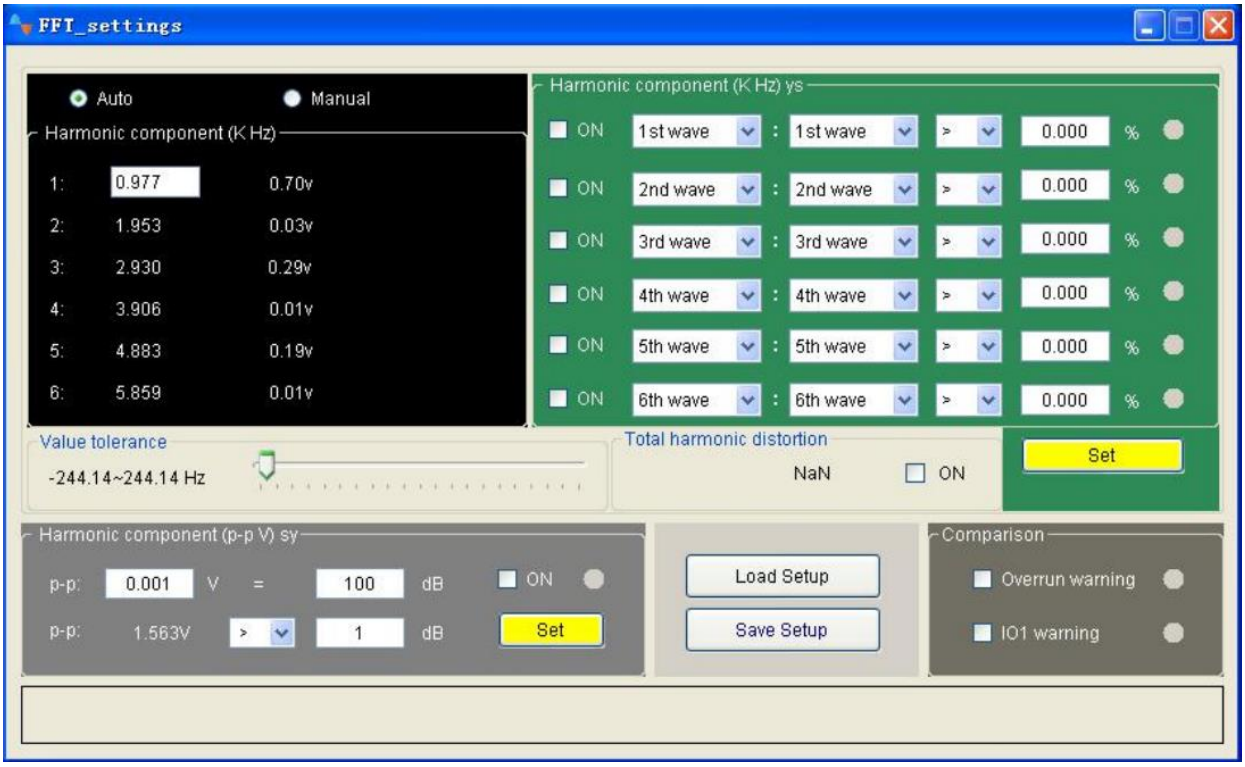
- Check the dB checkbox, FFT will display frequency domain waveform in dB. Note: The dB mode display does not support the specified frequency domain. If the FFT frequency domain was specified in the previous operation, you need to restore the FFT default display.



- FFT Advanced settings

Click the Set button to call out the FFT advanced window:





1. Select Auto mode, the oscilloscope will automatically identify the fundamental wave of the waveform, and display up to the 6th harmonic. Select Manual mode, the user enters the frequency as the fundamental wave in the text box of the 1st wave, and the software calculates and displays up to 6 harmonics based on the fundamental wave input by the user.

2. Value tolerance slide bar, set the software to find a wide range for each harmonic. Setting too narrow or too wide will affect the precise positioning of the harmonic peak search.

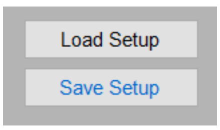
3. The Total harmonic distortion option is used to measure the THD of waveform distortion, mainly for sine waves.

4. With the Harmonic component (p-p V) function, the user can specify the dB value corresponding to the peak-to-peak value (p-p) of the waveform, and set the peak-to-peak comparison rule. When the rule is not met, it will alarm. The alarm supports software alarm and IO port alarm.

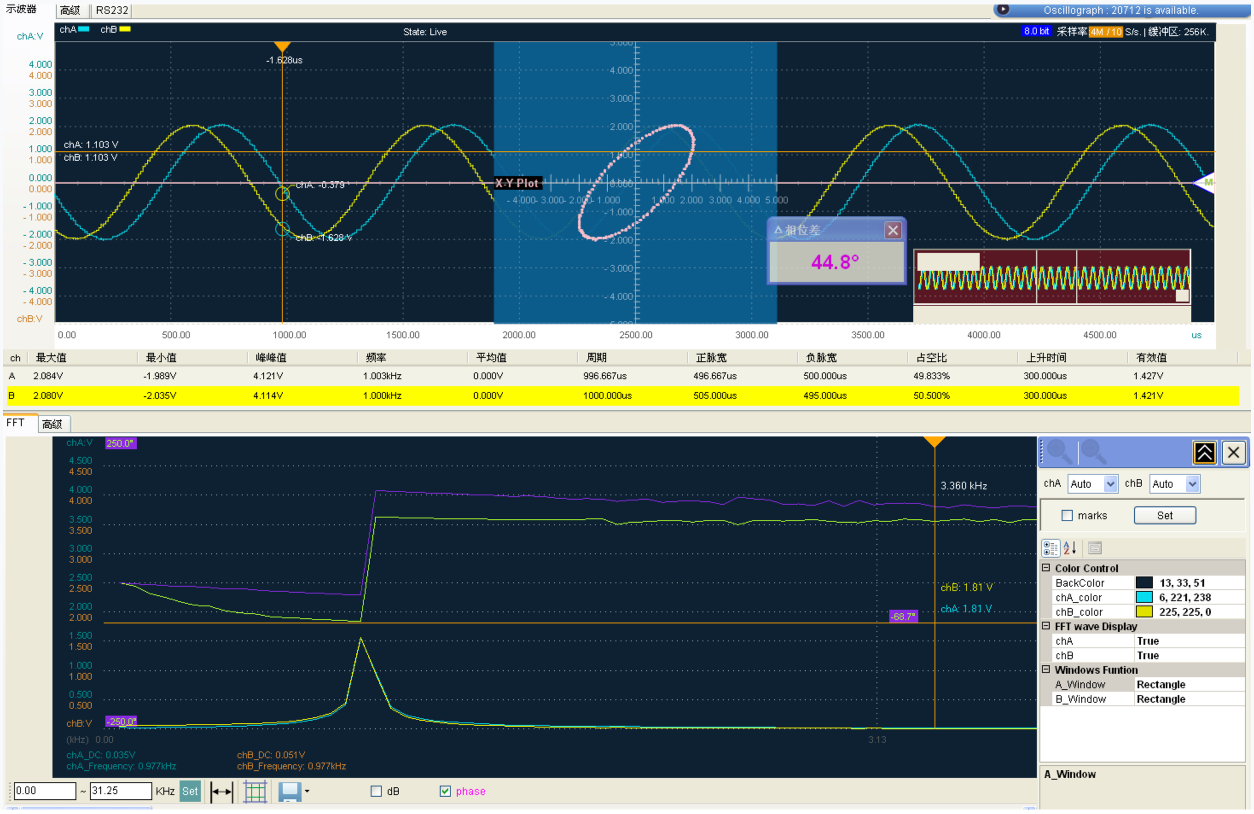
5. Harmonic component (K Hz), the user can specify the comparison rules between different harmonics, and alarm when the rules are not met. The alarm supports software alarm and IO port alarm.

6. The Comparison function displays the final result of the comparison between Harmonic component and Harmonic component rules. As long as one does not meet the rules, the final result is displayed in red, and all qualified are displayed in green.

7. Load the button to save the historical setting parameters. The user can save the set FFT harmonic comparison rules into a file, use the Load Setup button to browse the file to load the historical rules, and use the Save Setup button to save the settings into a file.



8. As shown in the figure below, the FFT function also supports the phase-frequency curve display, which can visually display the phase-frequency curve of the waveform participating in the FFT.

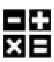


8.2.4.9.1 Video demonstration：

<LOTO Practice Essence (7) ||Oscilloscope for FFT spectrum analysis>
https://www.youtube.com/watch?v=ATlmAfjonCo&t=138s

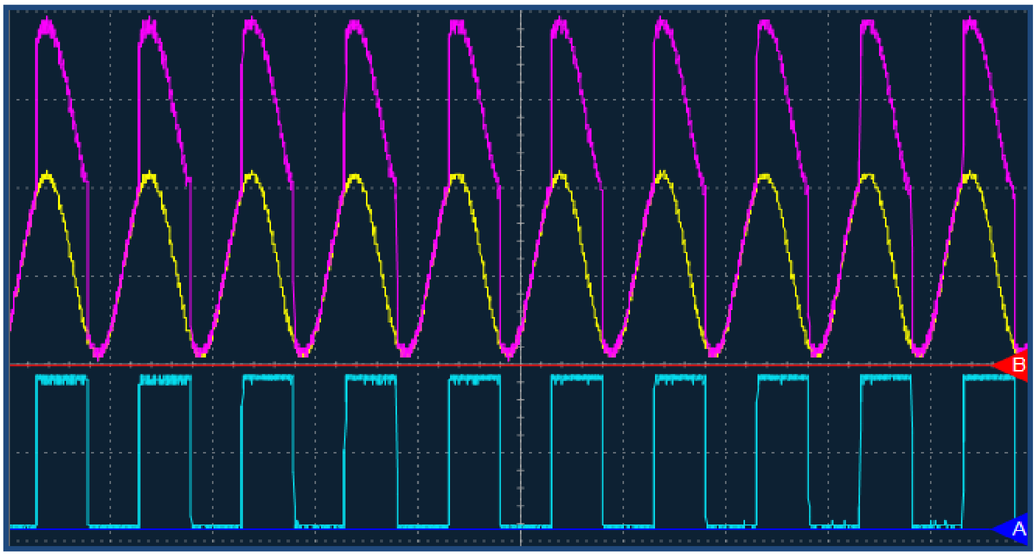
<loto oscilloscope__3D spectrum FFT function>
https://www.youtube.com/watch?v=D_wjgfClnoQ

8.2.4.10 Math Operations button

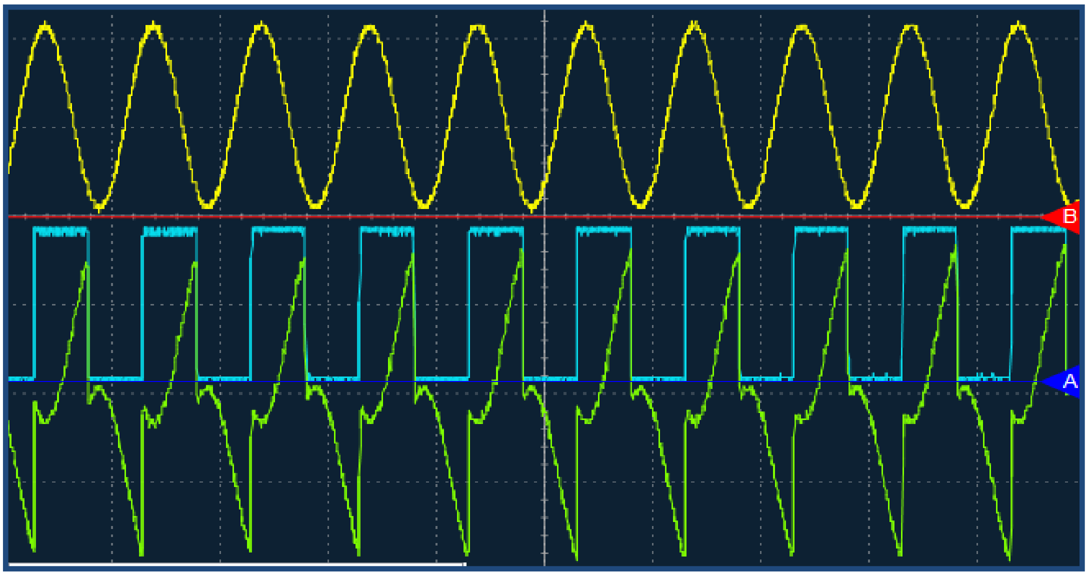
The  button performs mathematical operations between the signals from channel A and channel B like addition, subtraction, multiplication, or XY plotting. Once clicked on this button, you can select the desired operation:

A + B
A - B
A x B
A_B) X_Y)
A·B

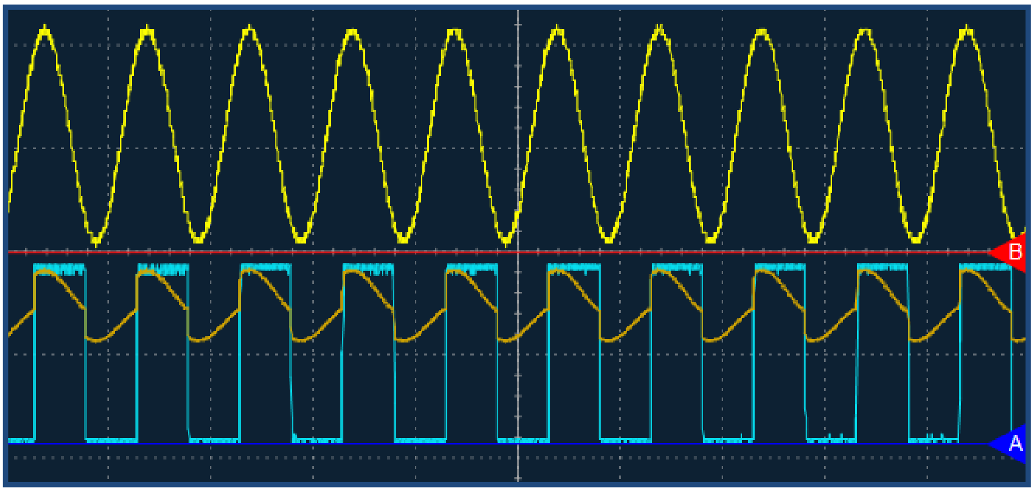
"A+B" means that the channel A waveform is added to the channel B waveform. Selecting this operation, the display area will show both channel A and channel B as stacked waveforms and the waveform resulting from the operation in purple color.



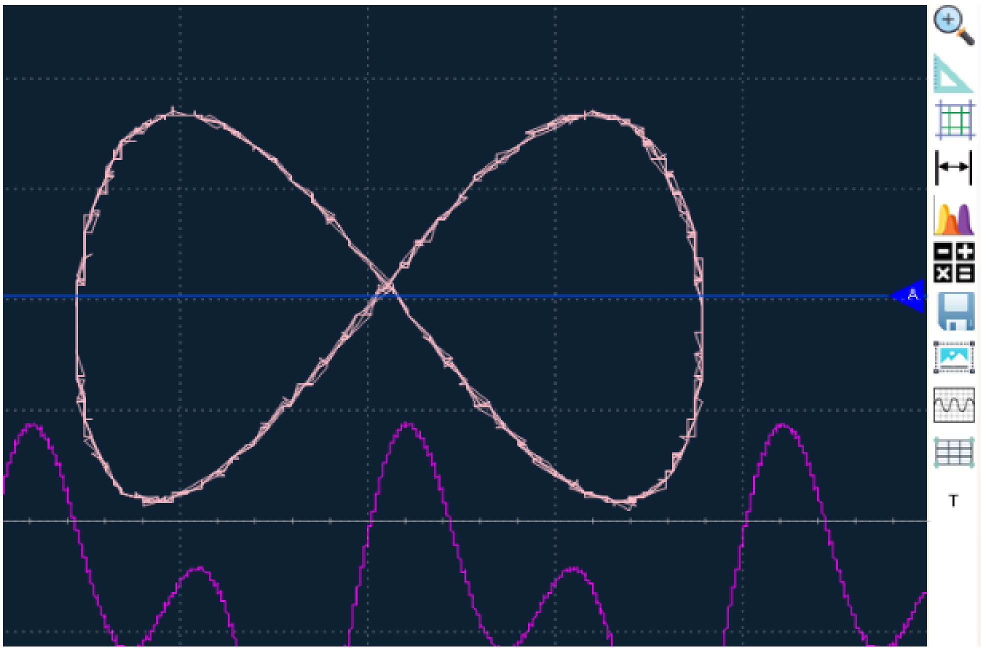
"A-B" is used to realize the subtraction between channel A and channel B signals. In similar way as in the addition both channels' waveforms are displayed together with the result of the subtraction, but this time in green color.



The "AxB" operation is used to multiply channel A and the channel B signals. The result is shown in the orange color.

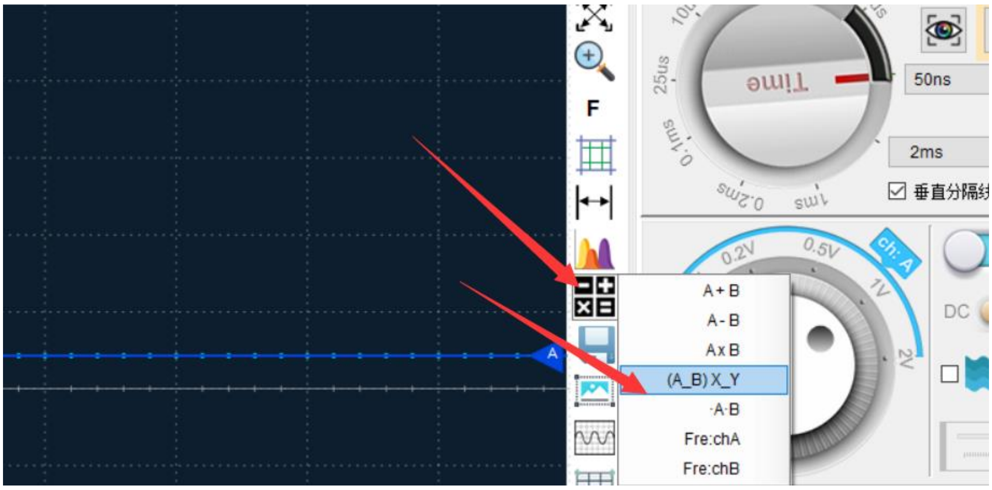


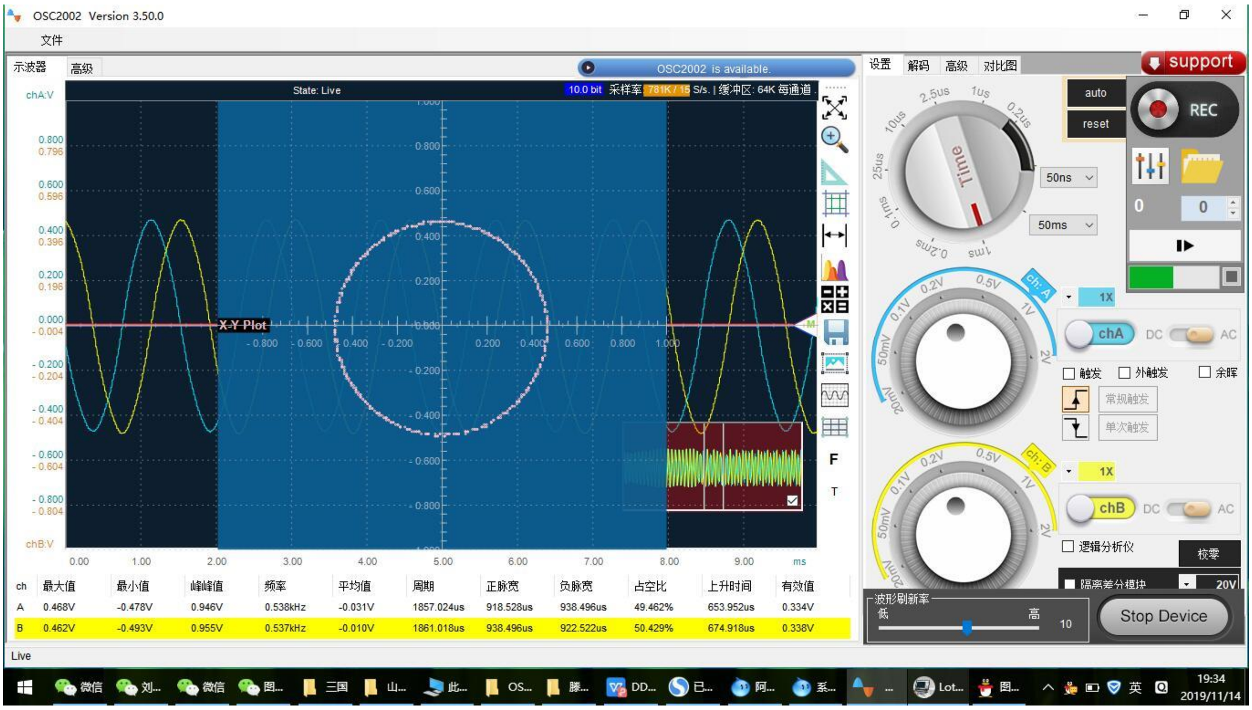
The "·A·B" button is used to hide the signal of the AB channel. After hiding, only the results of the A and B mathematical operations are displayed, as shown in the following figure:



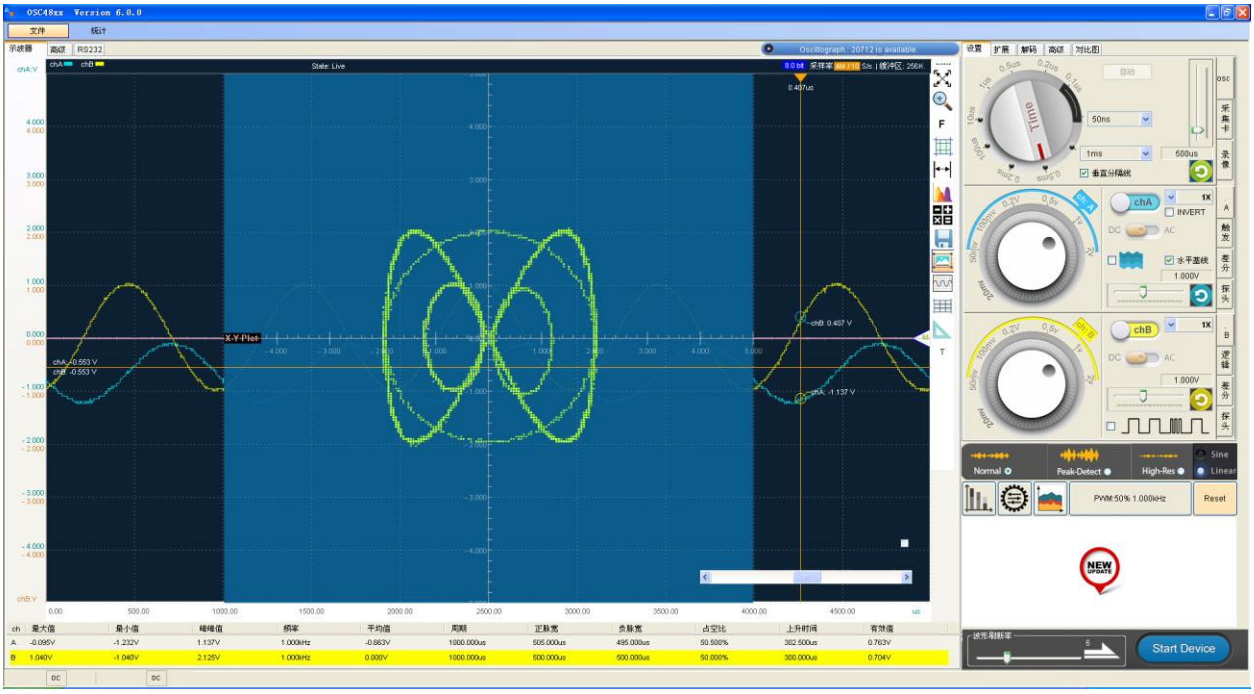
8.2.4.10.1 "Lissajous"/ X_Y drawing

XY Plot or “Lissajous curves” plot. This display mode plots one channel against another on the screen. One channel is plotted on X-axis, the other channel is plotted on Y-axis, together with the information of the measured frequency of each signal. From the resulting curve it is possible to calculate the frequency and phase relation of the signals.



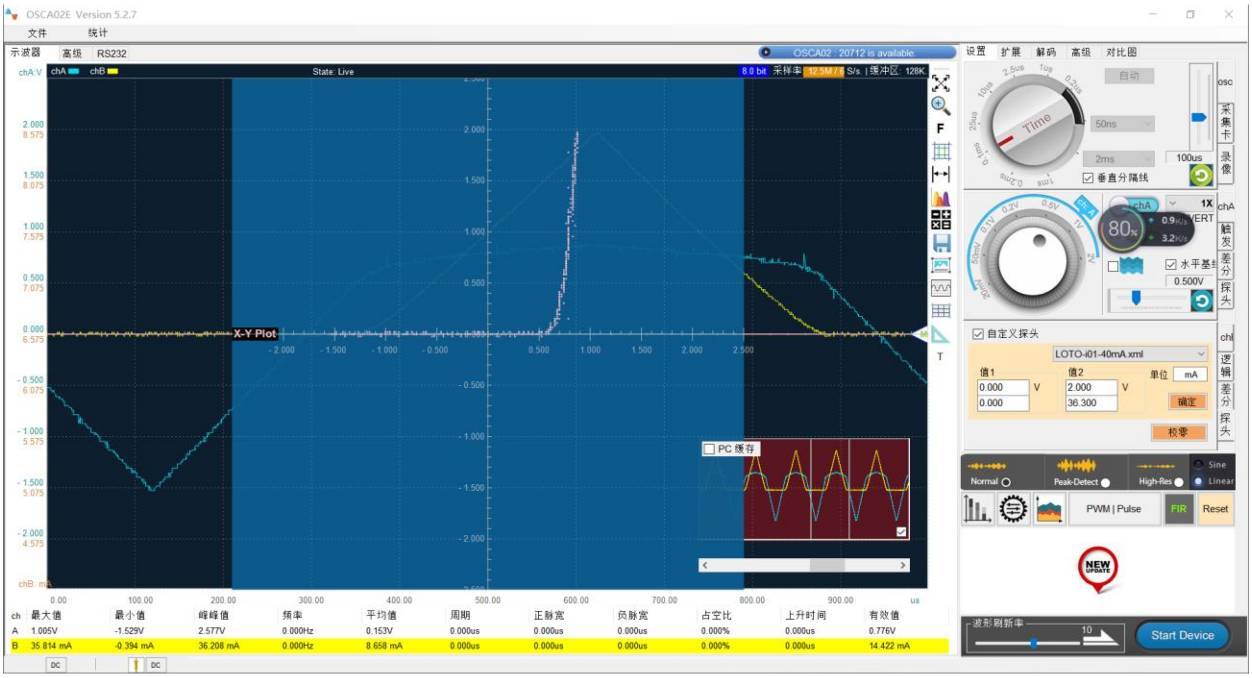


Through the "Lissajous figure", the relationship between the frequency and phase of the two measured signals can be displayed intuitively.



8.2.4.10.2 Diode UI characteristic curve drawing

Through the "Lissajous graph", the UI characteristic curve of the diode can also be measured. We connect the scanning voltage of the diode to channel A, and connect the current flowing through the diode to channel B using LOTO's mA current module, and then we can directly The UI characteristic curve of the measured diode is shown in the figure below:



The picture above shows the UI curve of a normal diode. If the diode is damaged as an open circuit, the curve will be a horizontal line, and if it is short-circuited, the curve will be a vertical line.

8.2.4.10.2.1 Video demonstration：

<LOTO oscilloscope: Measurement of UI characteristic curve (diode as an example)>

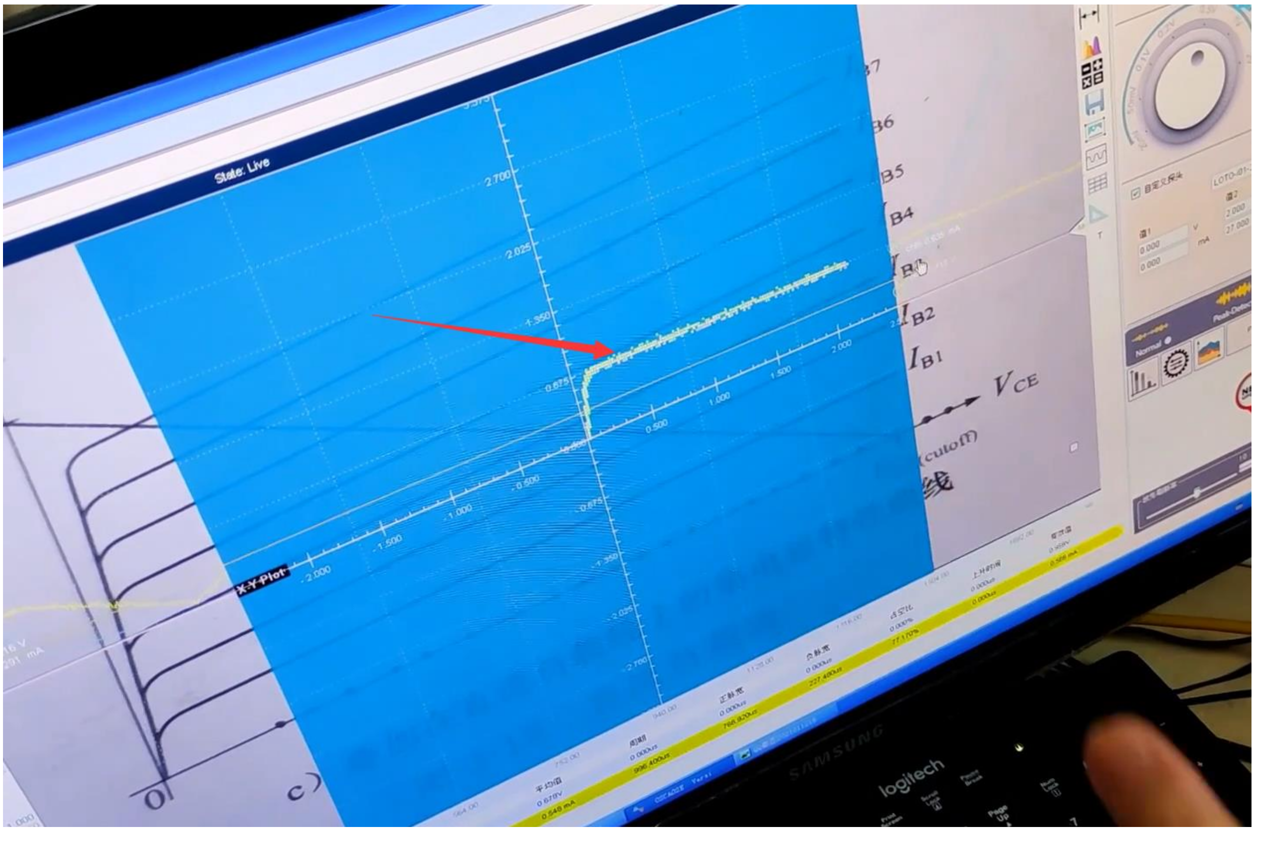
https://www.youtube.com/watch?v=bF-bpaadAp4

8.2.4.10.3 UI curve drawing of triode

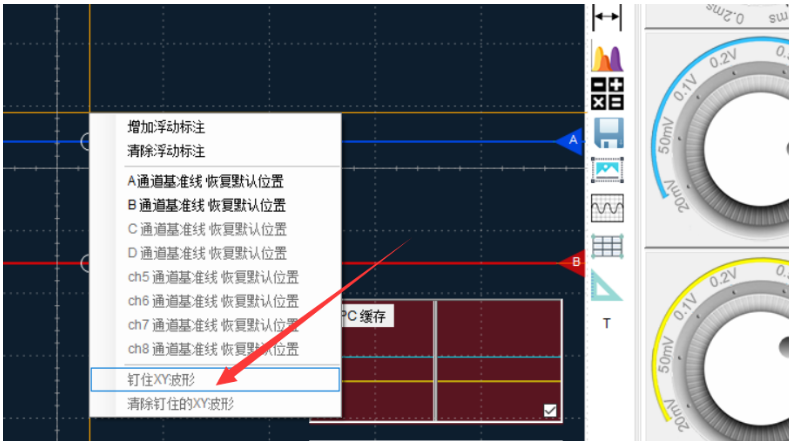
Through the "Lissajous figure", the UI characteristic curve of the triode can also be measured.

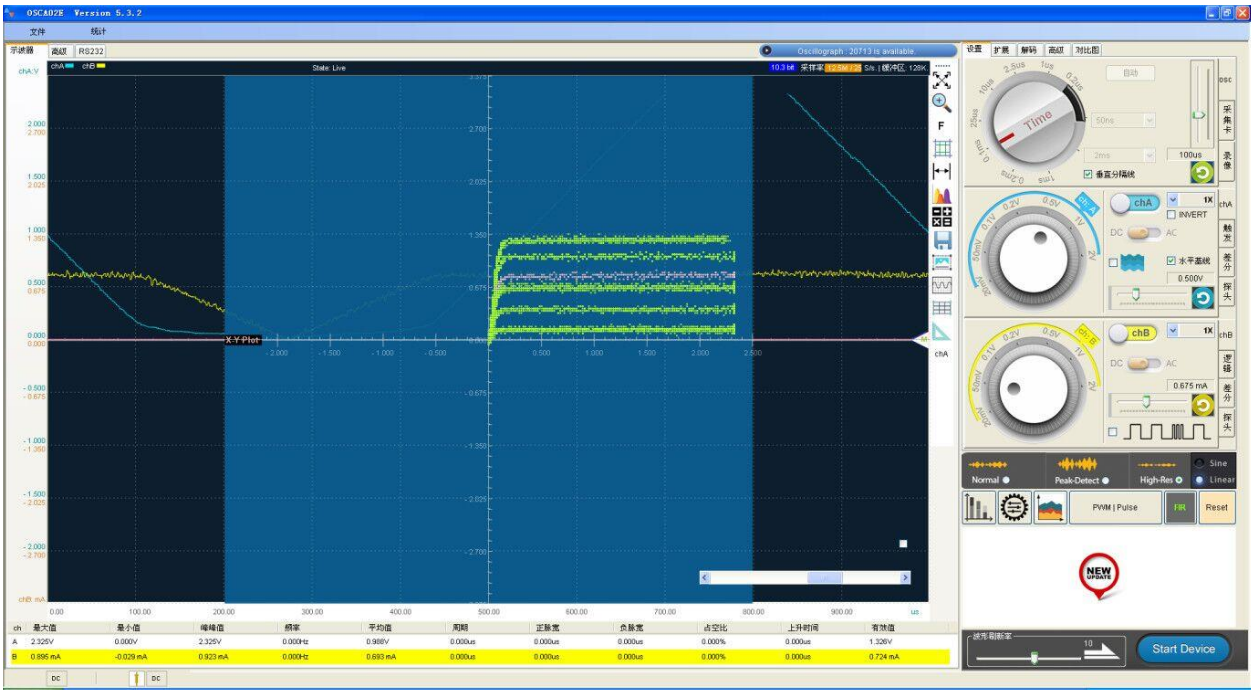
The same as the UI curve of the diode, we connect the scanning voltage UCE of the triode to channel A, and use the current IC of the corresponding triode flowing through the C pole to use LOTO.

When the mA current module is connected to channel B, the UI characteristic curve of the triode can be directly measured under a certain fixed base current IB.



Different from diodes, triodes need to measure multiple sets of such UI curves under different base current IB and display them on the graph at the same time, so we have a right-click function, which is to obtain a UI curve after changing IB every time, and then select Keep this curve fixed on the interface, as shown in the figure below:



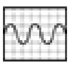


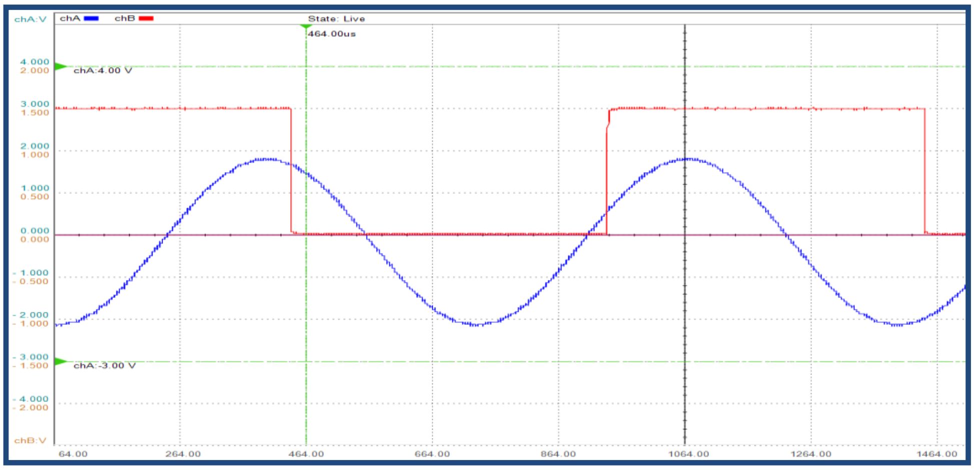
8.2.4.10.3.1 Video demonstration：

<LOTO lesson 6: Mapping of the output characteristic curve of a triode>

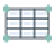
https://www.youtube.com/watch?v=1XEE0YSo32o

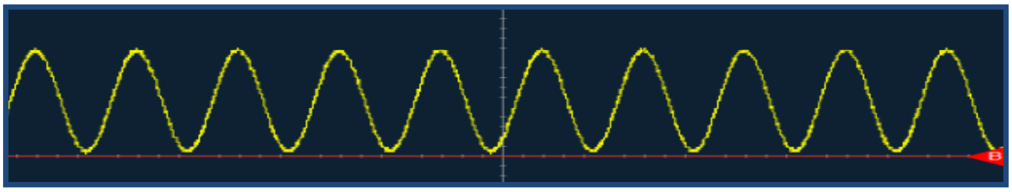
8.2.4.11 Colors Invert button

The  button switches the background color from dark to white and the other way around. The dark background is good for long-term observations to alleviate the eye fatigue. The white background is convenient for taking screen shots for making reports or for further image processing.

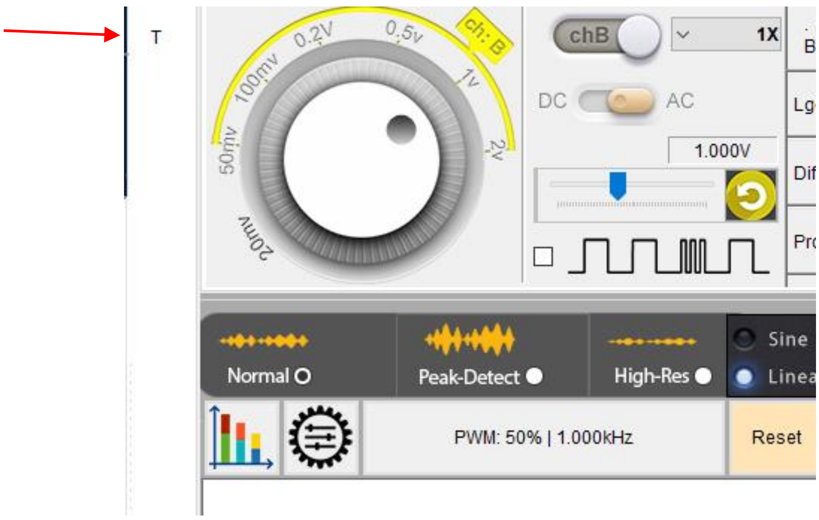


8.2.4.12 Grid Selection button

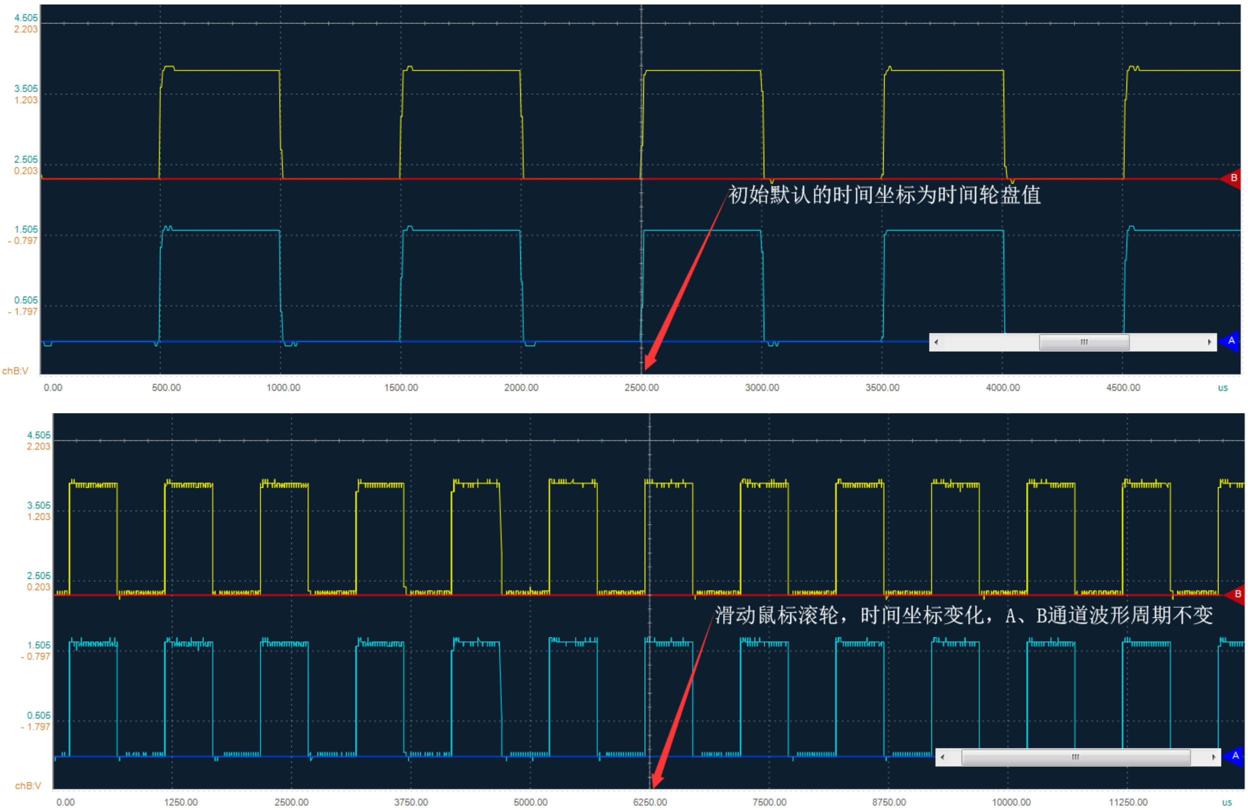
The  button hide or show again the grid displayed in the waveform area.



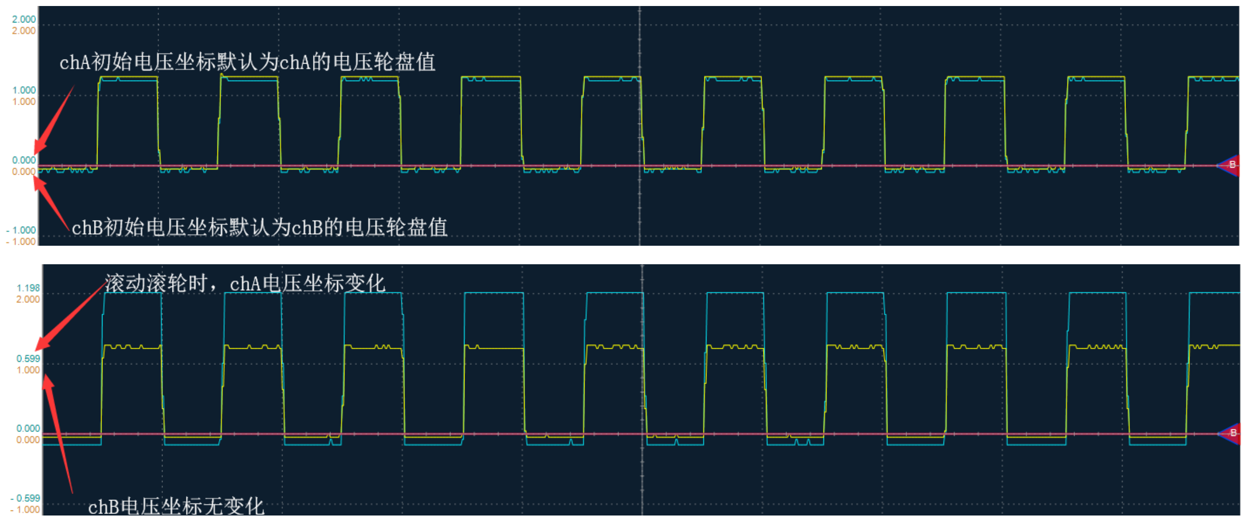
8.2.4.13 Mouse wheel Selection button “ T “



The “ T “ button determines which axis will be control by the mouse wheel when the cursor is located in the waveform area. When this button shows “T”, we scroll the mouse wheel will give the time axis a fine adjustment to zoom in or out the waveform horizontally.



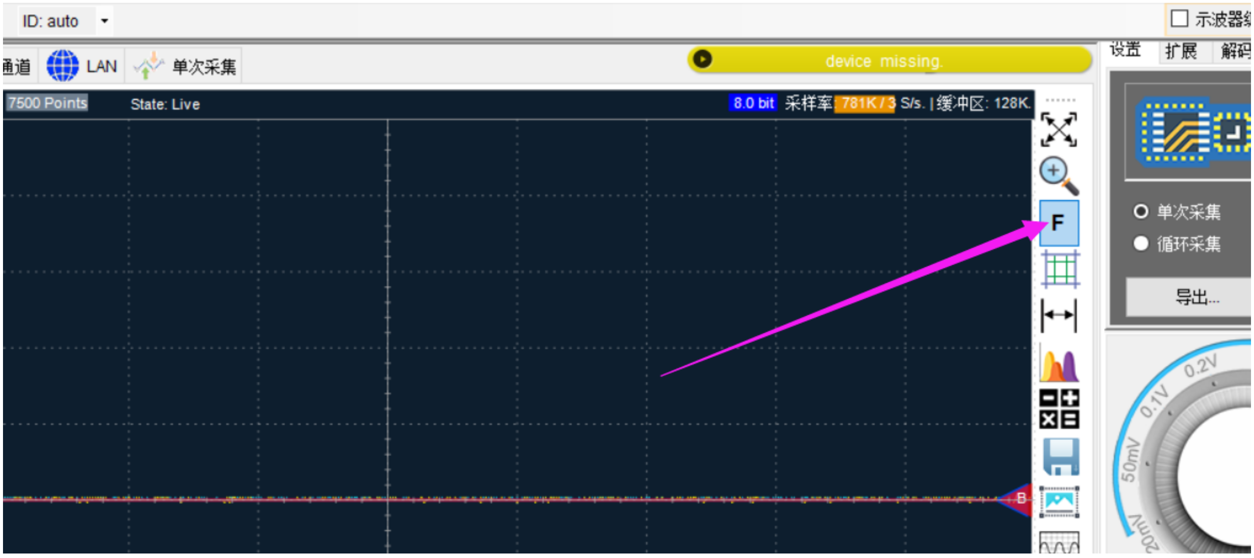
By clicking “T”, this button will show “chA”, and another clicking will show a “chB”, which means the mouse wheel scrolling will lead to a fine adjustment to zoom chA or chB vertically.



Also, we can do this with three controls in the right lower corner as bellow. However, we may not see these controls when running on a PC with a small screen of screen resolution below 1280*1024.

8.2.4.14 "F" waveform area full screen display menu item

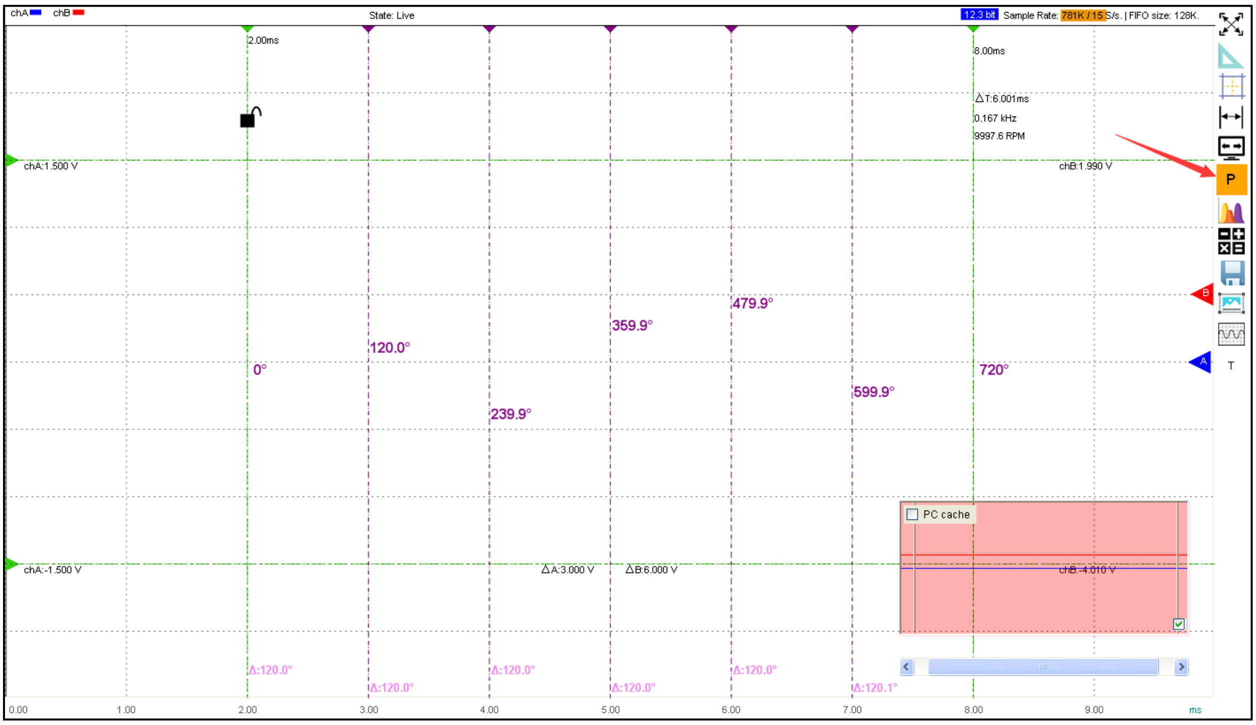
Click the "F" button in the menu to change the waveform display area to full screen display .Click the button again to restore the default.



8.2.4.15 " P ", 720°Phase measurement

Click the "P" button in the menu, and a 720° phase measurement line will appear in the waveform display area. Click the button again to restore the default.

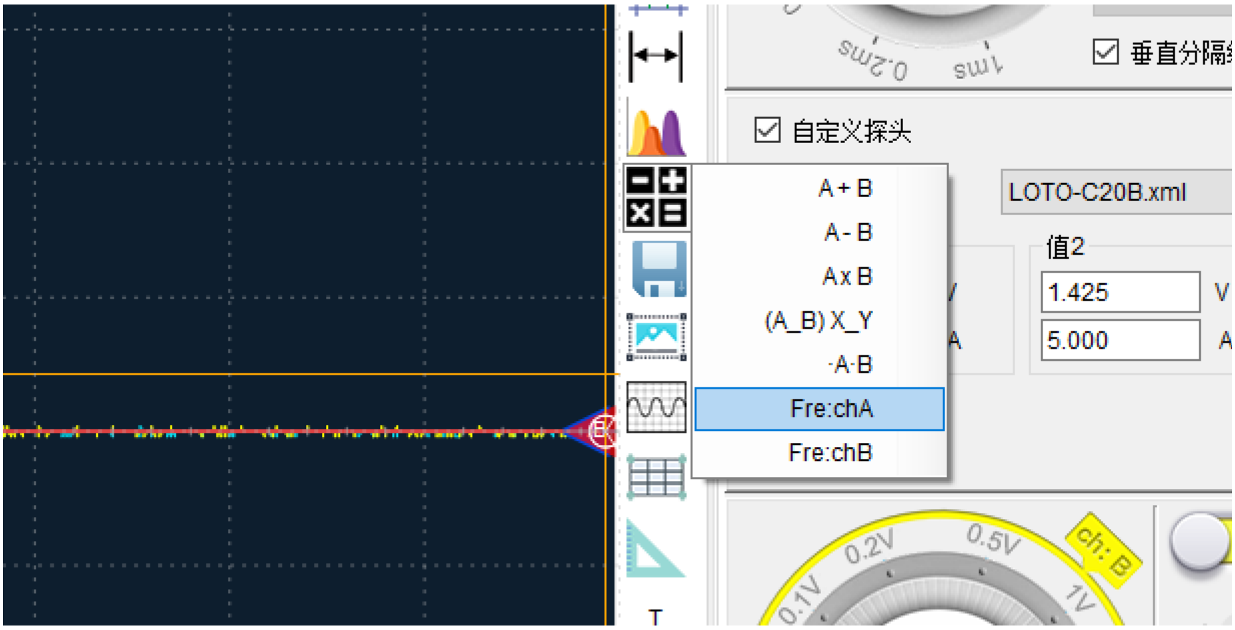
This function currently only appears in some models of software, which is convenient for the measurement needs of industries such as automobile maintenance.



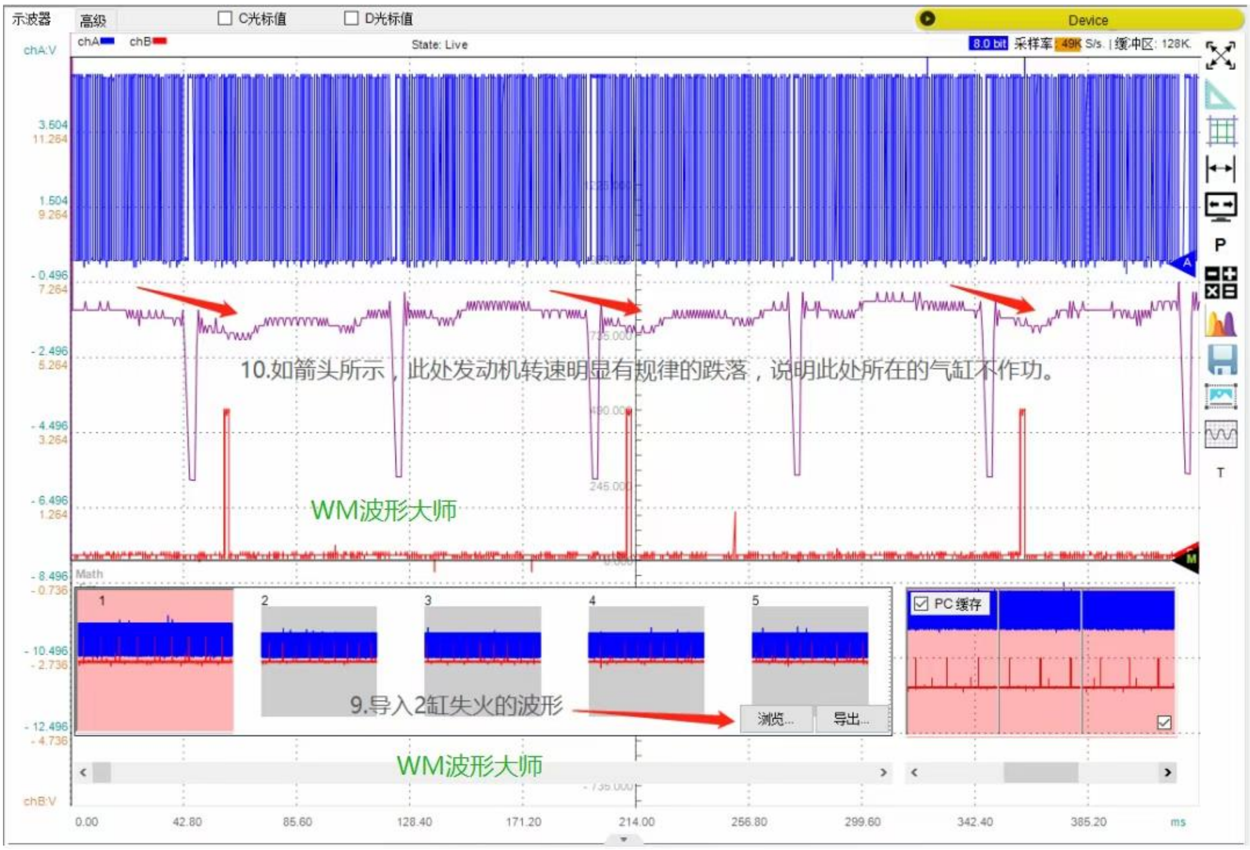
8.2.4.16 Speed/frequency change curve function

In some occasions, we need to observe the trend of frequency or rotational speed represented by the dense waveform on the screen to see if it is stable. At this time, the statistical curve function that only rents on a large time scale will not be able to do it. At this time, the speed/frequency change curve function mentioned here is needed.

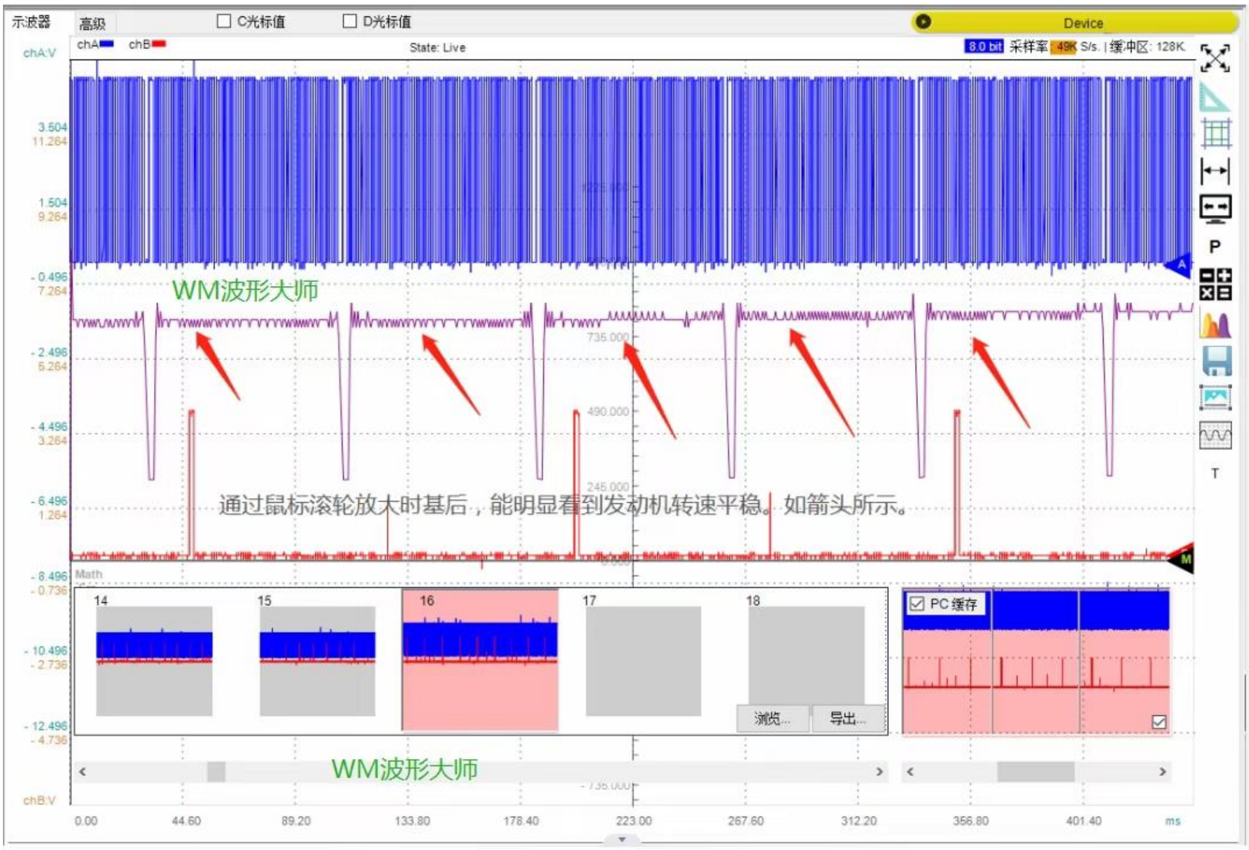
This function is in the "waveform calculation" button mentioned in 8.2.4.10, which can display the frequency/speed curve waveform of the waveform of channel A or channel B:



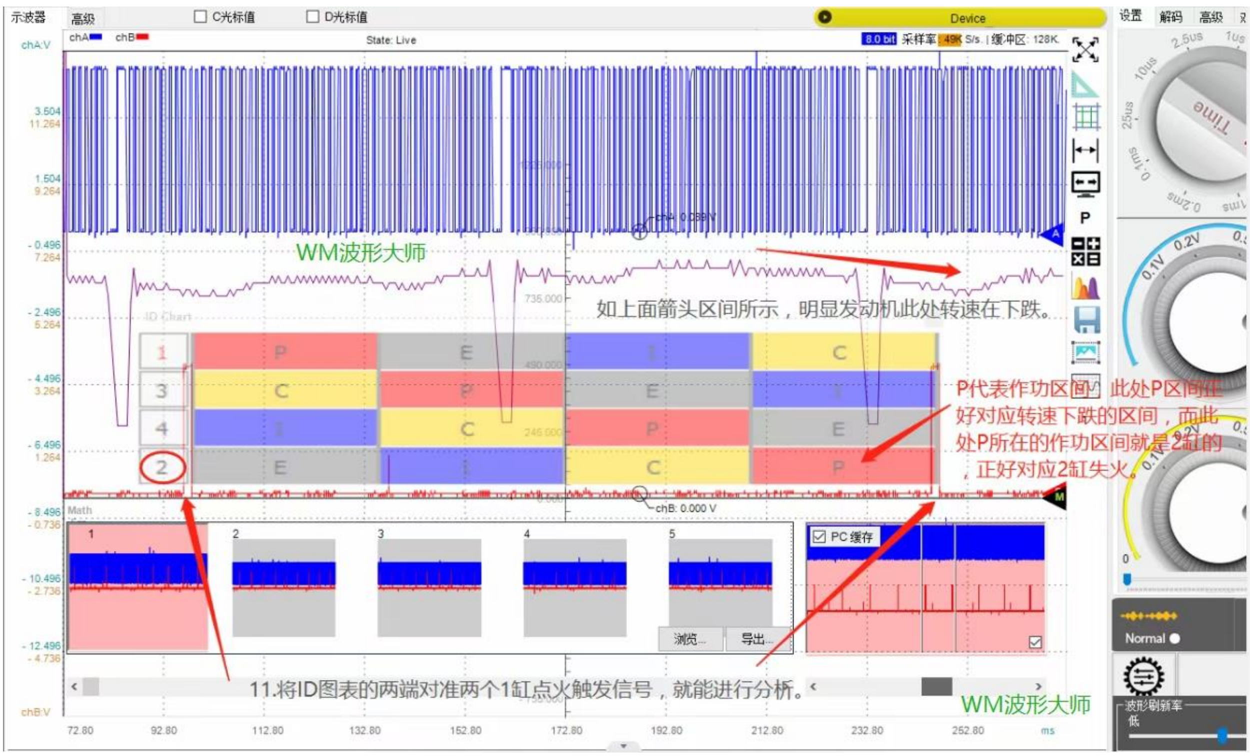
Let's take the OSC984 host computer software as an example. The blue waveform of channel A in the figure below is the crankshaft waveform of Baojun Automobile. With such a dense waveform, we cannot see whether the speed is stable with the naked eye. When we open the speed curve, as shown in the figure As shown by the purple waveform in , we can see that the speed drops intermittently.



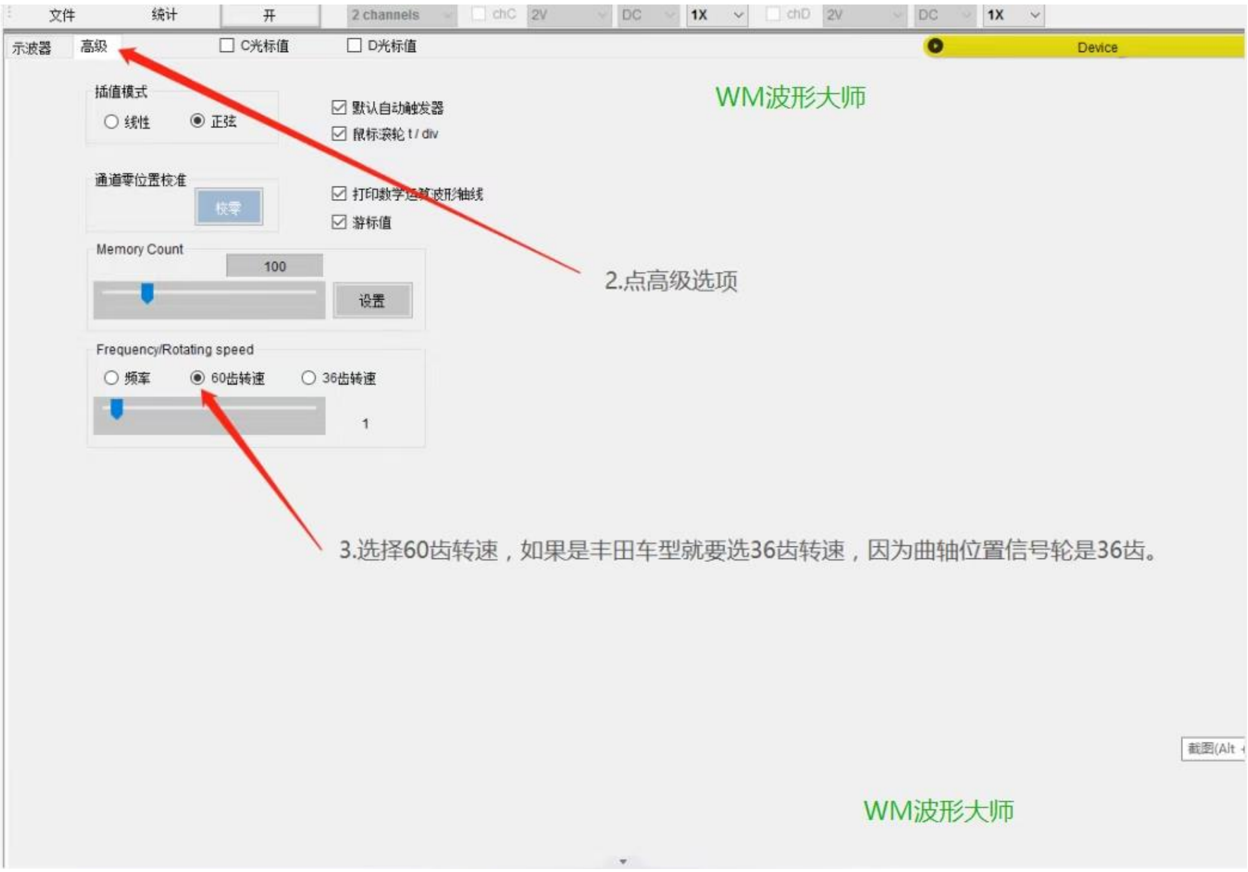
We collected another crankshaft waveform and compared it, we can clearly see that the speed of the following waveform is very stable, and there is no obvious speed drop, as shown in the figure below:



For this example automotive waveform diagnostic application, we can analyze the problem represented by the waveform:



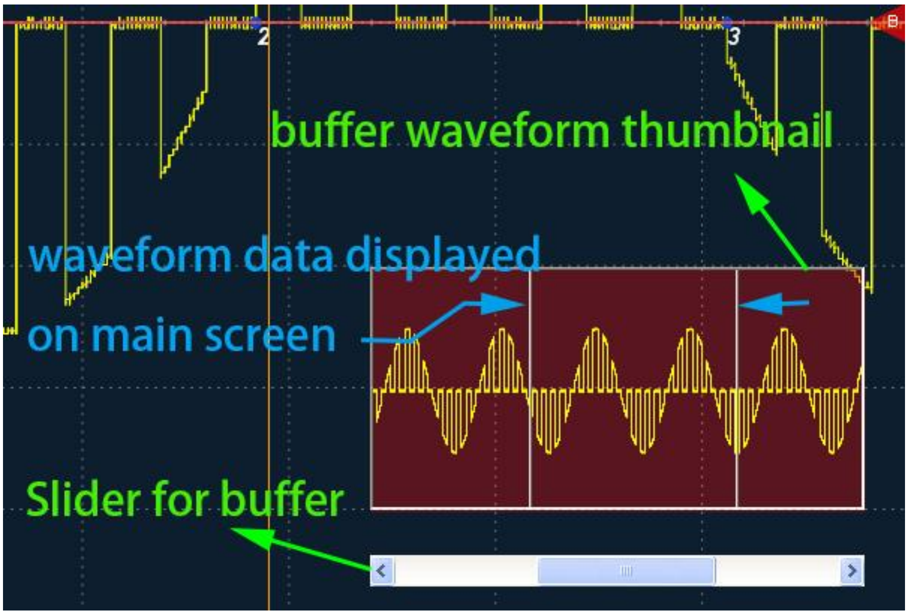
In the "Advanced" tab page, we can set the frequency of the coordinate axis of the speed-frequency curve or which speed to use, and also set the magnification of this curve, which is convenient for intuitive viewing of fluctuations:



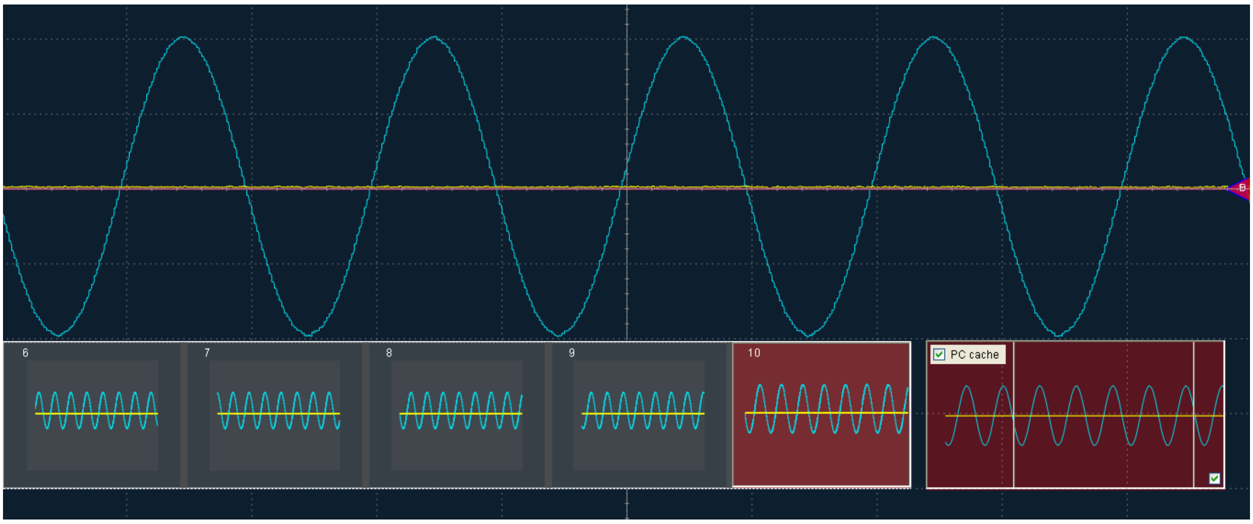
8.2.5 Waveform buffer(PC cache) and left/right shift

When the acquisition is paused, a waveform thumbnail as well as its slider of the whole memory buffer appears in the lower right corner of the waveform area. This slider shows the ratio and the position of the waveform data currently displayed on the screen, related to the size of the entire data buffer. The entire length of the slider represents the entire data buffer. The position and length of the light slider represents the data block displayed in the current drawing area relative to the entire data buffer. The length of the light slider can be used also to understand the ratio between the display data respect the total acquired data. The waveform thumbnail shows the waveform data displayed and the other waveform data in the buffer.

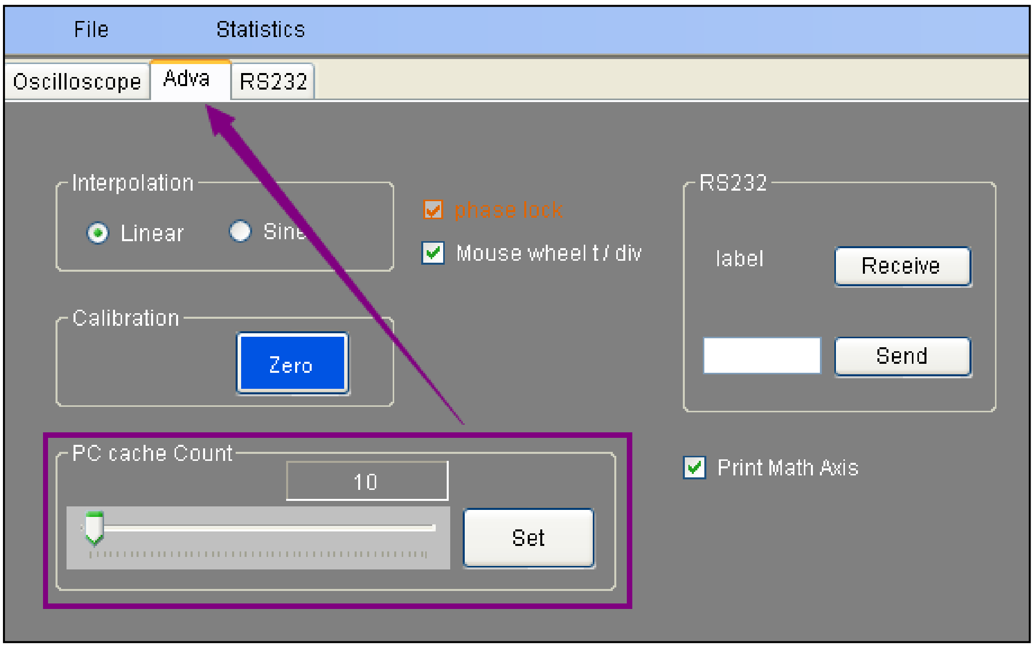
With the mouse you can drag the slider left and right in the entire data buffer, to show new areas of interest. Another way to move within the memory buffer is to use the left and right arrow keys of the keyboard. (Shortcut key (Ctrl + <-) / (Ctrl + ->) - Waveform move between left and right).



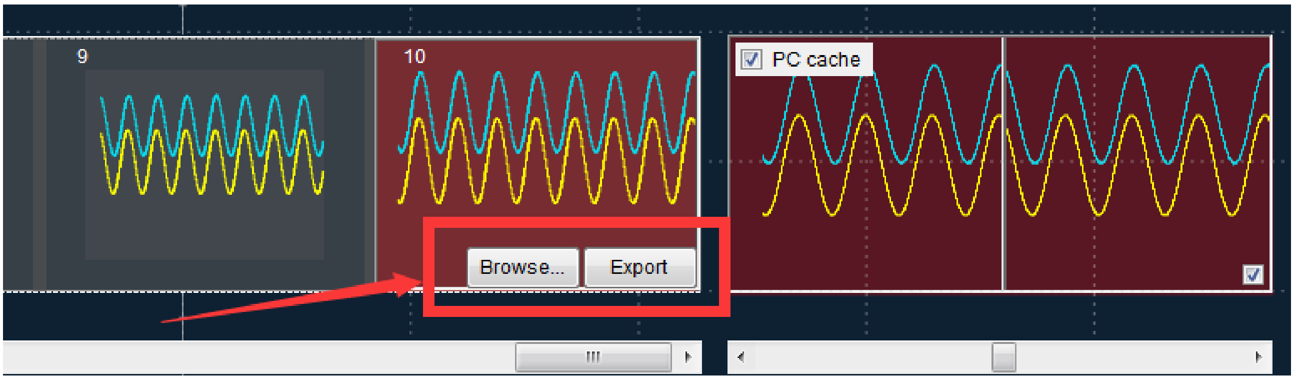
When the PC buffer function is checked, the software will queue up each frame of data collected by the oscilloscope on a first-in-first-out basis for frame buffer. When you find that there is a waveform of interest passing by on the screen, click the (Pause) button of the software, and you can choose to review a certain frame of waveform. The amount of one frame of data is the total data size of the time base gear buffer selected by the current user. Different time base gears have different buffer sizes, so how long a specific frame can be stored depends on the time base gear selected by the user.



The maximum number of PC buffered frames supports 500 frames, which can be set in the advanced page:



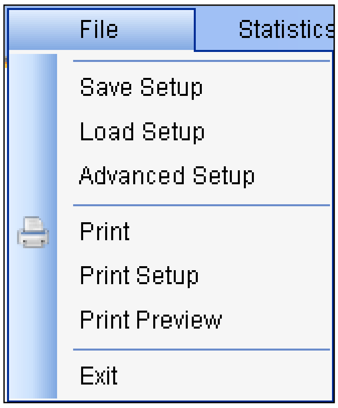
After the oscilloscope software is paused, the PC buffer function also supports: exporting buffered data frames to a file, and importing historical buffered frame data. In order to cache the frames of the historical PC of interest, perform local file persistent storage and review.



8.2.5.1 Video demonstration：

<LOTO oscilloscope software function demo "PC Cache Function">
https://www.youtube.com/watch?v=Q-DEDSef3Hw

8.2.6 Introduction to the Menu



8.2.6.1 File ⟶ Save Setup**:**

When saving you can set the destination path; the file suffix used for the saved file is **.set**. After saving, you can load a saved setup using the menu **File** → **Load Setup**.
Following are the settings saved with this operation:

No.	Setting
1	Time knob selection
2	Voltage knob selection (for all open channels)
3	Channel on or off status
4	AC/DC coupling status

8.2.6.2 File ⟶ Print **:**

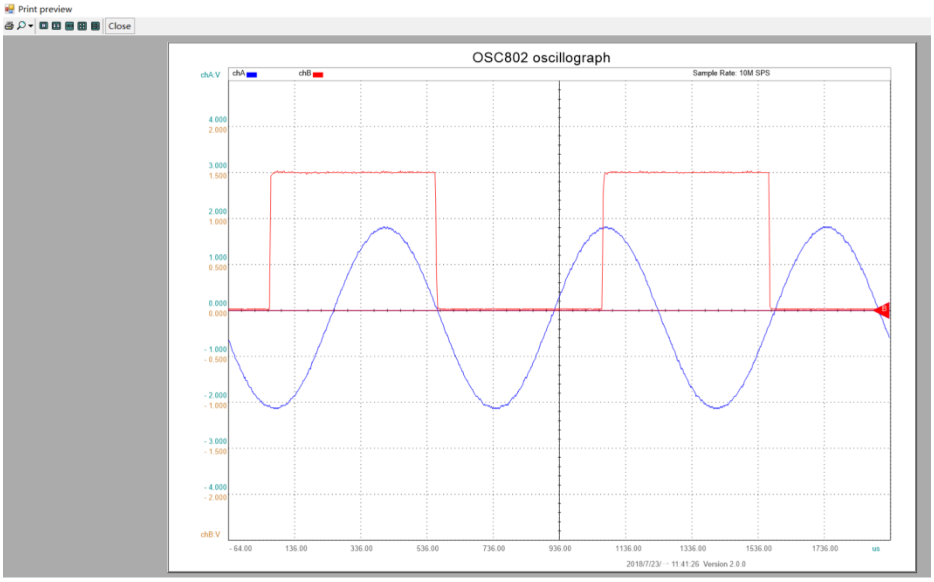
You can select a system printer and print the acquired signal data.

8.2.6.3 File ⟶ Print Setup **:**

You can choose the paper size and print format according your needs.

8.2.6.4 File ⟶ Print Preview **:**

Enter the print preview window and allows you to see what the waveform will look like on the screen before printing a hard copy.



8.2.6.5 File ⟶ Exit **:**

close the software.

8.2.6.7 “Advanced Setup”:

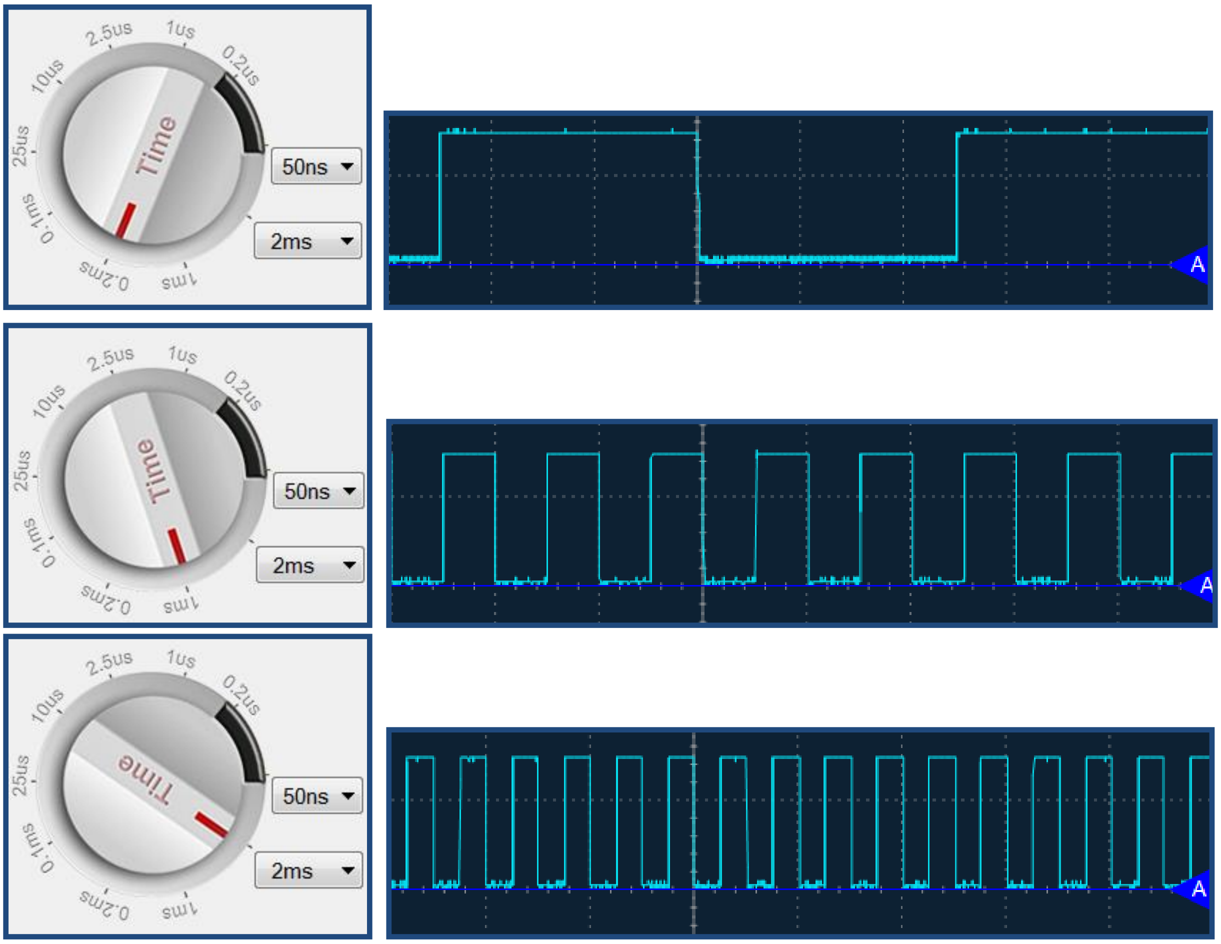
Save user interface custom settings, with memory capability. For example: FFT function is turned on by default when the software is turned on.

8.2.7 Introduction to the knobs

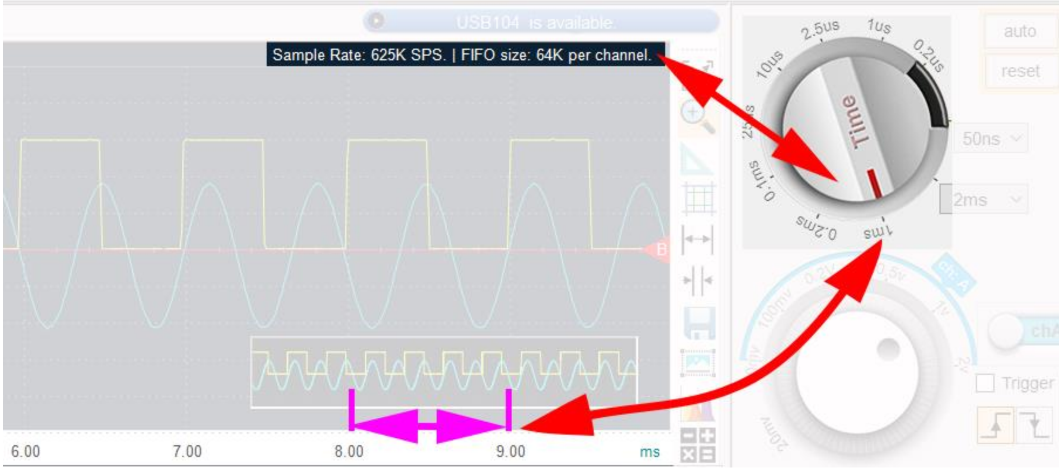
8.2.7.1 Time knob

All the channels of the virtual oscilloscope share the same time settings. The change of the time knob position affects each channel at the same time. The value selected by the time knob represents the time span represented by each horizontal division in the waveform drawing area. In the same way the time span of each horizontal division is the time value indicated by the position of the time knob. Therefore, the time axis coordinates changes according the different positions of the time knob.

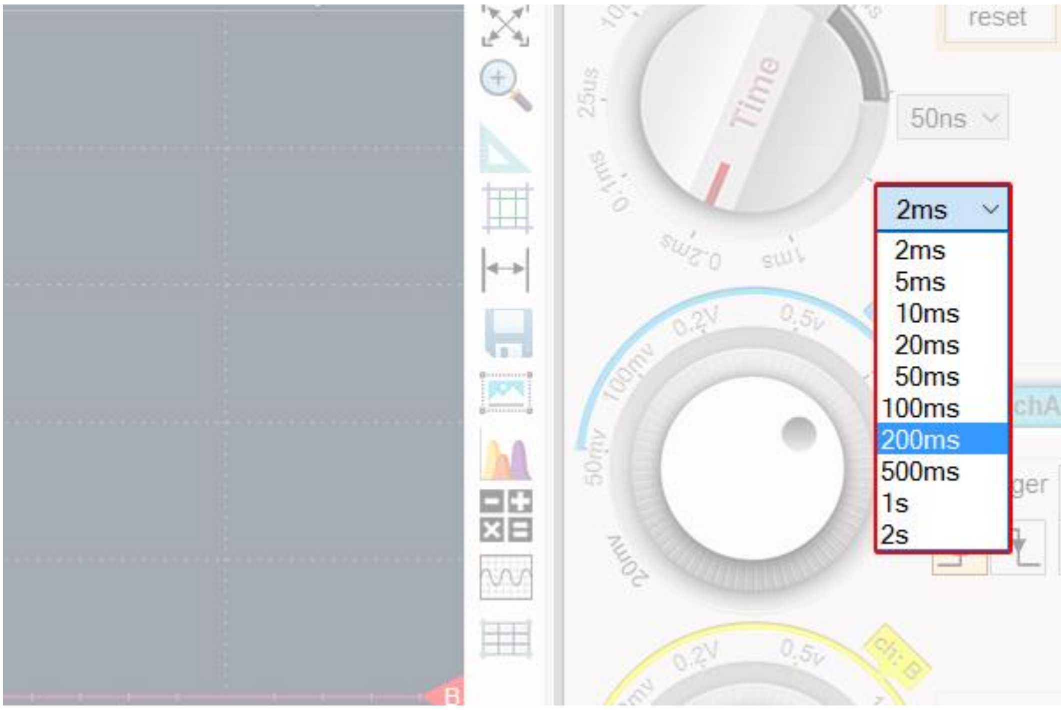
When the time axis is changed by the time knob, the device will select the most appropriate sampling rate, according the current time span and the memory depth on the device.



Sample of display of the same signal at different time scales



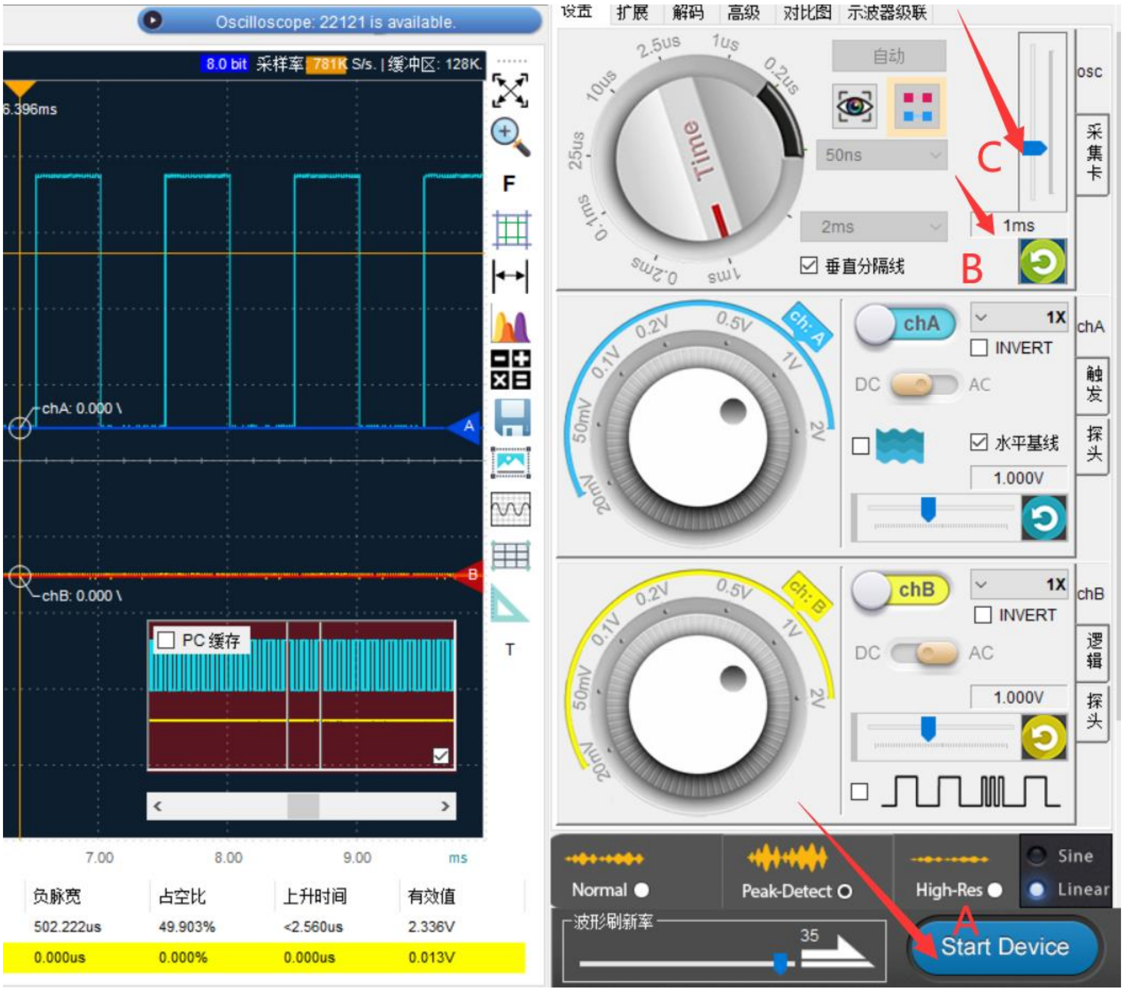
The time knob can be selected within a range from 5/50ns to 60s / division (depending on the model). Since there are 10 divisions on the horizontal axis, the time range of the waveform displayed on the screen vary from 50/500ns to 600s.



As shown in the figure above, our time wheel is set to the 1ms gear, as shown in the A position. Then the abscissa axis collected in our screen has a span of 1ms and is shown in position C. There are 10 horizontal grids in the waveform area, and the width of each grid is 1ms, such as position B. The entire screen width is 10ms.

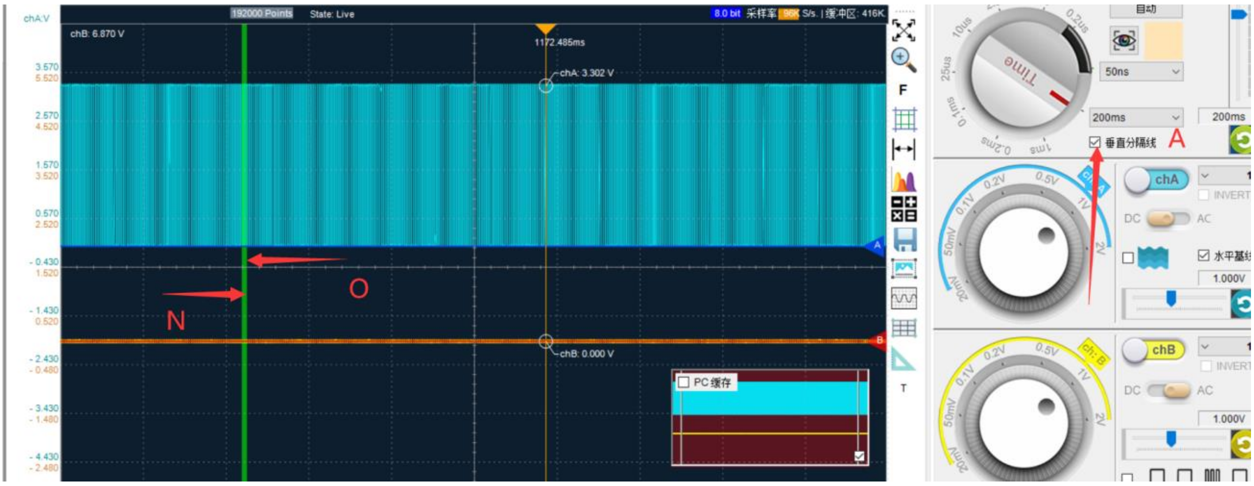
We set the time gear through the time wheel, and this function can only be used when the oscilloscope is in the acquisition state. If the oscilloscope is stopped, the time wheel is not allowed to operate, because the operation of the time wheel requires a real hardware setup action.

But when the oscilloscope is stopped, we can use the mouse or the slider to fine-tune the time slot. As shown in position A in the figure below, we stopped the oscilloscope device, and then we can pull the slider at position C to fine-tune the time gear. An equivalent and more convenient way of this operation is to scroll the mouse wheel. Both methods will make small changes to the current 1ms time slot, and the changed gear value will be displayed in position B. As shown below:



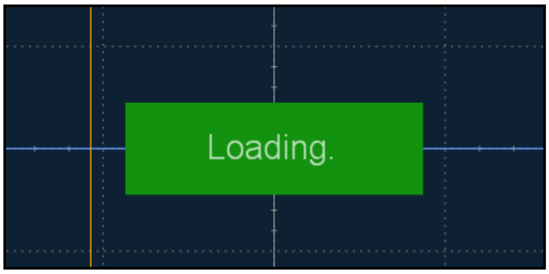
After fine-tuning it looks like this:

When the selected time base is large, such as greater than 200ms/per division, the drawing area will display the dividing line (green) for new and old data waveform refresh. If you do not need the new and old data dividing line, you can cancel the selection. dividing line), which is shown in the position A in the figure below.



In the figure above, the green vertical line is the old and new waveform data separation line. Its left side, as indicated by the N position, is the newly collected data waveform, and its right side, which is indicated by the O position, is the last time waveform. waveform data.

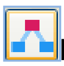
But when the time base gear is selected to switch between the large time gear and the small time gear, the oscilloscope needs time to complete the hardware switching, and Loading.. will appear in the software to remind the user to wait patiently. This prompt will disappear automatically after the switching is successful.

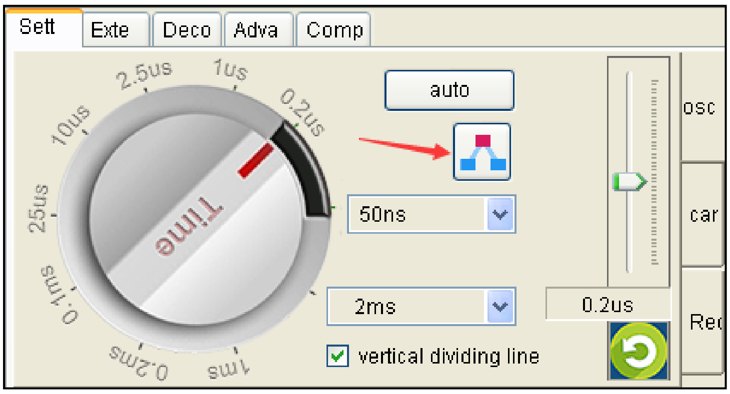


8.2.7.1.1 Combined acquisition / Sampling rate doubled

In LOTO's oscilloscope series products, some products support combined acquisition. The principle of combined acquisition is to merge the ADC resources of A and B channels into A channel, and the B channel will be disconnected and cannot be used. The original dual-channel ADC will use interleaved sampling technology to double the original sampling rate on chA. For example: OSCA02E series, OSC2002, OSCH02 series all support this function.

When operating the oscilloscope software, when the time base gear is adjusted to the us level, button  will appear on some products, indicating that the current time base gear supports the synthetic acquisition function.

Click the button  to turn on the synthetic collection function, and click the button again to release the synthetic collection function.

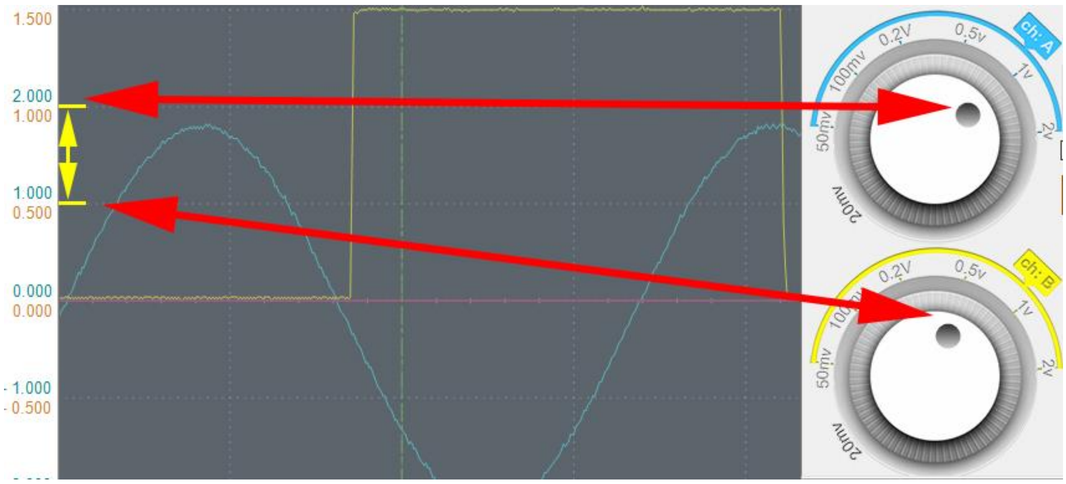


For some high-frequency signals, such as 50Mhz, 80MHz signals, etc., synthetic acquisition is not used. During normal acquisition, waveform signals will also be acquired due to the limitation of channel bandwidth. The actual situation is that the sampling rate is not enough to cause the acquisition to be aliased, and finally a complete waveform is presented, which is actually wrong.

After the synthetic acquisition function is enabled, the sampling rate and bandwidth of channel A are doubled. Channel B will lose the ability to collect.

8.2.7.2 Voltage Knob

Each channels of this virtual oscilloscope have its own voltage knobs. The value selected by the voltage knob of one channel determines the voltage span of each vertical division in the grid, referred to that channel waveform. The voltage grades differ depending on the model.



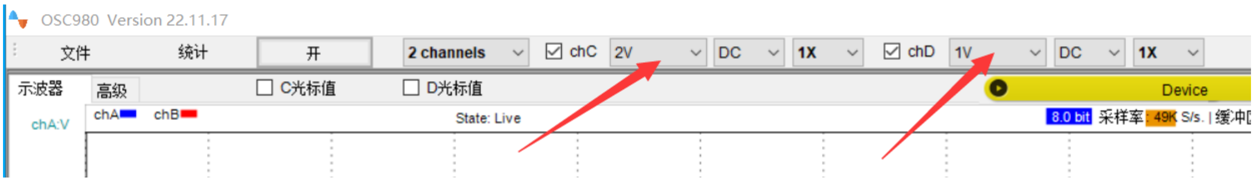
Changing the voltage scale with the voltage knob, the ordinate of the waveform in the display area will change accordingly.

If the amplitude of the signal has exceeded the upper or lower boundaries of the drawing area, it means that the selected voltage scale is too small, or the amplitude of the signal is greater than the maximum value supported from the device.

In this case, a red exclamation mark warning sign will appear on the voltage knob of the corresponding channel.

At this time, we need to increase the voltage gear selected by the voltage wheel. If the maximum voltage gear has been selected, there is still an over-range warning, indicating that the signal has exceeded the range, and the attenuation factor of the probe needs to be increased up.

For the four-channel OSC984 model, in the corresponding host computer software, in addition to the above-mentioned voltage gear knobs of the A and B channels, the voltage gears of the other two channels are set at the positions shown in the figure below:



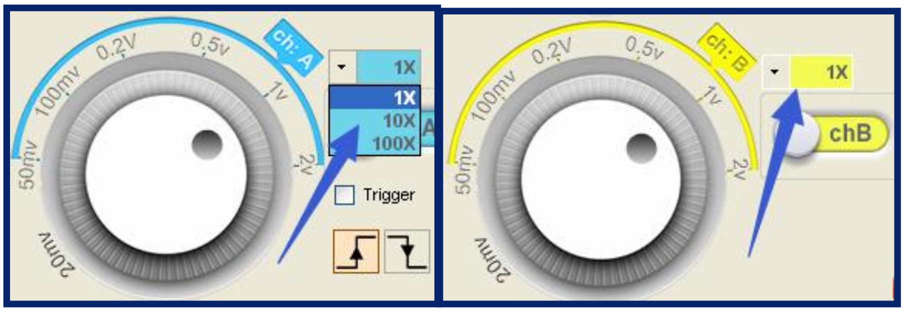
For a multi-channel oscilloscope composed of multiple standard two-channel oscilloscopes cascaded, except for the voltage gear knobs (wheels) of channels A and B, other channels are set on the cascading page, as shown in the figure below:

8.2.7.3 Probe selection

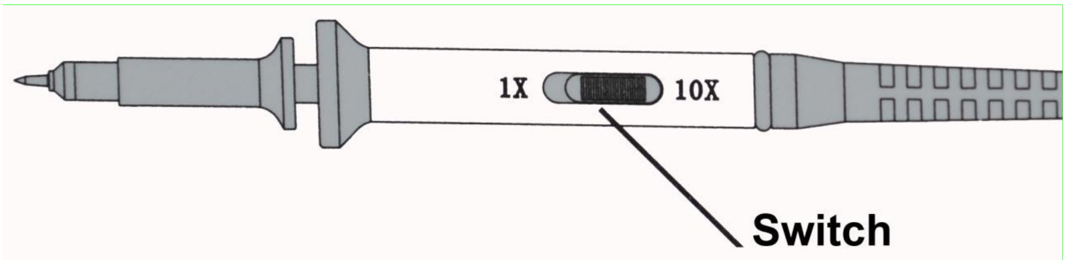
Using different probe divider affects the software's attenuation multiplier of the input signal, in particularly the Y-axis of the drawing area, as well as the measured value.

Probe Selection	Description	OSC482 Input Range	OSC802/A02/2002/ H02 Input Range
"1X"	Currently using a 1:1 probe	±5V	±8V
"10X"	Currently using a 10:1 probe	±50V	±80V
"100X"	Currently using a 100:1 probe	±500V	±800V

Probe selection setting area in the software:

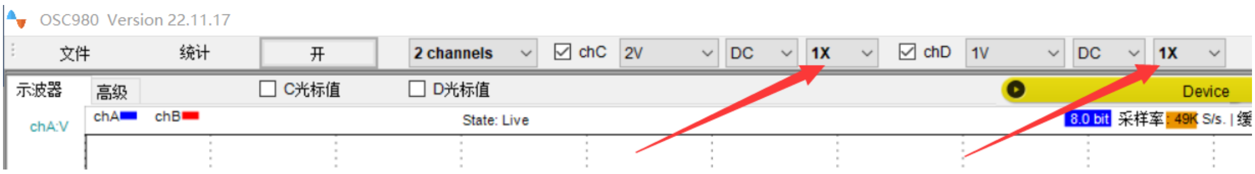


The corresponding selection switch of the probe:

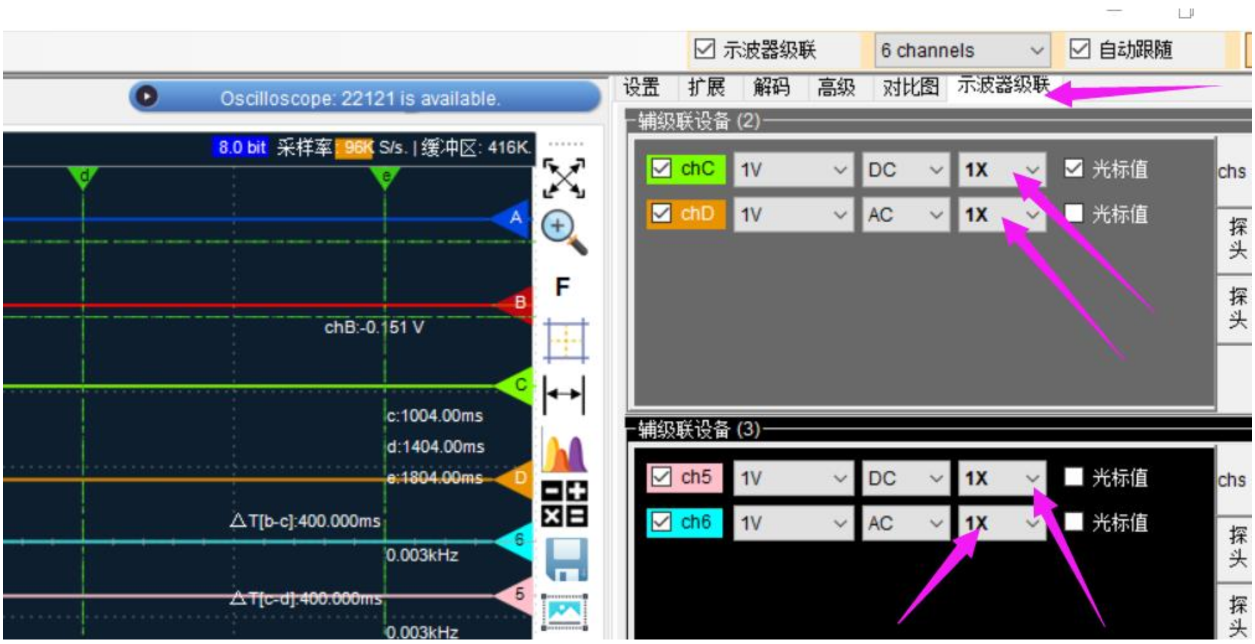


As shown in the figure, the probe multiple set by the software needs to be consistent with the attenuation multiple of the probe actually connected to the corresponding channel. If it is not the multiple relationship in the software setting list, we need to use the custom probe function of the LOTO oscilloscope, please refer to Chapter 13.

For the four-channel OSC984 model, in the corresponding host computer software, the attenuation multiple settings of the C-channel and D-channel probes are next to their corresponding gear settings. As shown below:



For a multi-channel oscilloscope composed of multiple standard two-channel oscilloscopes cascaded, except for the probe attenuation multiples of channels A and B, other channels can be set on the cascading page, as shown in the figure below:

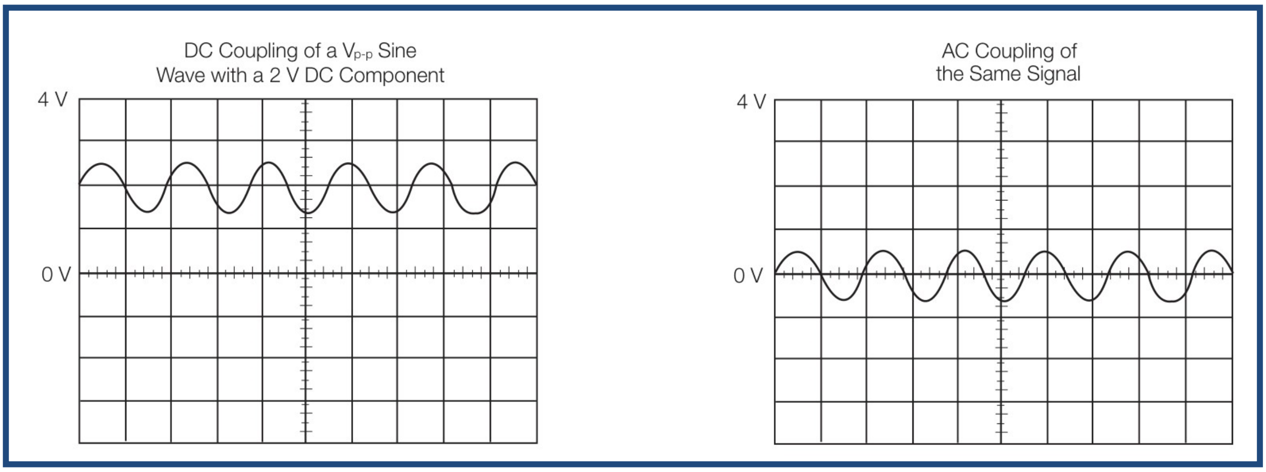


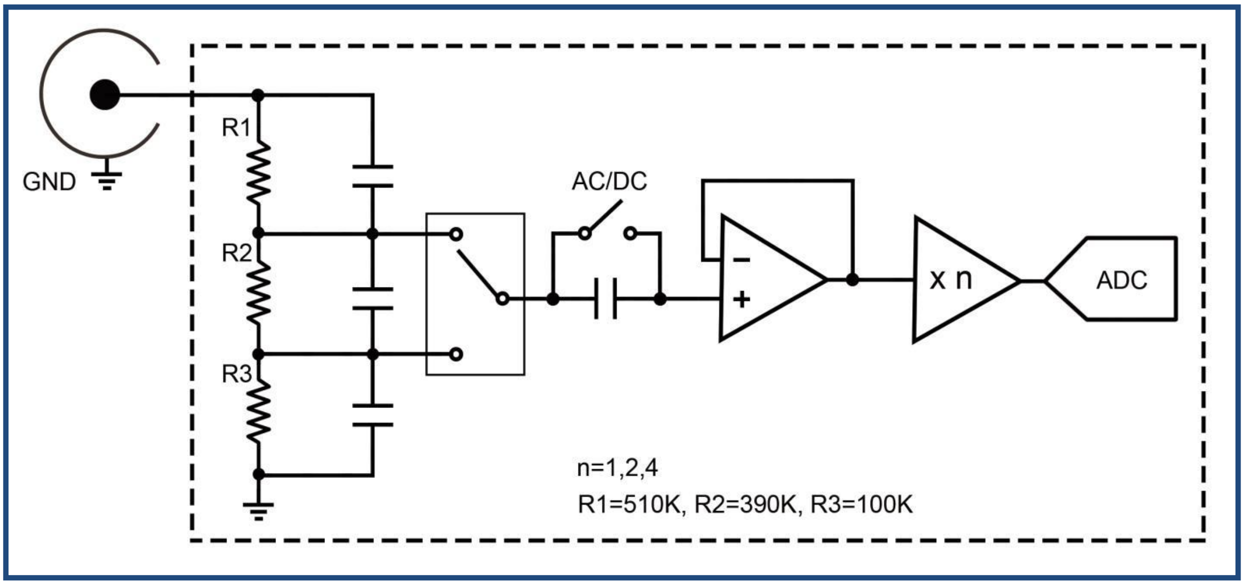
> **Note**
> The probe selection switch and the software probe selection are mechanical and human operations, and they should be kept on the same values. It is inevitable to forget to keep their settings consistent at some time. This may result in errors on waveforms or measured values.

8.2.8 Channel settings

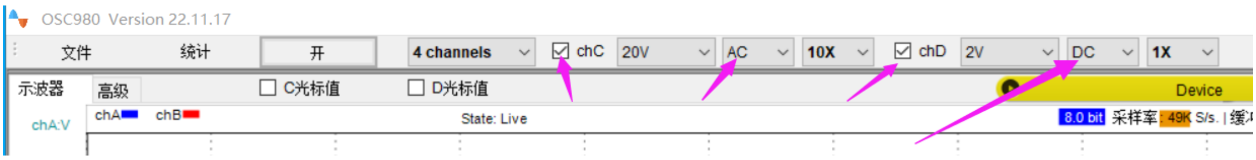
Channel A and channel B can be enabled and disabled with the following switch buttons. When the channel is enabled the button is respectively in cyan or yellow color, while when the channel is disabled the button become gray.

The signal coupling refers to the way the input signal is connected to the oscilloscope. When DC coupling (DC) is selected, the signal enters directly into the oscilloscope. When AC coupling (AC) is selected, the oscilloscope internally isolates the DC component of the measured signal by connecting a 0.1u capacitor in series to the input so that only the AC component of the signal is acquired by the oscilloscope. You can select the Input Coupling (DC or AC) for each channel. The default coupling mode is DC coupling.

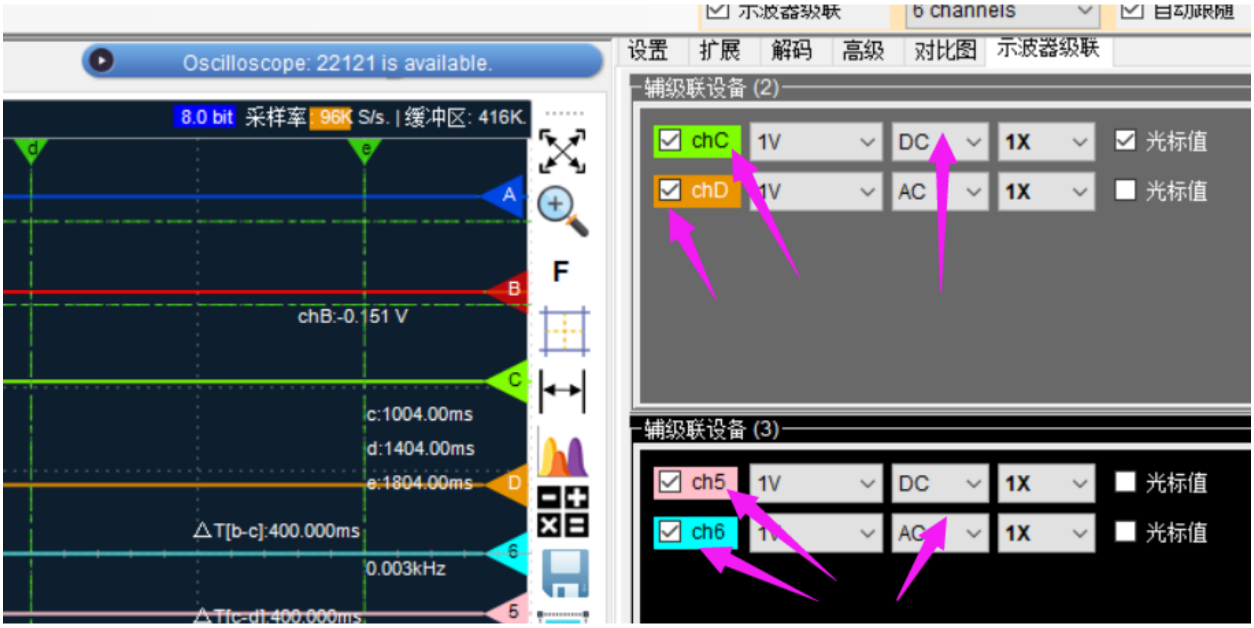




For the four-channel OSC984 model, in the corresponding host computer software, these channel settings are next to their corresponding gear settings. As shown below:

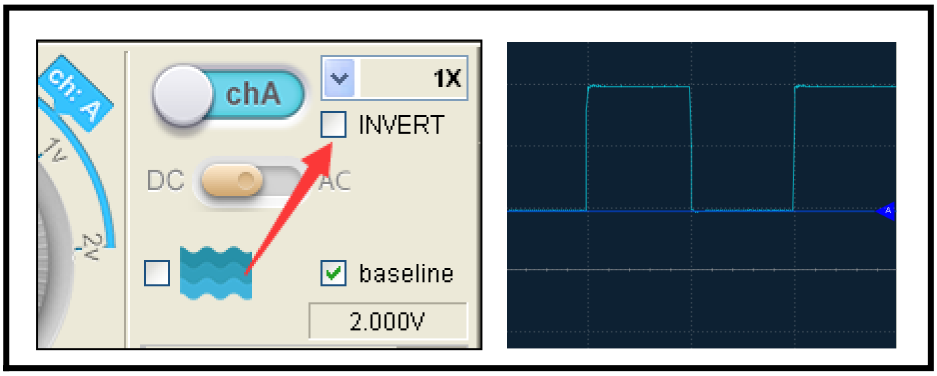


For a multi-channel oscilloscope composed of multiple standard two-channel oscilloscopes cascaded, except for the settings of A and B channels, other channels can be set on the cascading page, as shown in the figure below:

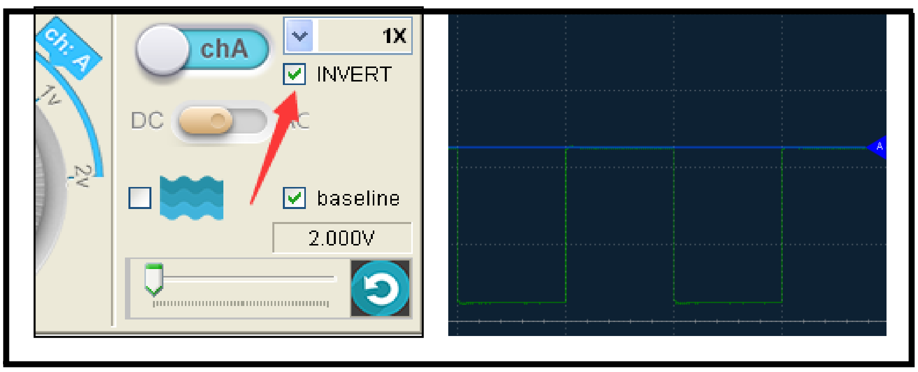


8.2.8.1 Waveform invert

The INVERT switch, the waveform of the corresponding channel is symmetrical about the X axis (upside down), as shown in the figure, the same output signal, before and after turning on the invert switch: Function disabled: The waveform is on the positive semi-axis above the baseline.



After the function is turned on: The waveform is flipped over the negative half axis below the baseline.



8.2.9 Waveform recording and playback

This function is deleted in the latest version of the software, and it can be perfectly replaced by using the PC cache function. For the PC cache function, please refer to Section 8.2.5.

8.2.10 Automatic measurements

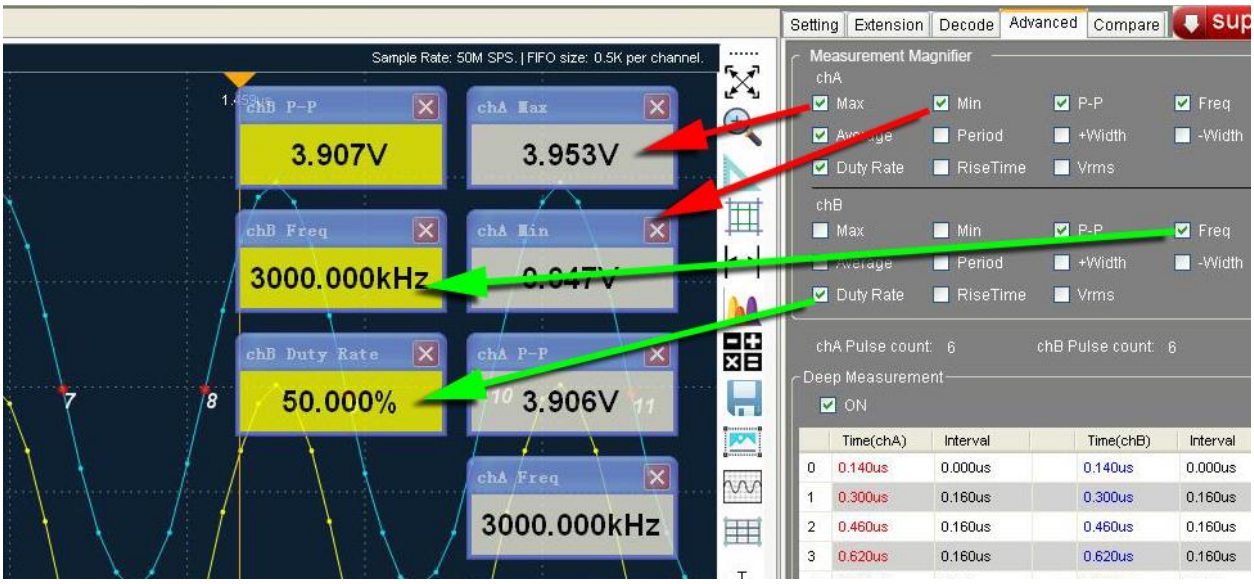
During the acquisition the signal statistics are computed automatically in real-time and showed in the measurement area below the waveform display area. These values are displayed only for the active channels.

Measurement	Description
Max	The highest voltage value of the current channel waveform on the screen, in Volts
Min	The lowest voltage value of the current channel waveform on the screen, in Volts
P-P	Peak to Peak Voltage (Max – Min), in Volts
Frequency	The average frequency of the current channel waveform on the screen, in Kilohertz (KHz).
Average	The average voltage of the current channel waveform on the screen, in Volts.
Period	The signal period, in microseconds (us).
+/-Width	Respectively the Width of a positive pulse at 50% crossings and the Width of the negative pulse at 50% crossings, in microseconds (us).
Duty cycle	Positive pulse width as percentage of period.
Rise Time	The time it takes for the waveform to rise from the 5% position of the peak value to the 95%, in microseconds (us).
Vrms	The root mean square value of the current channel waveform, in Volts.

Note
If the waveform of the current channel is displayed on the screen for less than one period or for more than 50 periods, the Period measurement will be displayed as 0

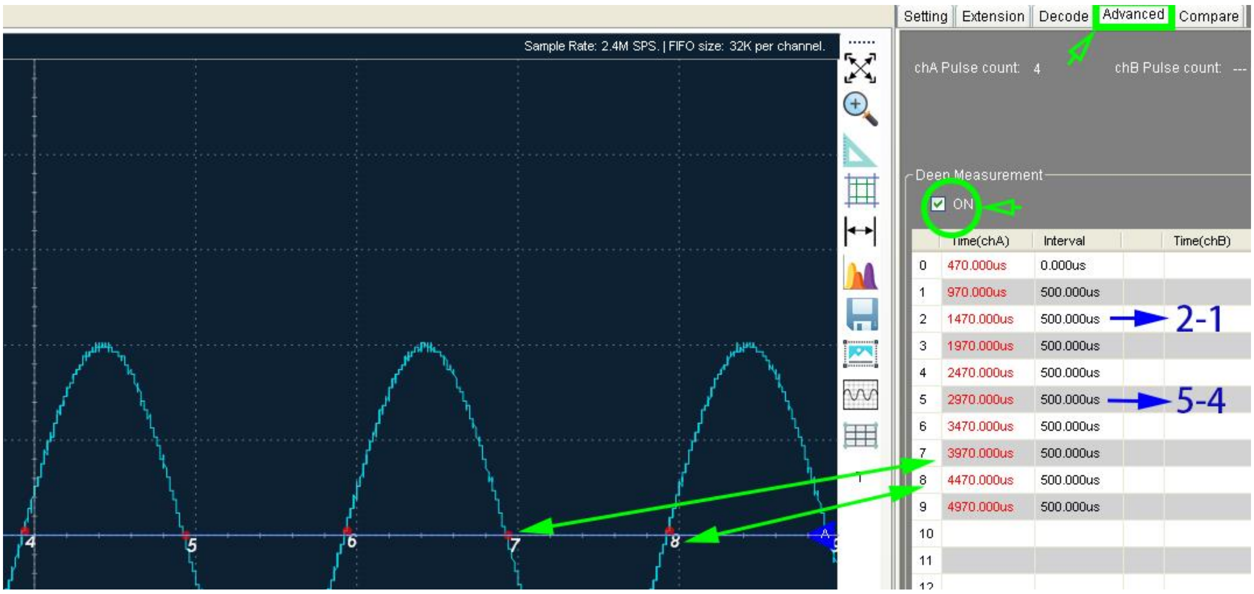
because the measure cannot be performed, but this don't mean that the signal period is actually 0.

8.2.11 Measurement Magnifier



The automatic measurements described in the previous chapter will be shown below the software, and sometimes we need to focus on a few measurements, and hope to be more intuitive and convenient to observe them at any time, we can use this magnifier function. In the Advanced tab on the right, we can choose which measurement to magnify. This function does not conflict with the automatic measurement described in the previous chapter, they can coexist.

8.2.12 Deep measurement



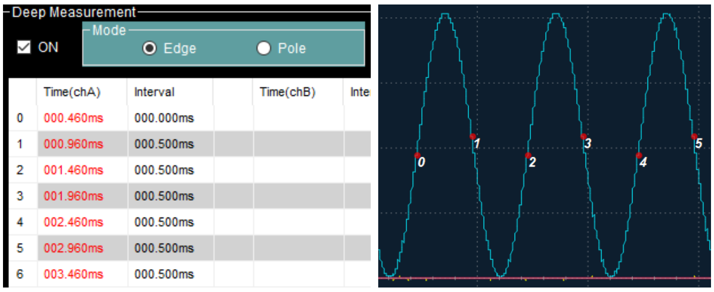
In the Advanced tab on the right side, we can choose to start the deep measurement function, as shown by the green arrows in the figure. When the deep measurement function is turned on, the data waveform displayed will automatically add numbered marker points near the zero-crossing point.

The number of marker points depends on the waveform displayed on the current screen. The numbered marker points are arranged in the table on the right .The time value of the marker points displayed in red as a column, as shown by the long green arrow in the figure.

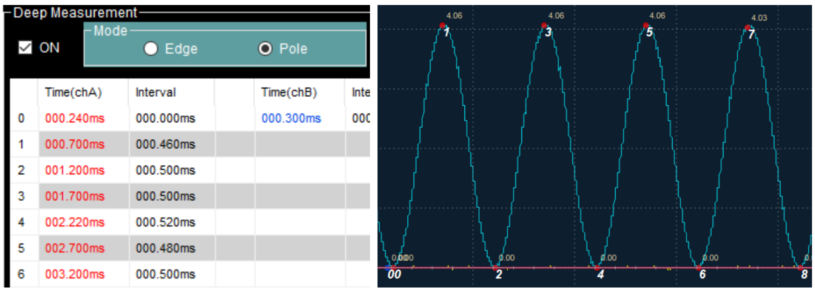
The next column displays the time interval between two adjacent numbered marker points, as shown by the blue arrow. The time interval of each row is obtained by subtracting the time of the current marker points from the previous one.

Currently, there are 2 types of multi-point measurement algorithms supported:

Edge labeling algorithm:

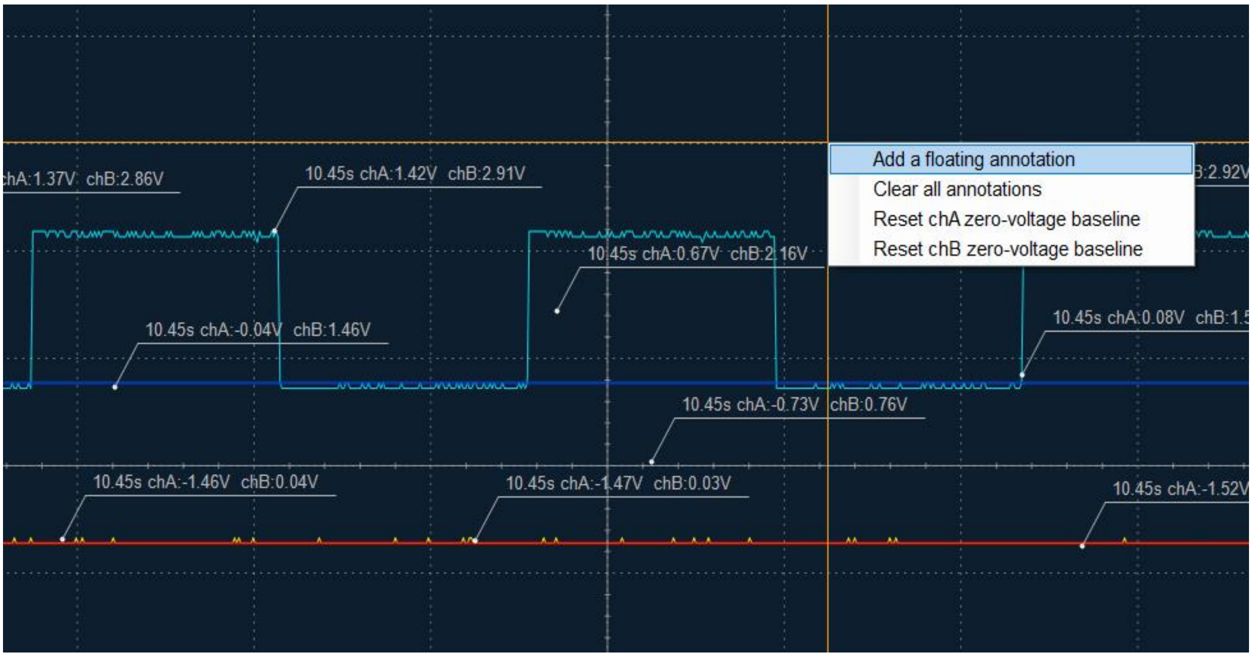


Extreme point labeling algorithm:

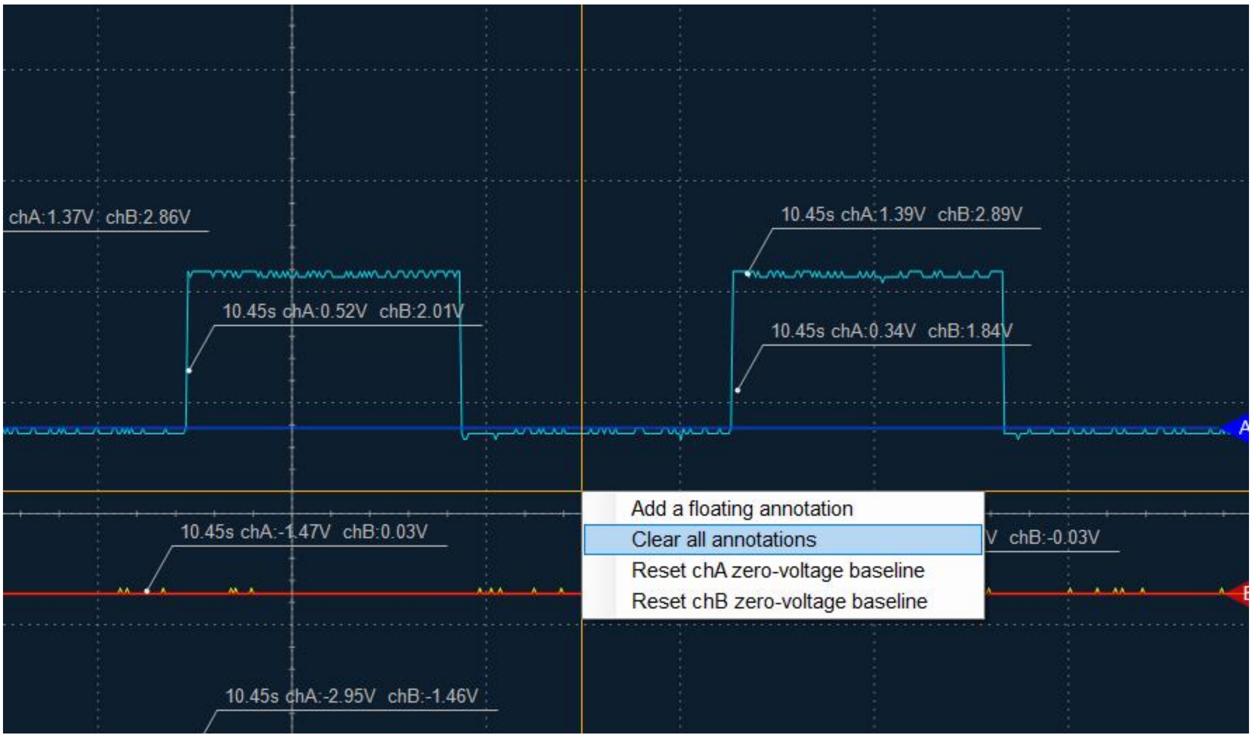


8.2.13 Annotations

The first item can be found in the right-click menu of waveform display area, and floating data annotation can be added. Data annotation can be dragged to any position in the waveform display area with the mouse, which will display the channel voltage value and the time value of the annotation point. Data annotations can be added and dragged continuously to places of interest, adding up to 10.



In the second row of the right-click menu, there are options to clear all data annotations, as shown in the following figure:



8.2.14 Trigger

LOTO oscilloscopes are divided into two trigger types, software trigger and hardware trigger. The OSC482 series is an entry-level model. For cost considerations, this system uses a software trigger mechanism. Other high-end models are hardware trigger mechanisms, such as (OSC802, OSCA02, OSC2002, OSCH02, OSC980). The principle of software triggering is to collect a piece of data, and use software methods to find the position that meets the trigger condition in the data, which requires CPU computing time, so it is not 100% guaranteed to find the trigger position, because the trigger signal may not be in the current section data. The principle of hardware triggering is that after the trigger conditions are set, these settings are monitored by hardware logic, and the hardware logic circuit searches for the position in the signal that meets the conditions, and then collects data after finding it, so for a single burst signal, the hardware trigger can guarantee 100% triggering. But for signals that appear repeatedly, the difference between software trigger and hardware trigger is not very different. You can set the trigger in the trigger setting window and enable the trigger by selecting the checkbox.

In addition, hardware triggers can cover more time slots than software triggers. The software trigger is triggered in the middle of the buffer by default, and the hardware trigger mechanism can choose where to trigger in the buffer.

8.2.14.1 Video demonstration：

<LOTO Oscilloscope || Trigger operations>
https://www.youtube.com/watch?v=jfDzUFgf-V4&t=5s

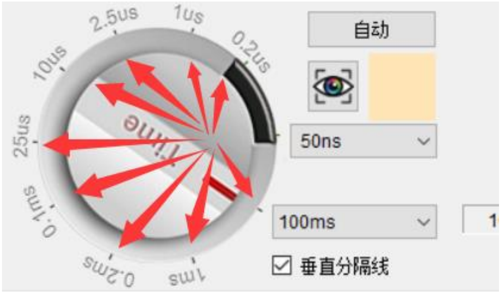
8.2.14.2 Trigger channel

In general, LOTO oscilloscope can be triggered only on channel A, so the signal to be triggered shall be connected to chA. If you purchase an external trigger module, you can use an external trigger channel independent of channel A and channel B. That is to say, if the external trigger function is used, the waveform of two channels AB can be captured and detected at the trigger time. This is very suitable for situations where trigger signals do not need to be displayed, and one channel A can be saved as a common waveform channel. For external triggering, please refer to the detailed introduction in Section 8.2.14.4.

8.2.14.3 Trigger time gear

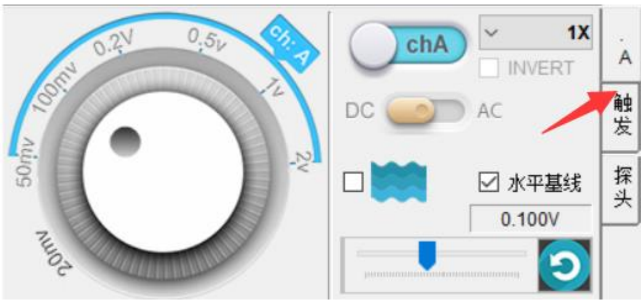
If you purchase the OSC482 series products, as shown in the figure below, the time gear marked with a black dot on the time gear wheel can be triggered, while other gears cannot be triggered:

If you purchase other series of products, such as OSC802, OSCA02, OSC2002, OSCH02, OSC980, almost all time slots smaller than 100MS can be triggered by hardware, as shown in the following figure:

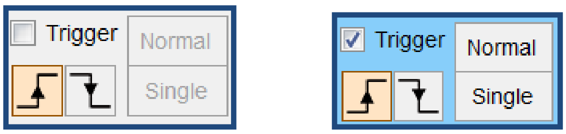


8.2.14.4 Trigger settings

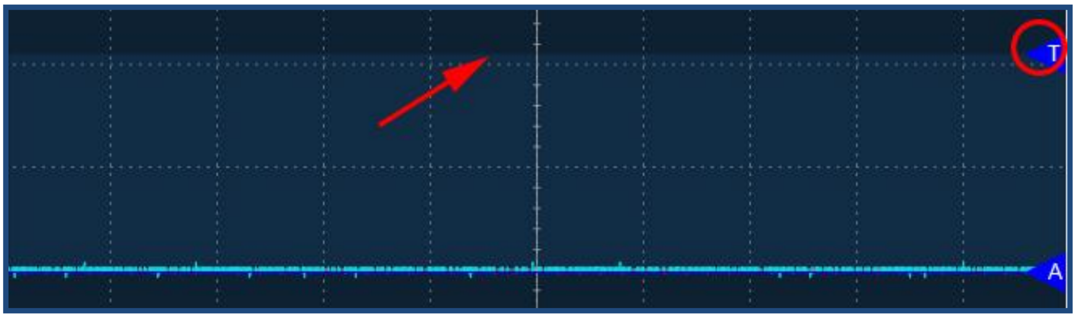
The triggering function is enabled only when the triggering module is checked, and terminated only when the triggering module is checked.



Some large time gear trigger functions are disabled, and the check box is gray and unavailable. The display time of a screen waveform in a large time gear is much longer than the observation speed. For example, it takes 30 seconds to display a screen waveform. Therefore, it is unnecessary to use the trigger function. You can completely pause the oscilloscope to collect and observe and analyze the waveform when necessary.

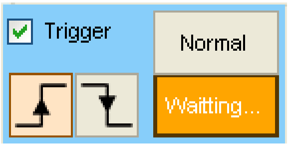


The trigger function can be used only on channel A, so when you need to apply the trigger to a signal, the signal should be connected to channel A. When the trigger is enabled, a triangle with 'T' inside will appear on the right of the waveform display area to allow the trigger level setting by dragging this blue triangle up and down with the mouse.

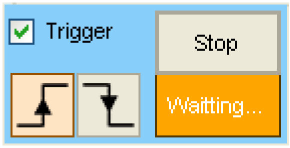


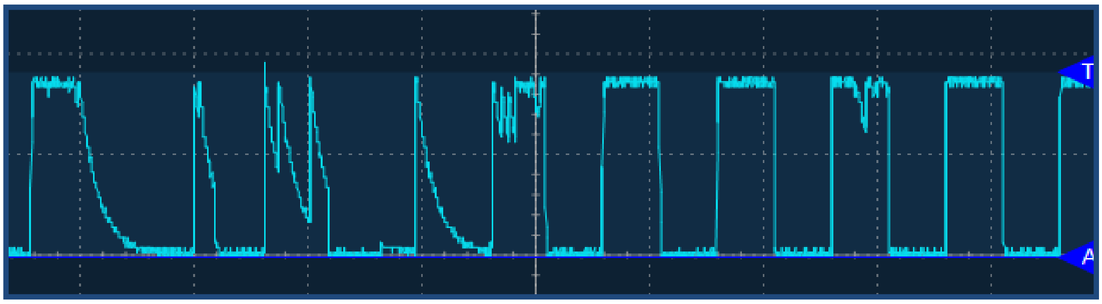
The trigger level is combined with the selected rising or falling edge to define the trigger condition. Considering as example the Normal Trigger with rising edge, the trigger condition is met when the input voltage of channel A goes from low to high respect to the voltage value set by the trigger level. When the trigger condition is met, the screen will display the entire waveform before and after the trigger condition, keeping it until the trigger function is checked or the next trigger occurs.

The Single trigger can easily capture accidental events, such as glitches with sudden changes in waveform amplitude. You can set the trigger level just above the maximum value of the normal signal, then click the **Single** button and wait for the trigger event to occur. When the signal fluctuates beyond the trigger level, the oscilloscope automatically records the waveform for a period before and after the trigger, and show on the software display area as shown in the following figure:

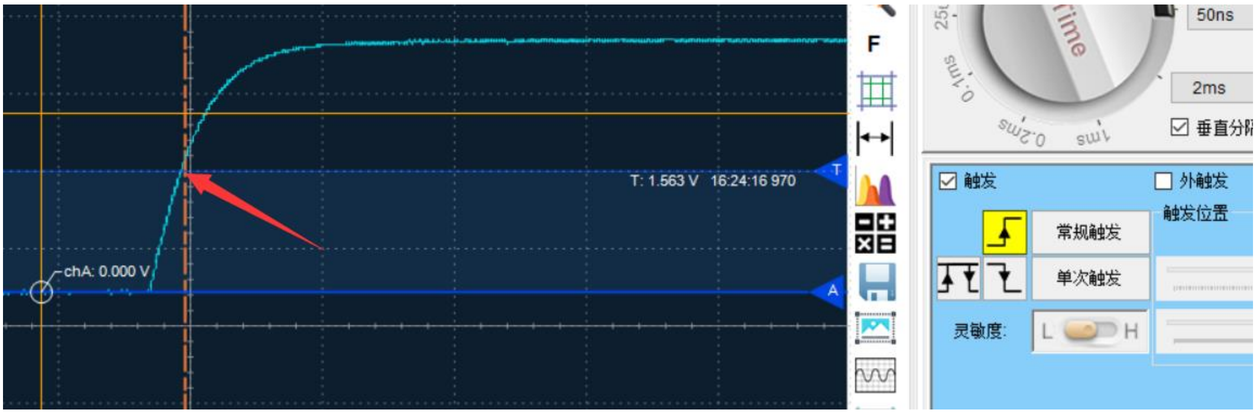


When the signal fluctuation is frequent, if you want to continuously capture and show the signal when matching the trigger criteria, then you need to use the Normal trigger with the **Normal** button. During trigger scanning, the **Stop** button with appear for stopping the operation. Some model device may not have this normal trigger feature.





Take the rising edge trigger as an example, when the input signal voltage of channel A goes from low to high and exceeds the voltage value set by the trigger level, the trigger condition is met. At this time, the screen will display the entire waveform state before and after the trigger condition occurs, and stay in this state without being updated until the trigger function is checked, or the next allowed trigger occurs.



The trigger function of LOTO oscilloscope is divided into two ways: regular trigger and single trigger.

The mechanism of the conventional trigger mode is that each time the waveform that meets the trigger condition is displayed on the screen, the next time a new waveform that meets the trigger condition arrives, the new waveform will be displayed, and the next tie the trigger condition is met, the latest waveform will be displayed. Always replace the last triggered waveform display with the latest triggered waveform display.

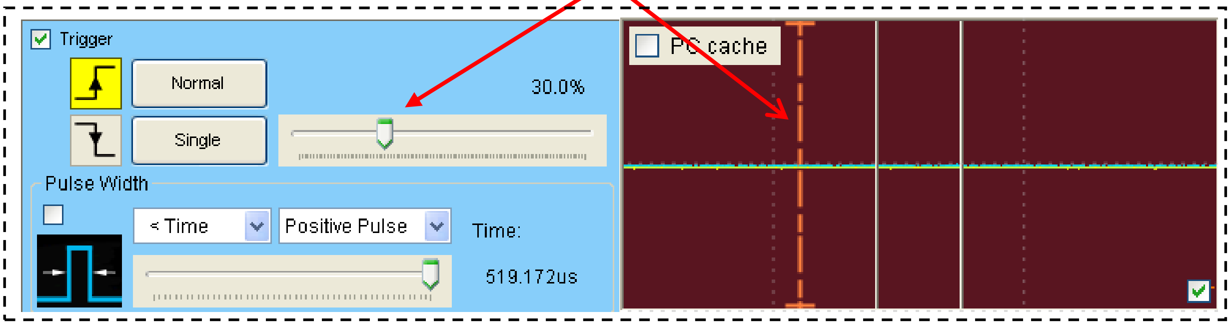
The mechanism of single trigger is that after clicking the "single trigger" button, only the waveform that meets the trigger condition for the first time will be displayed on the screen, and then the screen will not be updated until you click "single trigger" again to capture for the next signal, or you click the "General Trigger" button to switch to a new trigger mechanism, or cancel the trigger function, new waveforms will be collected and updated on the screen in real time.

8.2.14.5 Advanced trigger

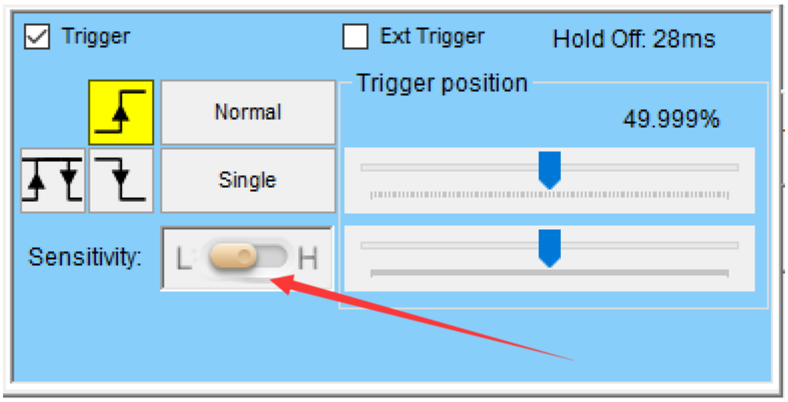
Some products support buffer trigger position selection and pulse width trigger. Later, in order to add other resources, we removed the pulse width trigger function from the standard products. Customers who need this function need to customize.

The OSC482 series does not support the function of setting the trigger position of the buffer. Other hardware departure models support this function. As shown in the figure below, in the software of models that support trigger position setting, you can drag the slider in the trigger setting area to set the trigger horizontal position of the waveform in the entire buffer. By default, we are at the middle 50% of the buffer. After triggering, we can see the waveform data of the front and back half of the buffer. If you want to see more waveforms after triggering, you can set the trigger position to a horizontal position near the front of the buffer.

Trigger Position



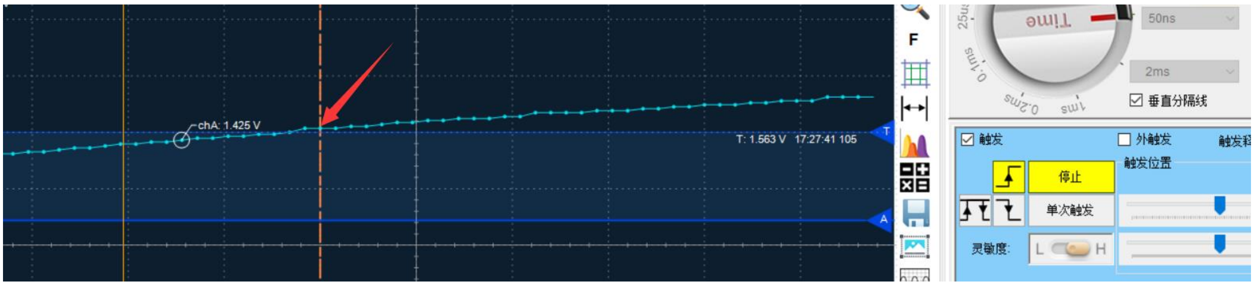
In the new version of hardware, the trigger sensitivity function is added to the hardware trigger mechanism. Some hardware versions do not support this function:



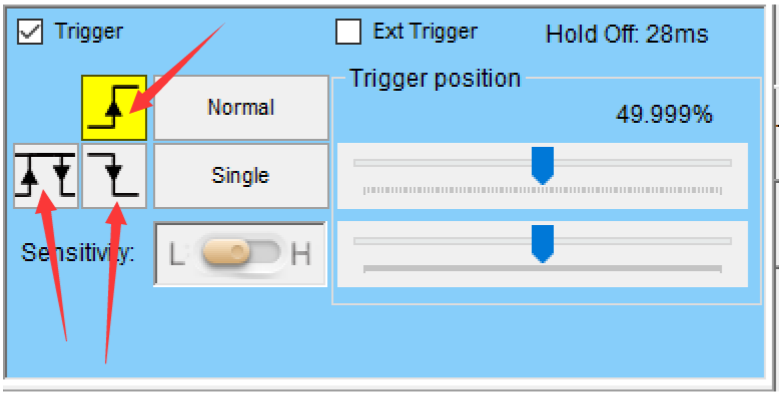
Select L (low sensitivity) in low gear and H (high sensitivity) in high gear The software defaults to low sensitivity trigger. This function is for the convenience of balancing the precise trigger position and anti noise interference.

If the sensitivity is too high, for some signals with dense small burrs, the up and down jitter of the burrs at the trigger level will cause false triggering. I choose the rising edge trigger. Maybe when the falling signal reaches the trigger level, a small burr just rises. This is because the trigger is too sensitive, and the falling edge signal may be captured as the rising edge. Therefore, when we observe that the edge trigger will often capture the trigger waveform of the reverse edge, it should be because the sensitivity is too high. At this time, we should set the sensitivity to low L.

On the other hand, when the sampling rate is very high to collect high-speed signals, the sampling points generally appear sparse. At this time, if the trigger sensitivity is not high, the trigger may shift several sampling points, or even dozens of sampling points. Then the waveform we see is not at the set trigger position, or even far away from the feeling. At this time, we should choose high sensitivity, as shown in the following figure:



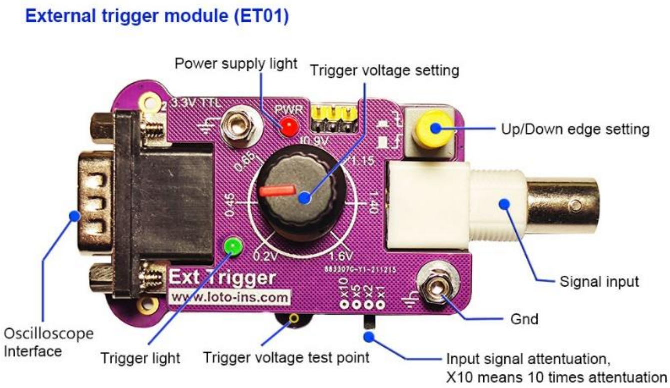
Triggered variable edge selection, generally there are two kinds of rising edge and falling edge. The version that supports hardware triggering adds double edge triggering, that is, as long as the jump edge crosses the trigger level, both the rising edge and the falling edge are considered to meet the trigger conditions.



For a more detailed description of the relevant trigger sensitivity, please refer to Appendix V: LOTO Oscilloscope Trigger Sensitivity.

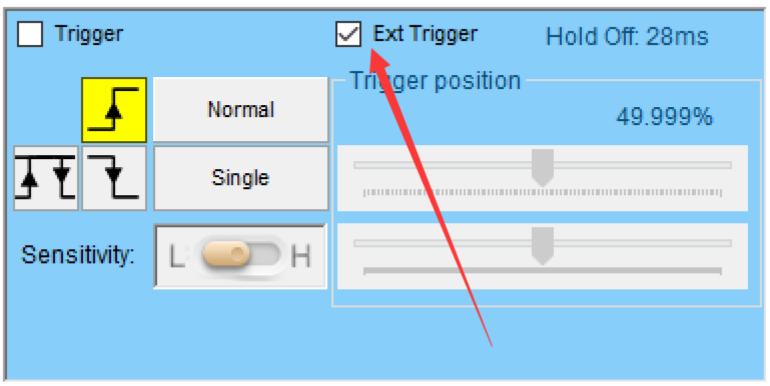
8.2.14.6 External Trigger

When the OSCA02, OSC2002 and OSCH02 series need to trigger the third signal and monitor the waveform of the other two analog signals at the trigger time, the external trigger module E01 can be purchased. The external trigger module needs to be plugged into the side expansion interface DE2 of the LOTO oscilloscope.



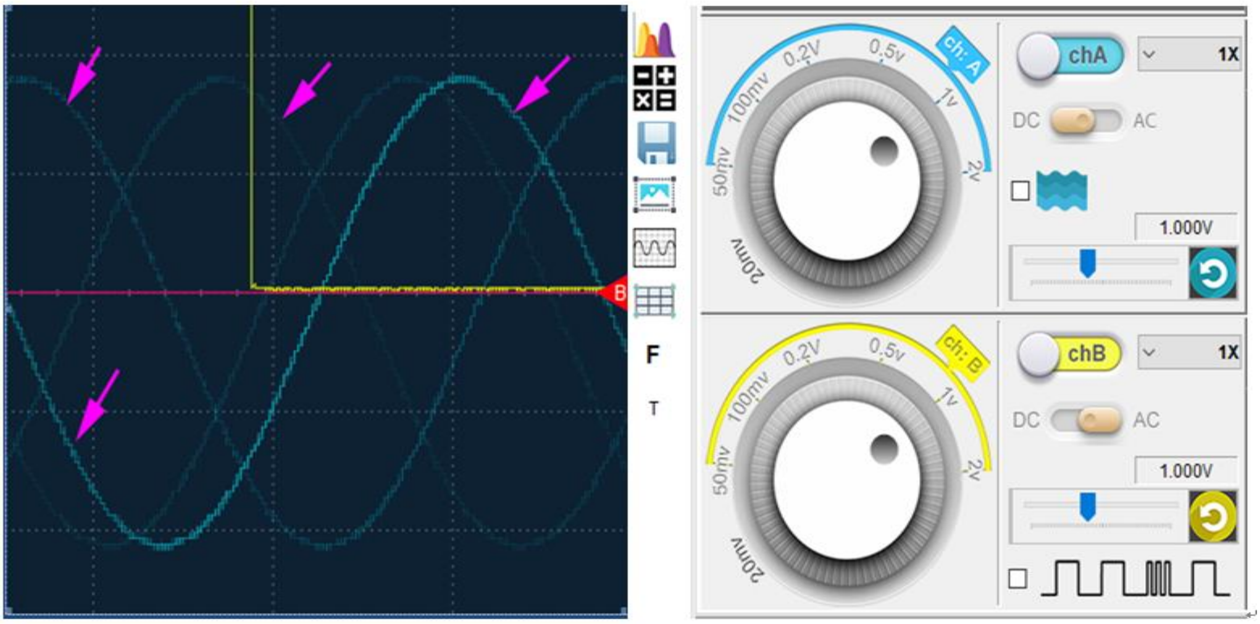
Parameter	ET01		
Channel Number	1		
Input Resistance	1MΩ		
Output interface	DE-15		
Output signal	3.3V ttl		
Trigger level	0.2V~1.7V Continuously adjustable (0V~5V range, other ranges are expanded by multiples)		
Trigger edge	Rising edge/falling edge selectable		
bandwidth	10K Hz		
Input range(4 gears)	X1	No attenuation	Input range: 0V~5V
	X2	2 times attenuation	Input range: 0V~10V
	X5	5 times attenuation	Input range: 0V~25V
	X10	10 times attenuation	Input range: 0V~50V

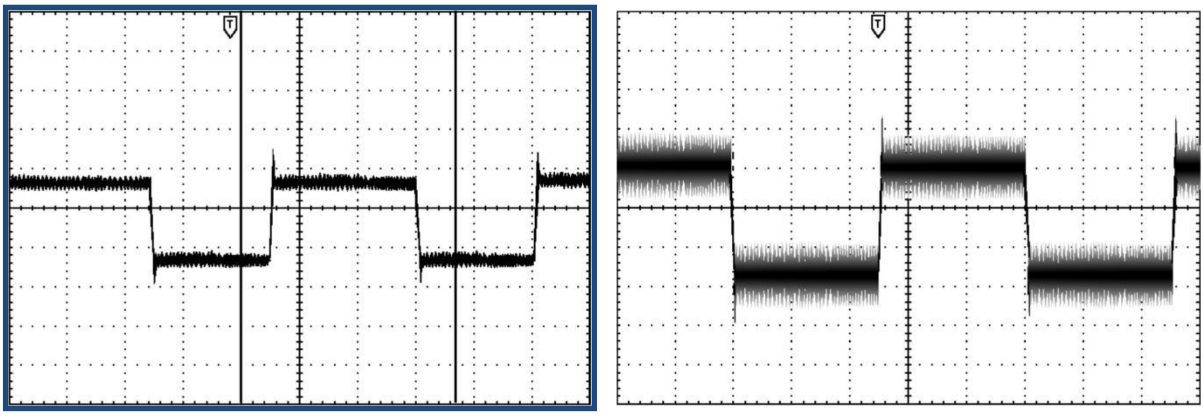
The external trigger only provides a trigger source different from chA in hardware, and the use method is the same in software. We only need to turn on the external trigger function in the following positions. As shown in the figure below:



8.2.15 Waveforms persistence

The waveforms persistence effect superimposes on the screen the data acquired in successive times and allow the observation of the spatial concentration of the waveform energy. When the Glow checkbox is set, the oscilloscope continues to acquire and display new data, but do not erase the previous collected data. The waveform parts with higher occurrence will have higher brightness; parts with fewer occurrences will appear less bright. The waveforms persistence superimposition has a significance as a statistical measurement, since it can show in intuitive way the distribution of noise on the time and space. This makes it suitable for the preliminary analysis of random noise.

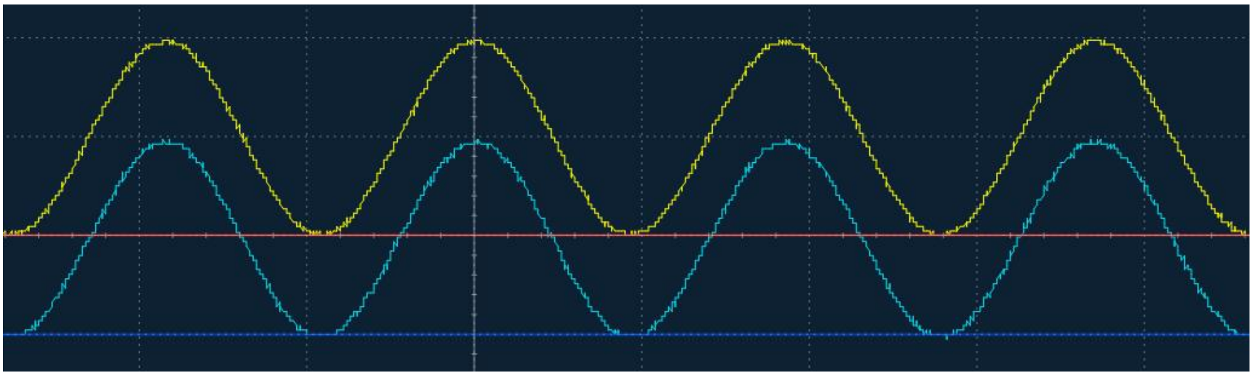




8.2.16 Acquisition Modes

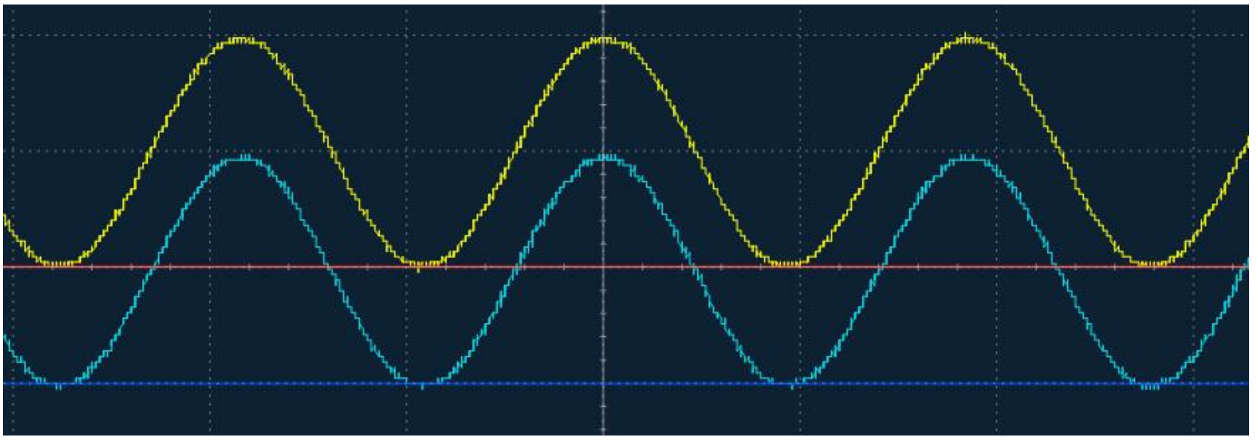
Normal mode:

This is the most common sampling mode. The oscilloscope store and display the samples according to the sampling rate.



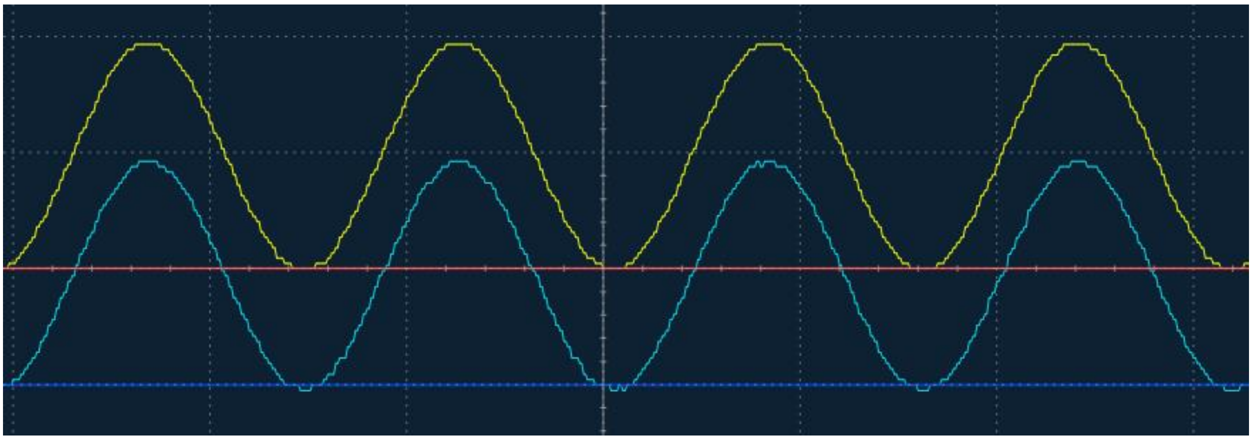
Peak Detect mode:

The oscilloscope samples always at the maximum sampling rate to find the maximum and minimum values within the time interval for each sample, and then it displays the maximum and minimum values on the waveform curve for each sample position.

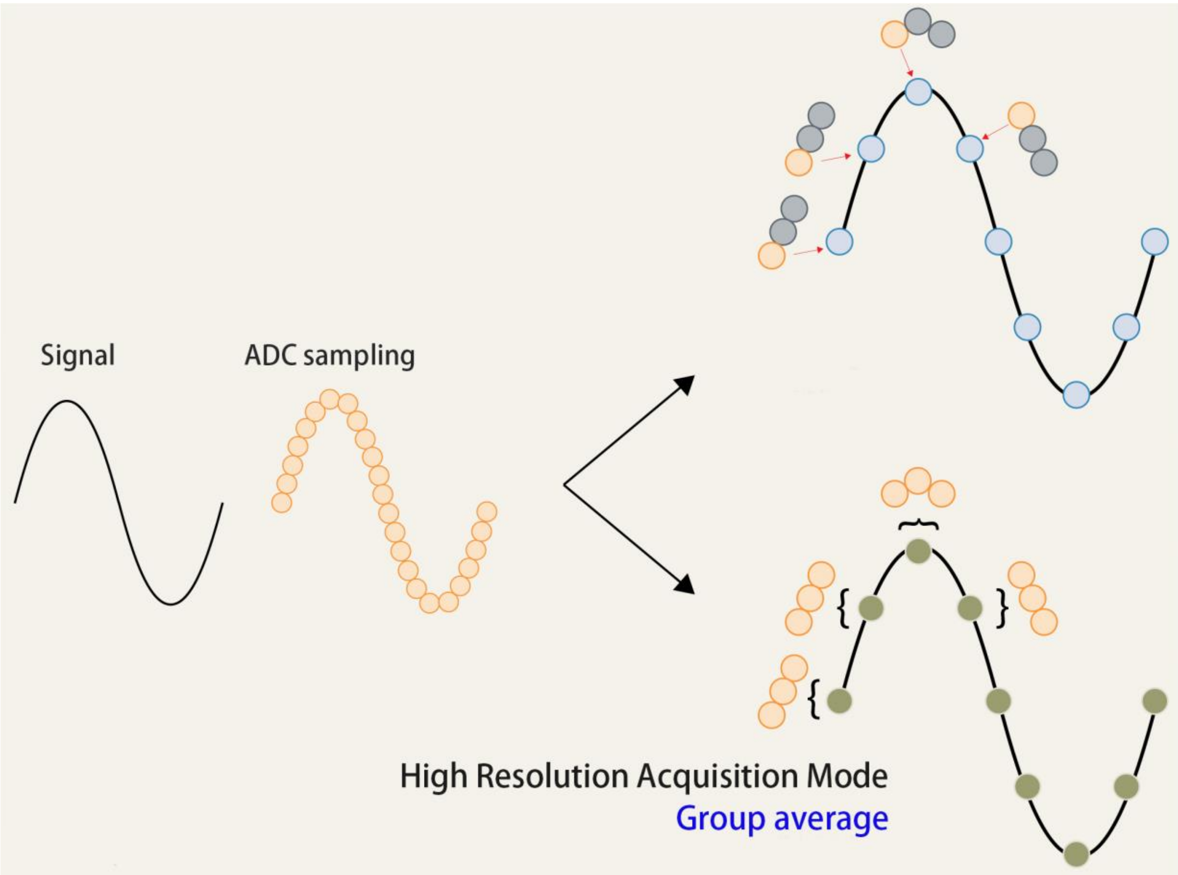


Peak Detect mode is best used for detecting glitches, viewing very narrow pulses or measures the amplitude range of the noise. For example, if the actual sampling interval of the oscilloscope is 2ms for one sample, the oscilloscope internally will use the maximum sampling rate and will collect / show 2 values every 2ms. These 2 values are the maximum and minimum values among n data points sampled within the interval of 2ms.

High-Resolution mode:



The High-Resolution sampling mode is an oversampling method that averages multiple adjacent samples to produce an averaged sample. This mode significantly reduces random noise and it is suitable for non-repeating waveforms and single-shot waveforms. Compared to the Peak Detect mode, the High-Resolution mode applies low-pass filtering to the signal, which cut out high-speed glitches.



> **Note**
> For displays with low resolution since the display area is too small, the software will move the mode settings window to the Advanced tab.

8.2.17 8 Bit ~ 13 Bit Vertical resolution

The High-Resolution sampling mode uses oversampling to improve vertical resolution from 8 Bit to about 13 Bit.

The input is sampled at a rate higher than the minimum required Nyquist sampling rate, fs. For example, when using an N-bit ADC without oversampling, an input signal of 100 Hz is sampled at 200 Hz (2 × 100 Hz) to get the digital output with the native ENOB of the ADC. When oversampling with a factor of k = 16, the same 100-Hz input signal is sampled at 3200 Hz (k × 2 × 100 Hz). The samples obtained by oversampling are low-pass filtered and decimated using a digital filter to achieve a reduction of the quantization noise. The signal at the frequency band of interest is not affected by the filter, and the result is an improved SNR. The improved SNR results in a higher ENOB performance.

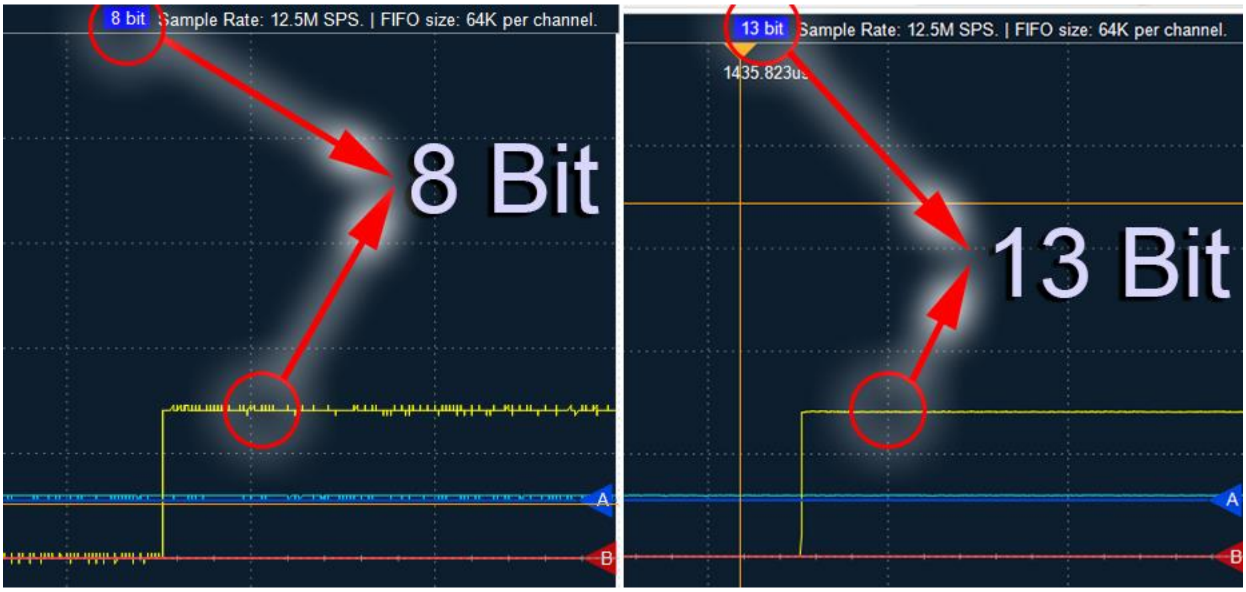
The group averaging in the high resolution sampling mode mentioned above is also essentially a digital filtering. The higher the sample extraction factor, the more the resolution is increased. When the extraction multiple is 16, it can be improved from 8 bit to 10 bit. Working in HiRes mode, the user cannot directly select how many bit resolutions to enhance, which dynamically changes with the time scale of the oscilloscope. The user does not know how much resolution is currently enhanced and how much bandwidth is dropped.

The steeper edges of the signal are easier to be under sampled in HiRes mode. For example, the signal edge originally has 16 sampling points. In order to improve a 2-bit resolution, it is necessary to average every 16 points. As a result, only one sample is left on the edge. Group averaging also significantly reduces the oscilloscope waveform update rate.

Group averaging can conditionally increase resolution without improving the accuracy of the oscilloscope. For example, a DC gain accuracy with a hardware 12-bit ADC oscilloscope can reach 0.5% of the ground. Using an oversampling and linear noise reduction technique to boost the oscilloscope to 12 bit resolution, the DC gain accuracy is still the level of an 8-bit ADC oscilloscope: ±2%.

Table 1. Relationship Between Oversampling Factor, SNR and Extra Bits of Resolution

Oversampling Factor, k	SNR Improvement (dB)	Extra Bits of Resolution
2	3	0.5
4	6	1
8	9	1.5
16	12	2
32	16	2.5
64	18	3
128	21	3.5
256	24	4
512	27	4.5
1024	30	5
2048	33	5.5
4096	36	6



8.2.17.1 Video demonstration：

<LOTO Oscilloscope || 8-bit to 13-bit vertical resolution>

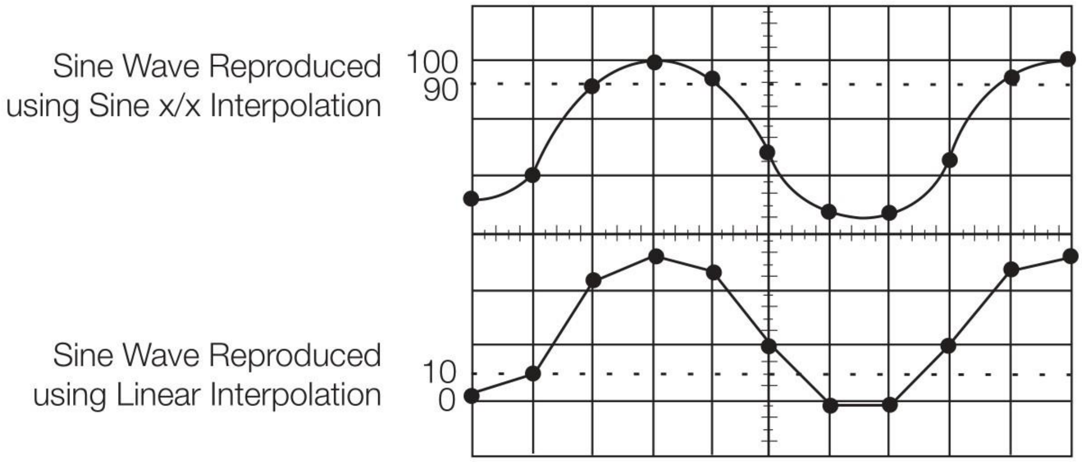
https://www.youtube.com/watch?v=8hX7UjYnypM

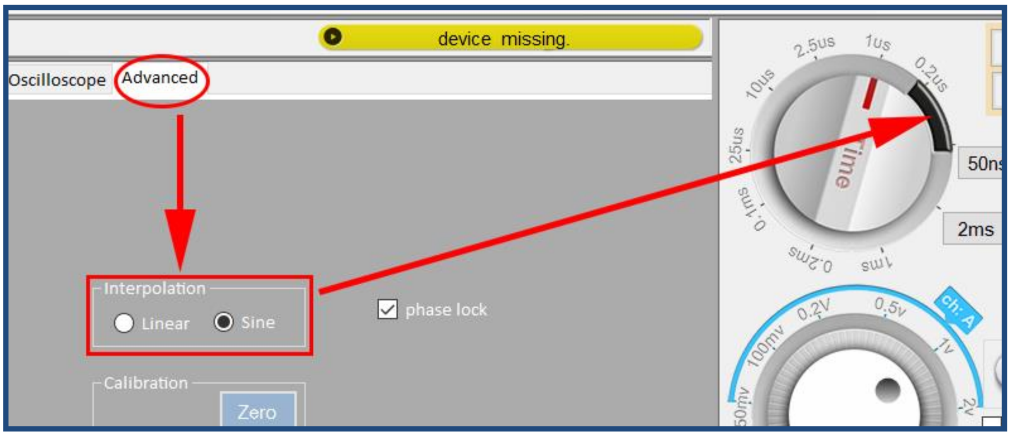
8.2.18 Sine / Linear Interpolation

The samples interpolation consists in filling the gaps between the ADC samples by inserting calculated values according to a specific algorithm, so to improve the visualization of the signal details.

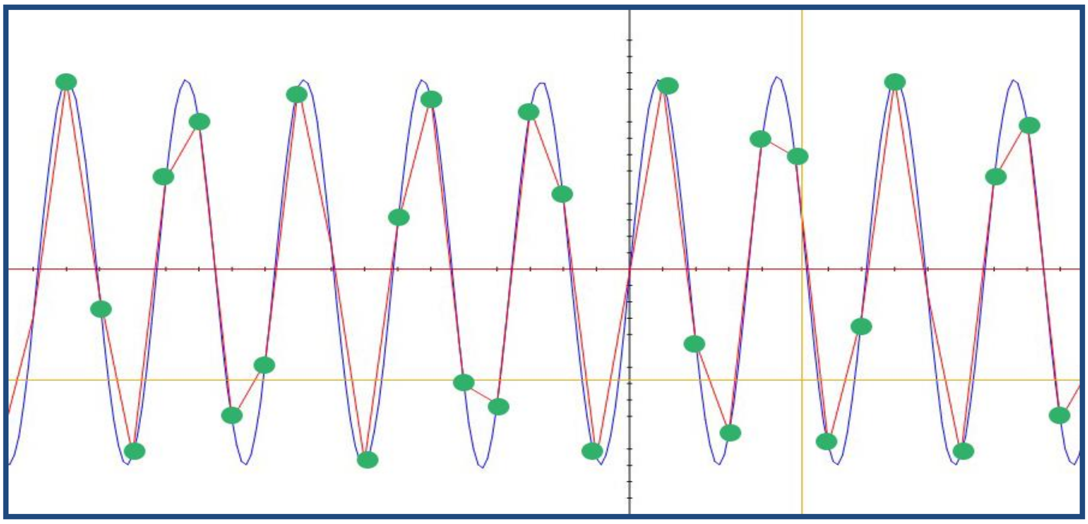
The highest real-time sampling rate of OSCxxx is different from each other. The default interpolation algorithm is performed with sine function 4x. For example, a device with 80MSa/s per channel, for smooth signals, it corresponds to a 320MSa/s of sampling rate.

Another available interpolation algorithm is the linear interpolation, a simple and light method which calculates the interpolated values with the linear interpolation among adjacent samples.





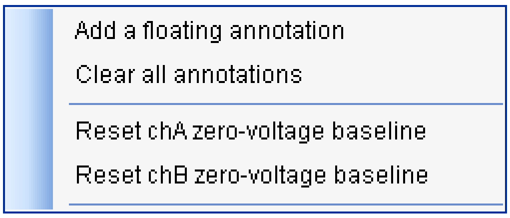
On the advanced page or the button in the lower right corner of the display page , you can choose among **Linear** or **Sine** interpolation algorithm. The selected algorithm will use the time slots (0.2us and 50ns) in the black sector on the time knob.



The green dots in the above figure are the actual ADC samples, the red curve is the waveform curve generated by the linear interpolation algorithm; the blue curve is the curve generated by the sine interpolation 4x.

8.2.19 Right-click mouse Menus

In the waveform display area, click the right mouse button and the following menu will appear.



8.2.19.1 "Add a floating annotation"

Please refer to 8.2.13.

8.2.19.2 "Clear all annotations"

Please refer to 8.2.13.

8.2.19.3 "Reset chA zero-voltage baseline"

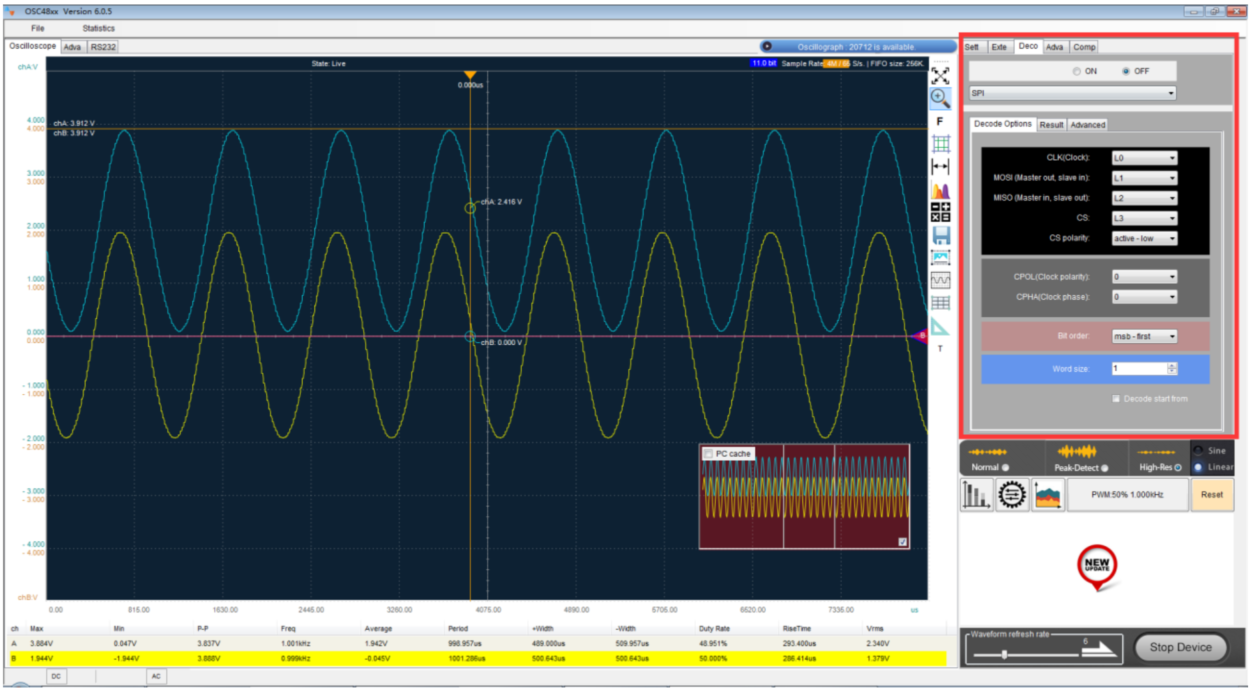
Restore the A channel baseline to the default position. (The default position is the middle position of the Y axis of the waveform display area.)

8.2.19.4 "Reset chB zero-voltage baseline"

Restore the B channel baseline to the default position.

8.2.20 Decode and Setting the decode parameters

Only several model devices support this feature. If you cannot find this feature in your software, it is not available for your device model. Usually, we may find this decoding feature in a "Decode" tab panel on the right.

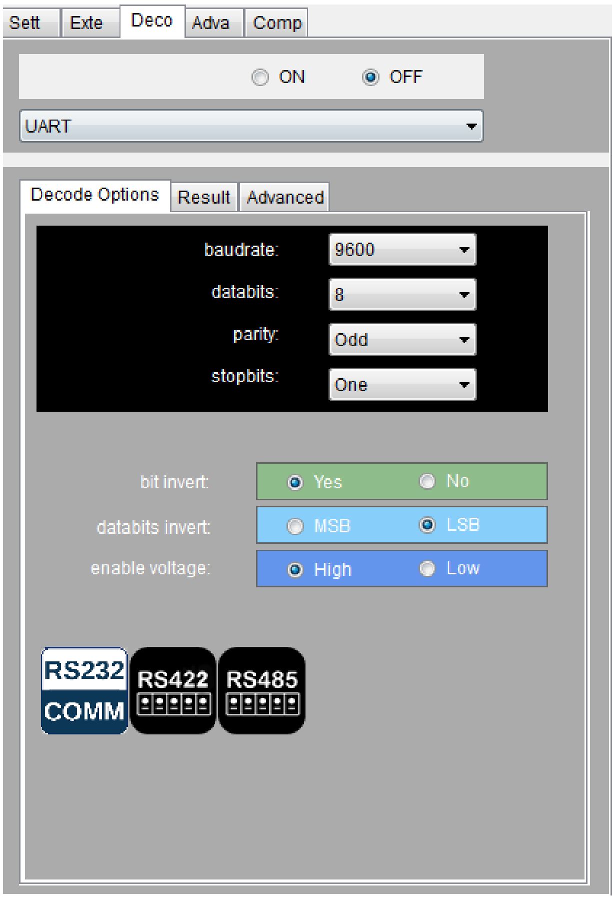


After using the trigger capture communication signal, the oscilloscope software enters the decode mode by selecting the corresponding communication protocol, such as "UART" or "IIC".

Clicking on the protocol menu item, "On" appears in front of the menu item, indicating that the communication protocol is used for decoding. Clicking "OFF" will terminate the decoding.

After the communication protocol is selected, click the right mouse button in the waveform display area. The "Setting the decode parameters" option in the right-click menu will be activated and changed to the usable state. Click "Setting the decode parameters" to pop up the protocol parameter setting window.

Usually, the software will display this setting window actively when the selected protocol is selected for the first time to remind the user to complete the parameter setting. Then, during the use, you can also use the options in the right mouse button menu to call out this window for parameter adjustment.



For different protocols, the setting items appearing in the protocol parameter setting window are different;

"Decode Options" panel item, select the basic parameters of the protocol.

The "Result" panel item displays all the decoding results of the entire buffer.

8.2.20.1 “bit invert”

Whether the data bit is processed by "bitwise inversion".

For example: 0x01, 8-bit binary is: 0000 0001.

bit invert Yes, the result is: 1111 1110 (bitwise inversion).

bit invert No， the result is: 0000 0001 (Normal, no inverted).

8.2.20.2 “databits invert”

The order in which each frame of data is sent is from the high bit or from the low bit.

For example: 0x01, 8-binary digits: 0000 0001(The leftmost 0 is the high position, and the rightmost 1 is the low position)

data bits invert LSB, the result is: 1000 0000(send from low position)

data bits invert MSB, the result is: 0000 0001(send from high position)

8.2.20.3 “enable voltage”

When there is a communication data signal, whether the data signal is high level or low level.

(if IO port defaults to low level when no data is sent, the data is high level when sending data; then you should choose "enable voltage high"). **For example: RS232.**

(if IO port defaults to high level when no data is sent, the data is low level when sending data; then you should choose "enable voltage low"). **For example: TTL.**

8.2.20.4 Failure or Error

Protocol parsing failure or error may be caused by the following reasons:

1. Protocol parsing parameter setting error;

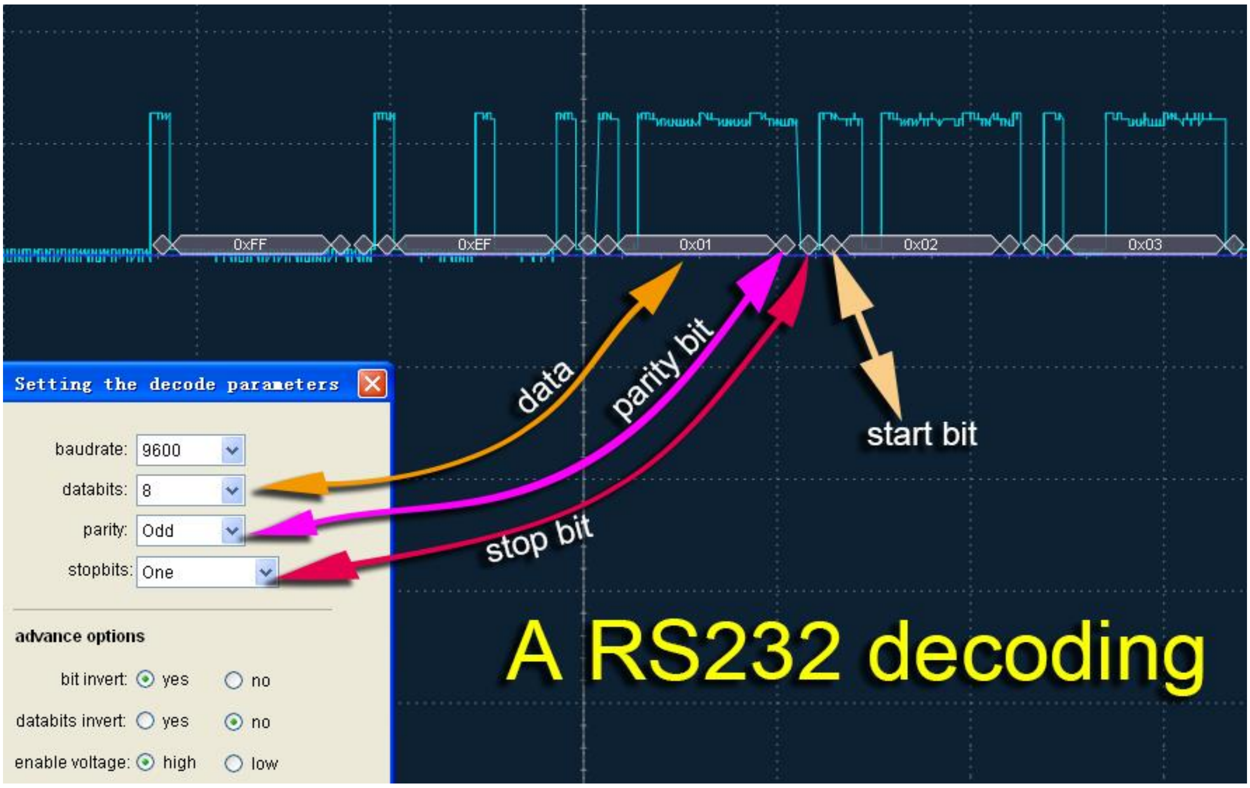
2. In the screen display area, the periodic waveforms are too dense to calculate the interval time correctly.

3. In the screen display area, the periodic waveform is too sparse. There is not a complete frame of data in the display area.

8.2.20.5 Examples

The following uses several practical examples to illustrate the decoding steps and the results displayed.

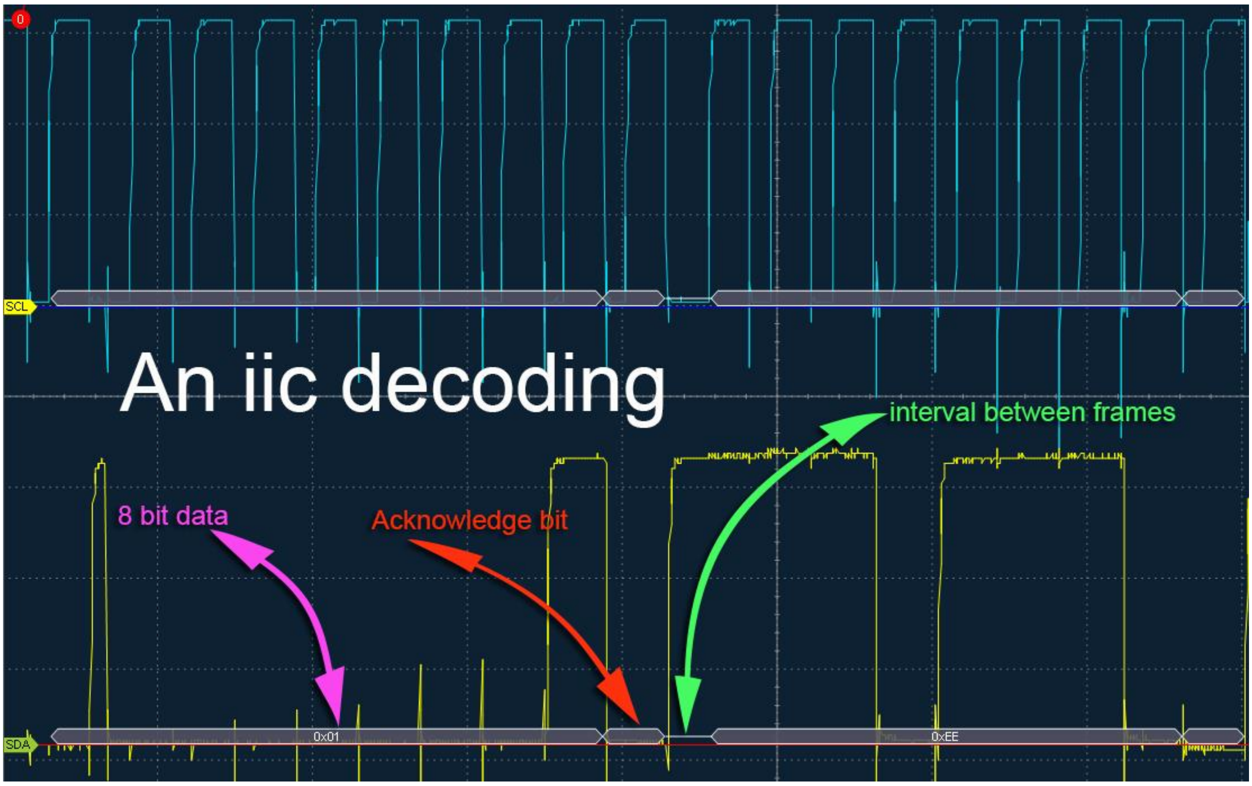
1. This is an example diagram of RS232 protocol decoding in UART communication:



Procedure	Description
1	BaudRate is 9600, which is set according to the frequency when the hardware sends data in this example. Calculated by: 1.0 / 9600 ≈ 0.000104166666666667. More accurate calculations are given to the oscilloscope software. We estimate that the final value is approximately: 104.17us. (How much time each bit takes to measure the length in seconds)
2	DataBits is 8bit, and the value is obtained according to the hardware circuit parameters in this example.
3	Parity is Odd. (If not, it can be set to "None" and the oscilloscope software will ignore it).
4	StopBits is One, and the value is obtained according to the hardware circuit parameters in this example.
5	Start bit + DataBits + Parity + StopBits = 1 + 8 + 1 + 1 = 11 bit.
6	11bit * 104.17us ≈ 1145.87us. (Time of each frame of data)
7	In the oscilloscope software Time gear, the X axis of the 1ms gear position takes 1ms per grid. The X-axis grid is about 1145us, which is a relatively appropriate time gear. The periodic waveform density interval in the screen is appropriate, which can read the decoded data well, and the success rate of oscilloscope software in analyzing data will be greatly increased.
8	For transient discontinuous signals, it is recommended to use the trigger function

	to capture; Use the software's pause and left and right move shortcuts (Ctrl + <-) / (Ctrl + —>) to help analyze the signal.
9	In this example, when RS232 sends a signal, each bit will be “bitwise inverted”, so select: “bit invert Yes”
10	In this example, each frame of data is sent from the high bit of 8 bits, so choose "databits invert No".
11	In this example, when no data is sent, the IO port defaults to low level and the data sent is high level, so choose "enable voltage high".
12	The data sent is: 0xFF 0xEF 0x01 0x02 0x03. Finally, according to the display results, the sending is correct.
13	What needs to be emphasized is Parity. If the check bit is found to be inconsistent with the check bit after analyzing the received waveform, it will automatically assume that this is a frame of error data and it will be discarded without the block diagram showing the parsed result (but with the waveform).

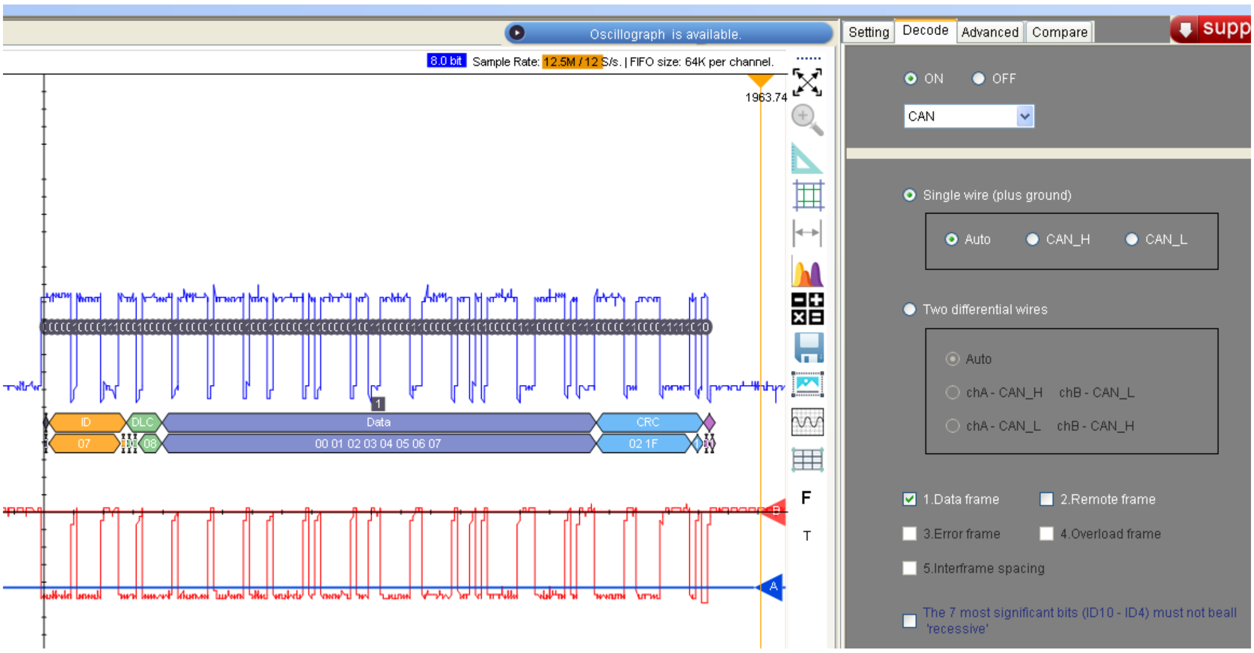
2. This is an instance drawing of IIC decoding:



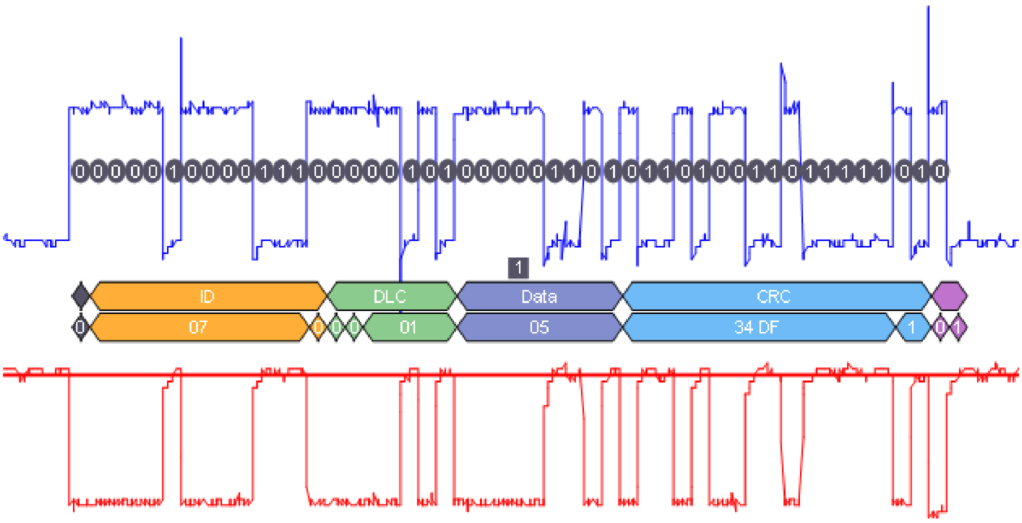
Procedure	Description
1	Probe of channel chA is connected to the SCL circuit.
2	Probe of channel chB is connected to the SDA circuit.
3	Select the appropriate oscilloscope software Time gear.
4	For transient discontinuous signals, it is recommended to use the trigger function to capture; Use the software's pause and left and right move shortcuts (Ctrl + <-) / (Ctrl + —>) to help analyze the signal.
5	In this example of IIC communication, each bit has not been "bit-reversed", so

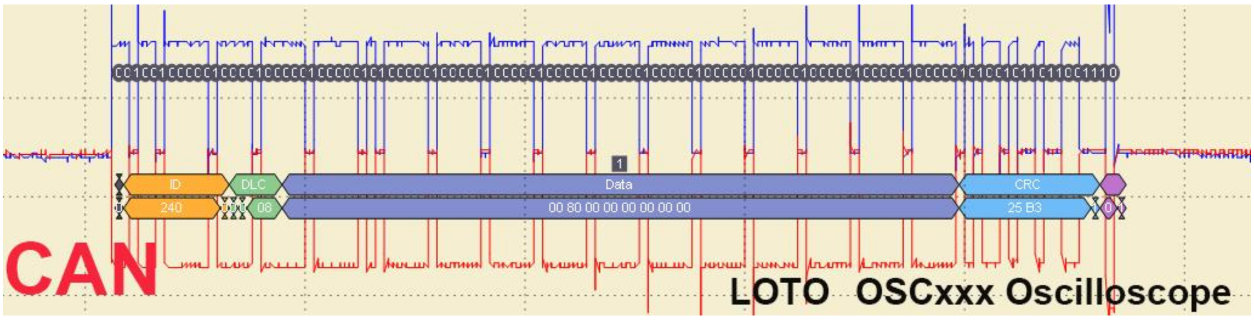
	choose "bit invert No".
6	In this example, each frame of data is sent from the low bit of 8 bits, so choose "databits invert Yes".
7	When selecting the oscilloscope software Time gear, at least ensure that one frame of complete data can be included in the software waveform display area.

3. This is a CAN decoding example diagram:

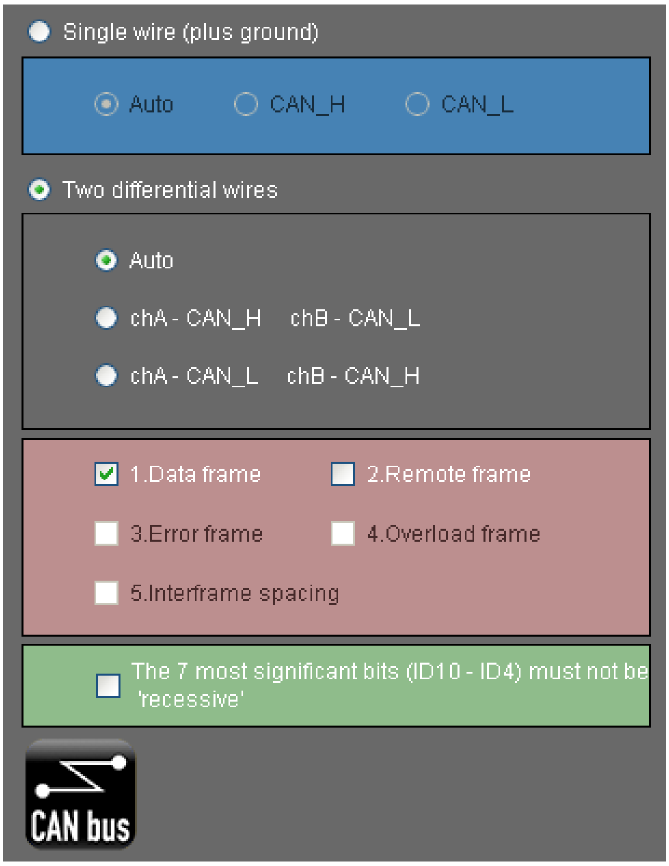


Automatically identifying the communication rate of the measured CAN signal during decoding; Supports any one or more of the five frame types of the CAN protocol for simultaneous decoding;





Automatic recognition of CAN_H line and CAN_L line is supported during decoding;



8.2.20.1 Video demonstration：

<LOTO oscilloscope software function demo "I²C Decoding">
https://www.youtube.com/watch?v=rmzBPBtL0-g

<LOTO Oscilloscope || How to use the build in RS232 COM Decoder?>
https://www.youtube.com/watch?v=A5Xwk5Z9jFM&t=155s

<LOTO oscilloscope SW function---Lin bus decoding operation demonstration>
https://www.youtube.com/watch?v=IqcQNQAvxYE&t=13s

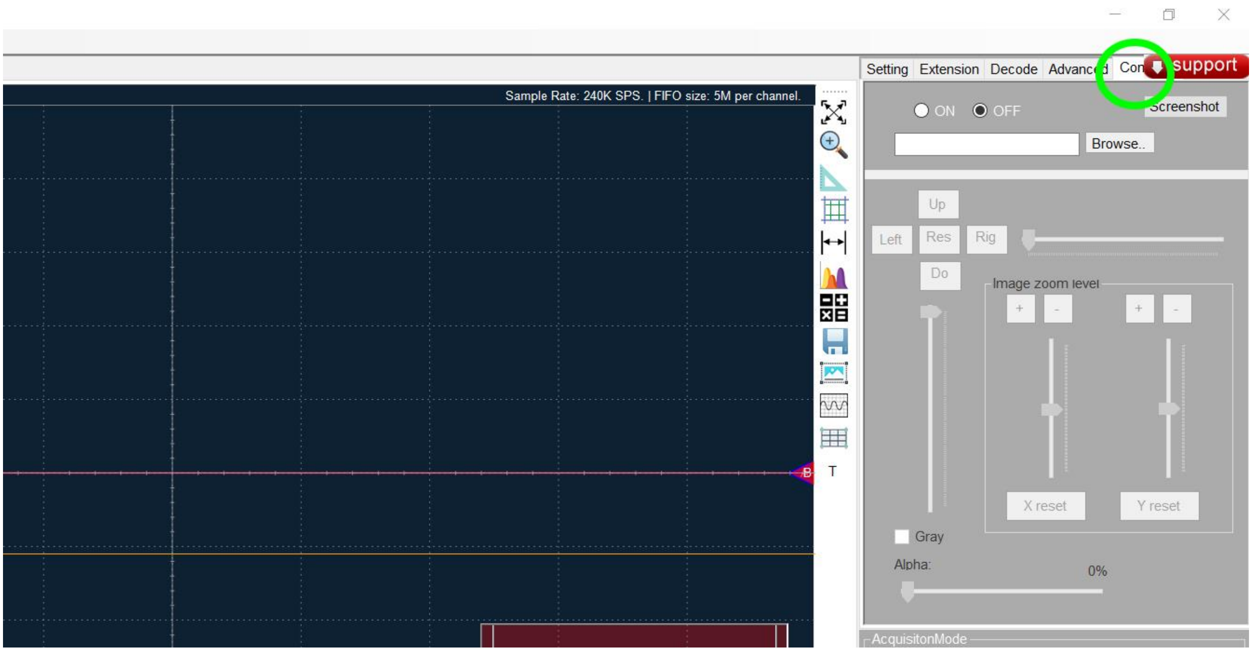
<LOTO oscilloscope UART RS232 decoding optimization>
https://www.youtube.com/watch?v=WoKt5OaquyE&t=4s

<LOTO oscilloscope software function demo "CAN Decoding">

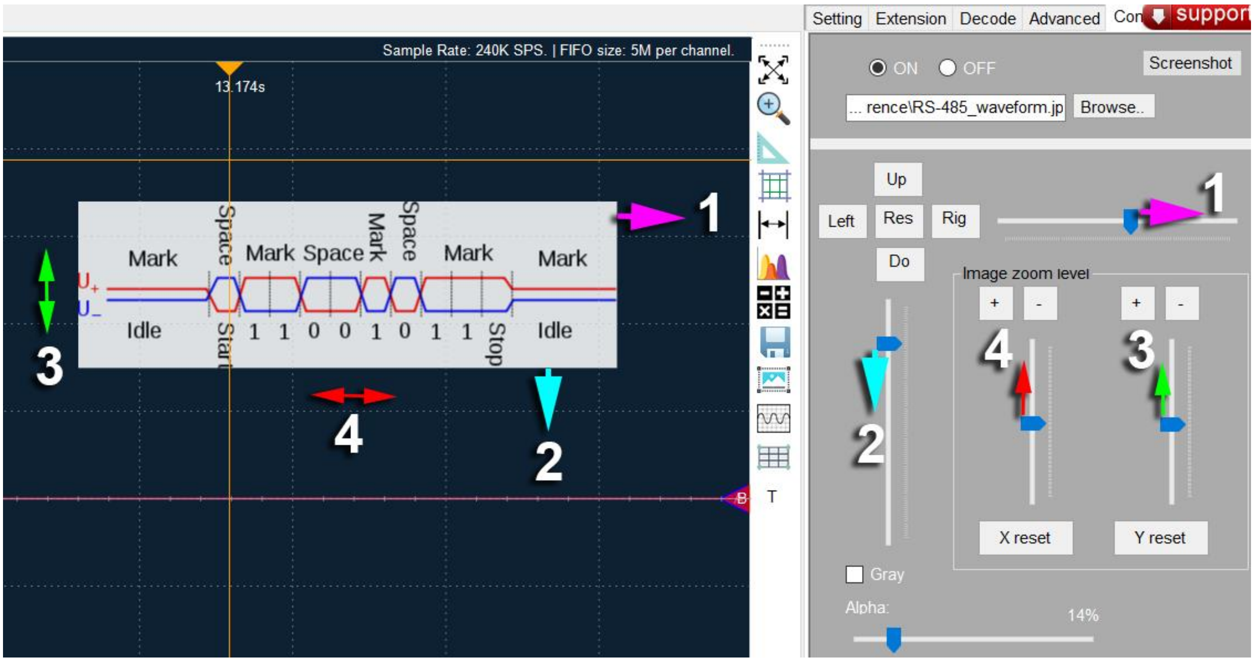
https://www.youtube.com/watch?v=1vRoUId8orU&t=783s

8.2.21 Reference comparison

This function allows users to select picture files from PC and import them into waveform plotting area as the background of acquisition waveform drawing. Used for real-time acquisition of waveforms and imported pictures for reference or comparison. The reference comparison function is shown in the tab page on the right side of the software as follows :



Select the "browse" button, import the picture file, and select the "on" button, the picture will be displayed in the drawing area as the background. The imported image can be moved left and right or zoomed in and edited, and changed to grayscale or transparency.



1	Move the Image to the right
2	Image down
3	Image enlargement in Y direction
4	Image enlargement in X direction

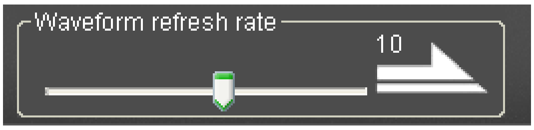
8.2.22 Software update

In the lower right corner of the display interface, click the red new update button to see the latest version of all series software of Loto oscilloscope, and the user can download the latest version of corresponding software.

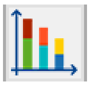
8.2.23 The function of waveform refresh rate function

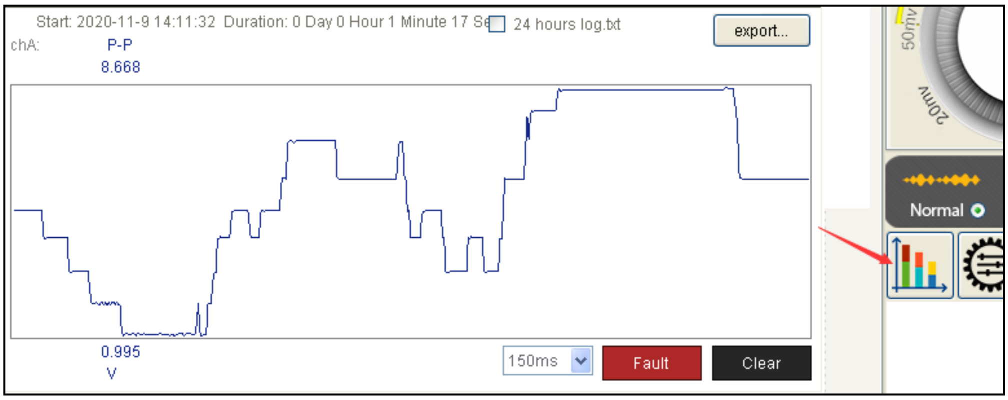
At the bottom of the lower right corner of the software interface, click the blue button of the waveform refresh rate to adjust the refresh rate of the oscilloscope measured waveform by moving the cursor left or right. A higher refresh rate can be selected on the display with better screen display effect to better display the waveform.

The actual display speed is related to the machine performance.

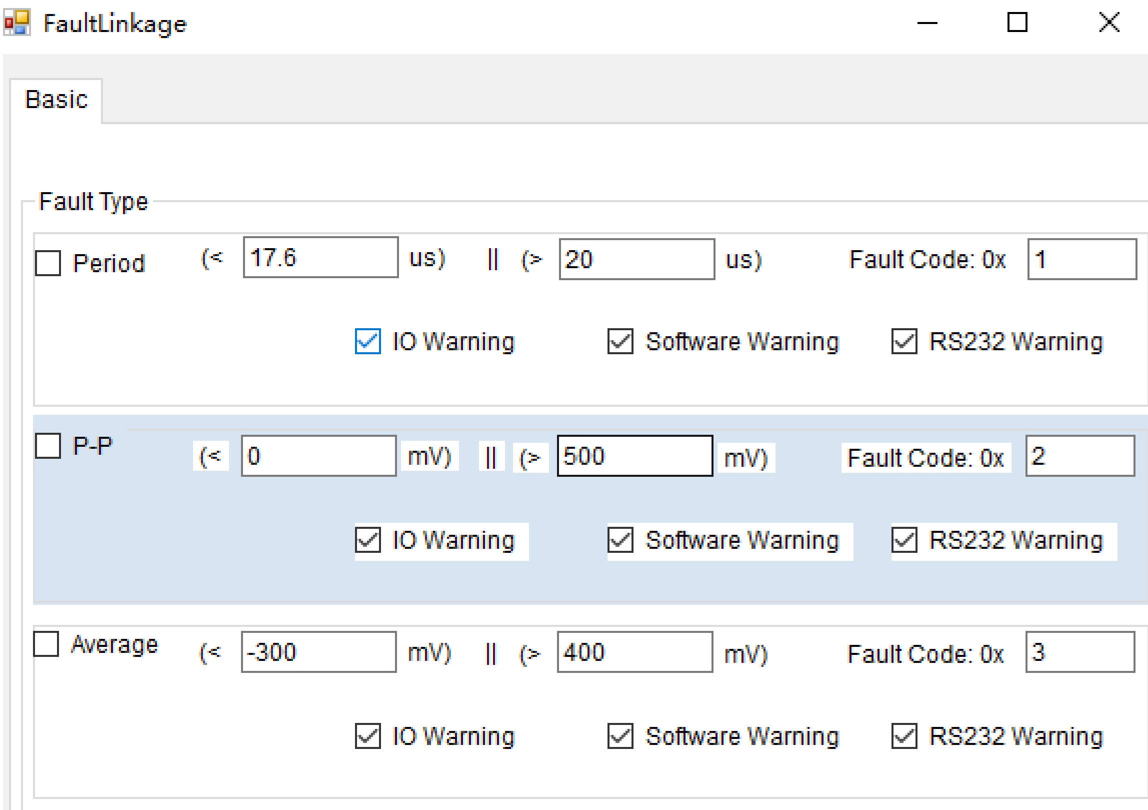


8.2.24 Variable statistical function of signal

In the lower right part of the software display interface, click  the button can realize the statistical function of any variable of the signal, for example: measure the change of the maximum value and peak value variable of a signal as shown in the figure below.



Click the fault button in the lower right corner of the signal variable statistics page, and a fault linkage dialog box will pop up. The user can freely set the fault range and select the alarm mode, such as IO warning, software alarm and RS232 alarm. It is very convenient for us to measure whether the signal is normal or not. The dialog page is shown in the figure below:

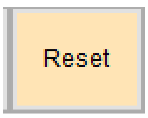


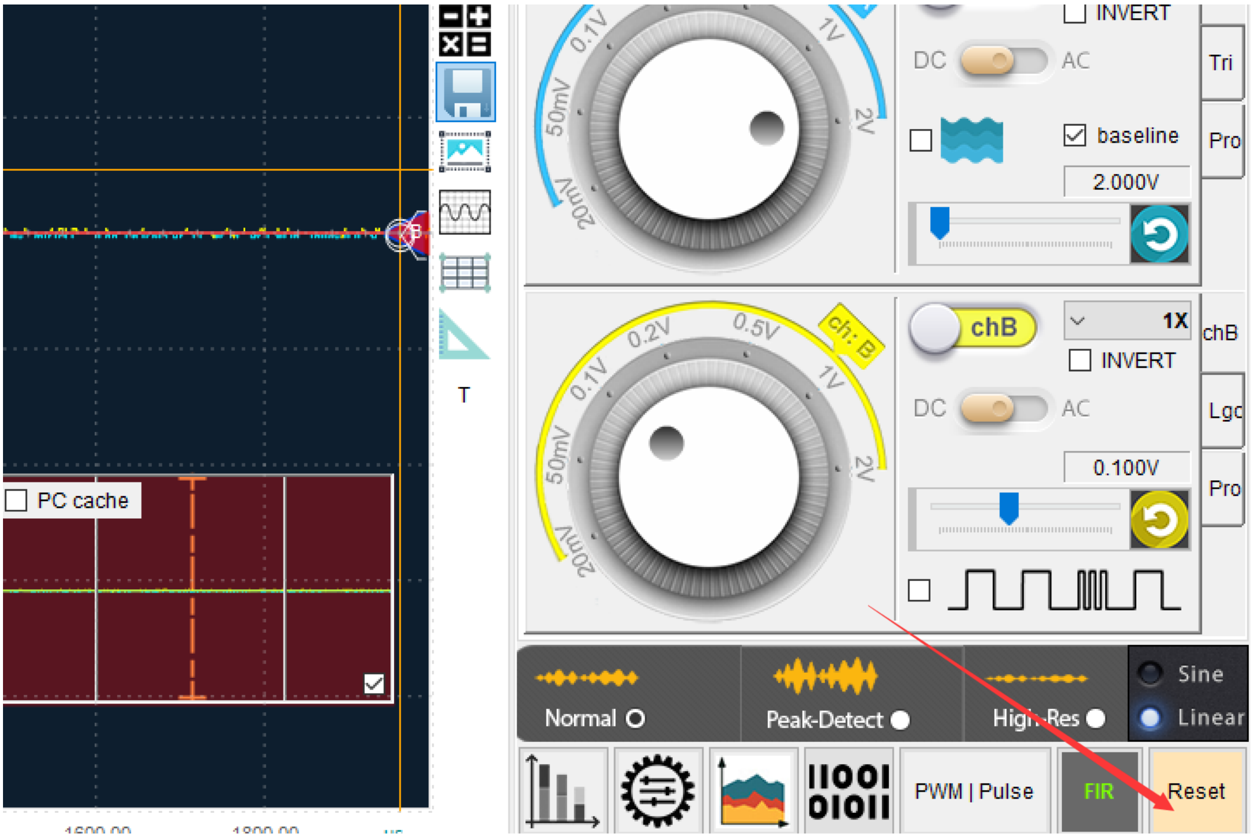
8.2.24.1 Video demonstration:

<LOTO Oscilloscope || Statistical curve & Fail/Pass detection :>
https://www.youtube.com/watch?v=Yus6_9tI018&t=78s

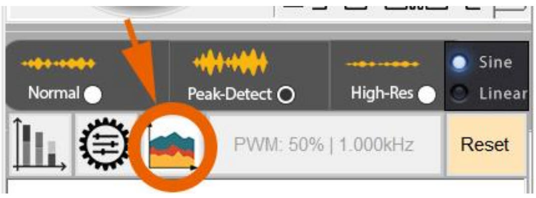
8.2.25 Reset button

The user clicks the reset button in the lower right corner of the software interface, which will restore the oscilloscope to the factory settings. When the oscilloscope crashes or other unexpected circumstances occur, the reset button can be used to reset the oscilloscope, and the reset button is shown in the figure below:

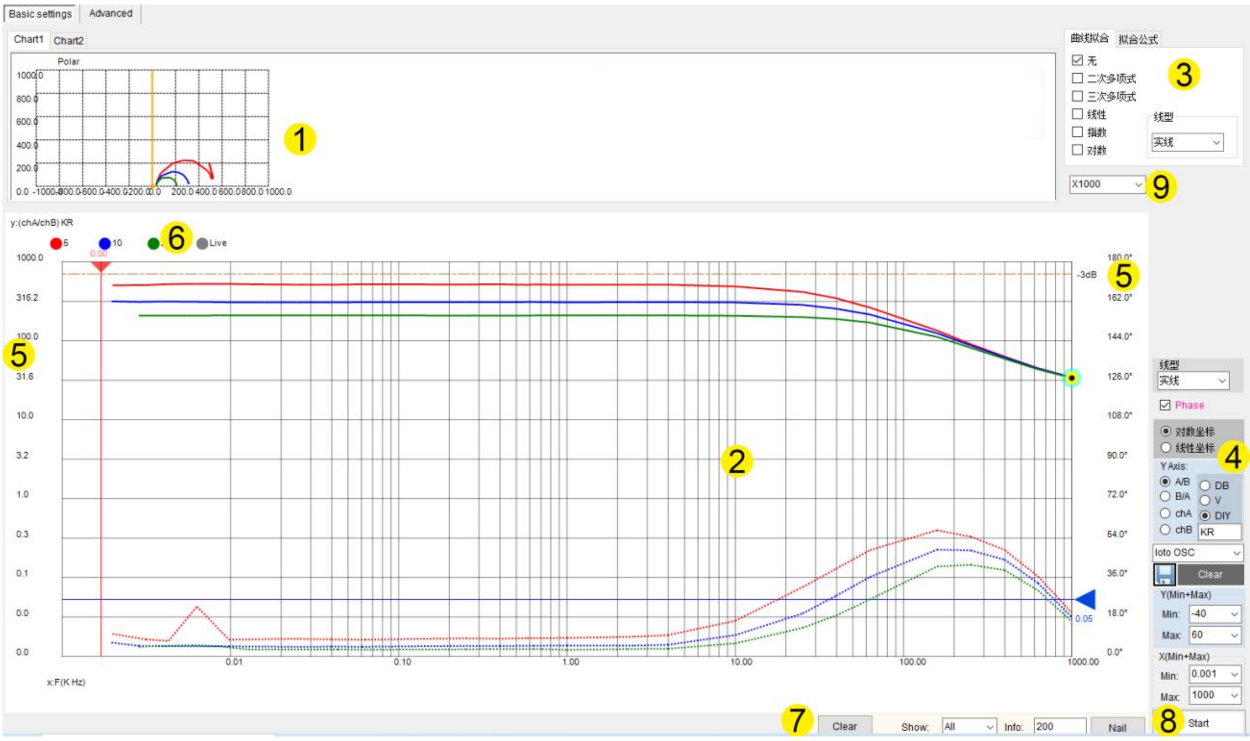




8.2.26 Frequency response curve mapping （20230708）



When this button is clicked, a program for measuring the frequency response will be started. If there is a customer whose screen resolution is insufficient to display the button at this location, the customer can find the independent program in the oscilloscope software directory and start it manually. The program running interface is shown below:

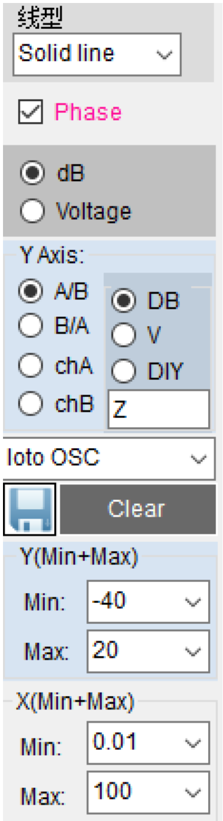


ID	description
1	Nyquist plot/polar plot display area
2	Frequency response curve (amplitude frequency/phase frequency) display zone
3	Curve Fitting Settings zone
4	Frequency response display setting area, see 8.2.26.1 for details
5	Amplitude Coordinate/Phase Coordinate
6	History Curve Marker
7	History curve setting zone, see 8.2.26.2 for details
8	Start/pause frequency response curve drawing
9	Vertical axis multiple display

The frequency response curve function is generally used together with the frequency sweep. The frequency sweep function of the signal generator gradually changes the signal frequency from the initial value to the set value. Coordinate, at the same time, you can select the amplitude of the detection channel or the amplitude ratio of the two channels as the ordinate, so as to draw the frequency response curve. It should be noted that this function is used in conjunction with the oscilloscope function. You can pause at any time during the mapping process to allow time for the oscilloscope to switch gears or do other preparations.

In the signal generator sweep function, we have introduced the "Freq response auto" function. During the scanning process after checking, the oscilloscope will cooperate with the signal generator to automatically switch the appropriate time base gear.

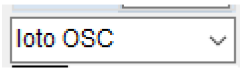
8.2.26.1 Frequency response curve (amplitude frequency/phase frequency) display zone (20230708)



Line style: Choose whether to use points or connect adjacent points with lines when drawing the frequency response curve.

Phase: After checking, the phase curve will be displayed at the same time, and it will always be displayed with a dotted line.

Y Axis: Set the data source of the ordinate of the amplitude-frequency curve, which can be chA, that is, the peak-to-peak voltage of channel A; or the peak-to-peak voltage of channel B, or the gain of dividing A by B, or B/A gain. For the display of the coordinate scale value of the ordinate, you can choose DB (logarithmic value display) or V (voltage value display) or DIY (custom display)



Here you can choose whether the current frequency response curve is to use the data measured by the LOTO oscilloscope or to use simulated fake data. The default is to use the measured data of the LOTO oscilloscope, and the simulated fake data is mainly used for demonstration and self-study


Save button, you can choose to save the data of one or all frequency response curves that have been drawn.

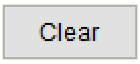
This button is to clear the frequency response curve that is currently in progress or completed, and place the starting point on the leftmost side of the drawing area, so that it is convenient to start the latest frequency response curve from scratch next time. This function is important and is used every time a new frequency response curve is started.

DB/Voltage: The horizontal and vertical coordinates of the amplitude-frequency curve use linear or logarithmic.

Y(Min+Max): The range of the ordinate of the amplitude-frequency curve, that is, the lower limit and upper limit of the coordinates. Here is where the user can set the coordinate display range. If the customer chooses the logarithmic method to draw the frequency response curve, then Y here is the logarithmic value. If linear plotting is selected, the value here is the corresponding voltage or gain.

X(Min+Max): The range of the abscissa of the amplitude-frequency curve, that is, the lower limit and upper limit of the coordinates. This is where the user can set the coordinate display range. If the customer chooses the logarithmic method to draw the frequency response curve, then X here is the logarithmic value. If linear drawing is selected, the value here is the corresponding frequency, and the unit is KHz.

8.2.26.2 History curve setting (20230708)



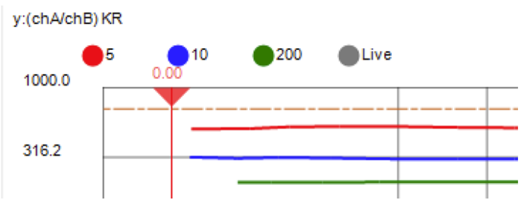
This button is different from the Clear button mentioned above. Click this button to clear all saved and displayed historical frequency response curves.

Show: Here you can choose which frequency response curves to display, you can display all of them, you can also select one of the historical curves or only display the frequency response curve currently being collected.

Info: When the acquisition of the frequency response curve is paused, you can enter the name of the current frequency response curve here, and after saving the historical curve, this curve will be associated with this name.

This button is to save the current (live) frequency response curve into the history curve, and use the above Info to set its name.

8.2.26.3 History Curve Marker (20230708)

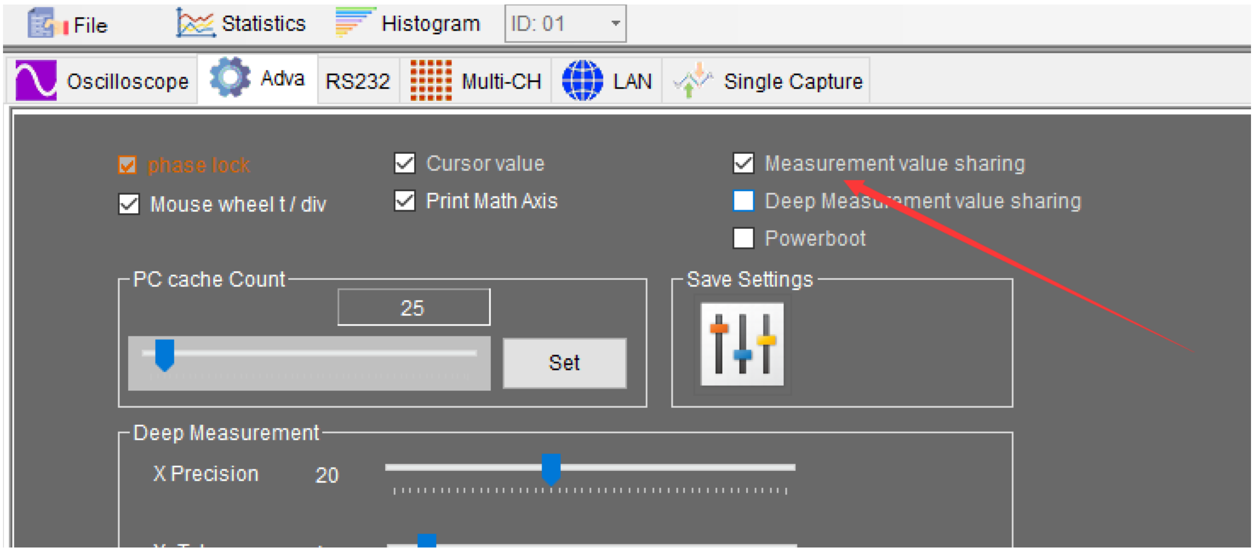


The frequency response curve will use different colors every time to distinguish it. As mentioned above, after saving the frequency response curve as a historical curve, a name will be defined by the customer. The color and name will be displayed here, which is convenient for customers to distinguish intuitively the case of the curve.

8.2.26.4 Problems that may arise (20230708)

Note: At the beginning of the scan, we need to click the "Clear" button several times to clean up the unnecessary parts that are not expected.

When using the frequency response curve function, you need to turn on the [Measurement Value Sharing] function:



Refer to Appendix IV: LOTO Oscilloscope Detection Frequency Response Characteristic Curve and 11.4 Signal Generator Sweep Function Setting Section of this document for the introduction of using signal generator sweep to achieve frequency response curve drawing.

8.2.26.5 Video demonstration：

<LOTO oscilloscope frequency response curve sweep and mapping function introduction>

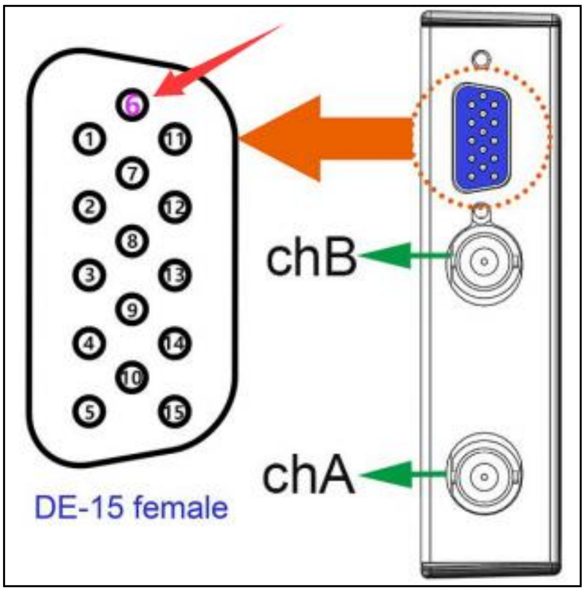
https://www.youtube.com/watch?v=UJpgIUxasKw

<LOTO oscilloscope signal source ||frequency sweep optimization||frequency response curve>

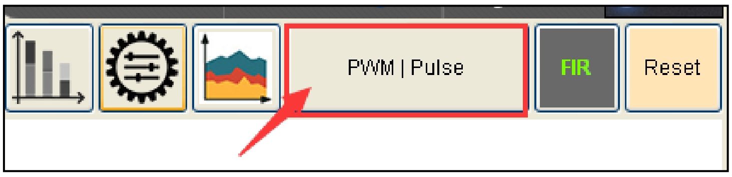
https://www.youtube.com/watch?v=frLPk0uoD_c&t=11s

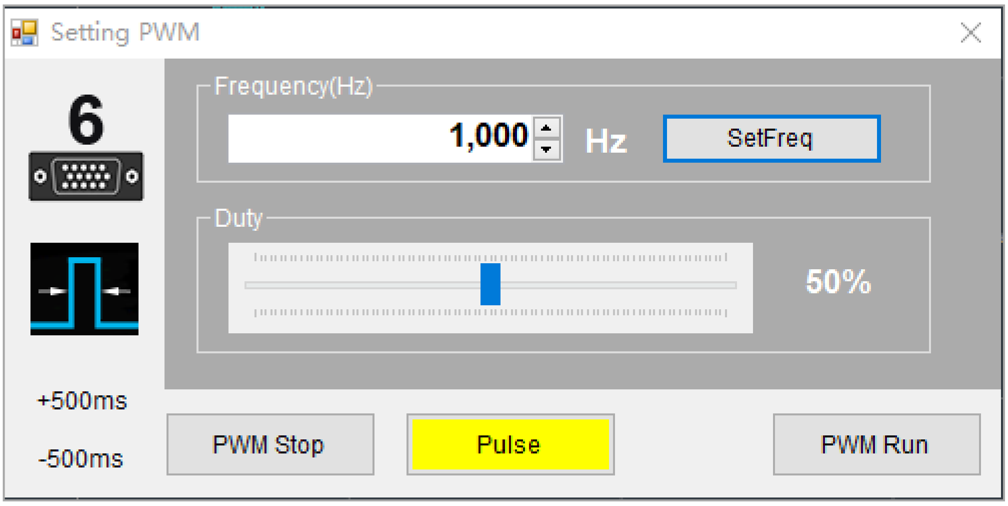
8.2.27 Standard square wave/PWM/ pulse

After the oscilloscope is successfully powered on and connected to the software. The 6-hole DE1-15 of the oscilloscope defaults to output a 1Khz square wave of about 1.5V, which is convenient for users to perform probe compensation and calibration.



Click the PWM/Pulse button, and the PWM setting window will pop up. In the window, you can adjust the standard square wave to a PWM wave, or you can modify the PWM wave to send a single pulse.





Send a single pulse:

1. Set the frequency and duty cycle.

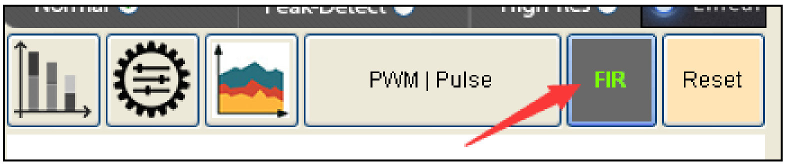
2. Click the PWM Stop button to stop generating continuous PWM waves.

3. Click the Pulse button to send a single pulse.

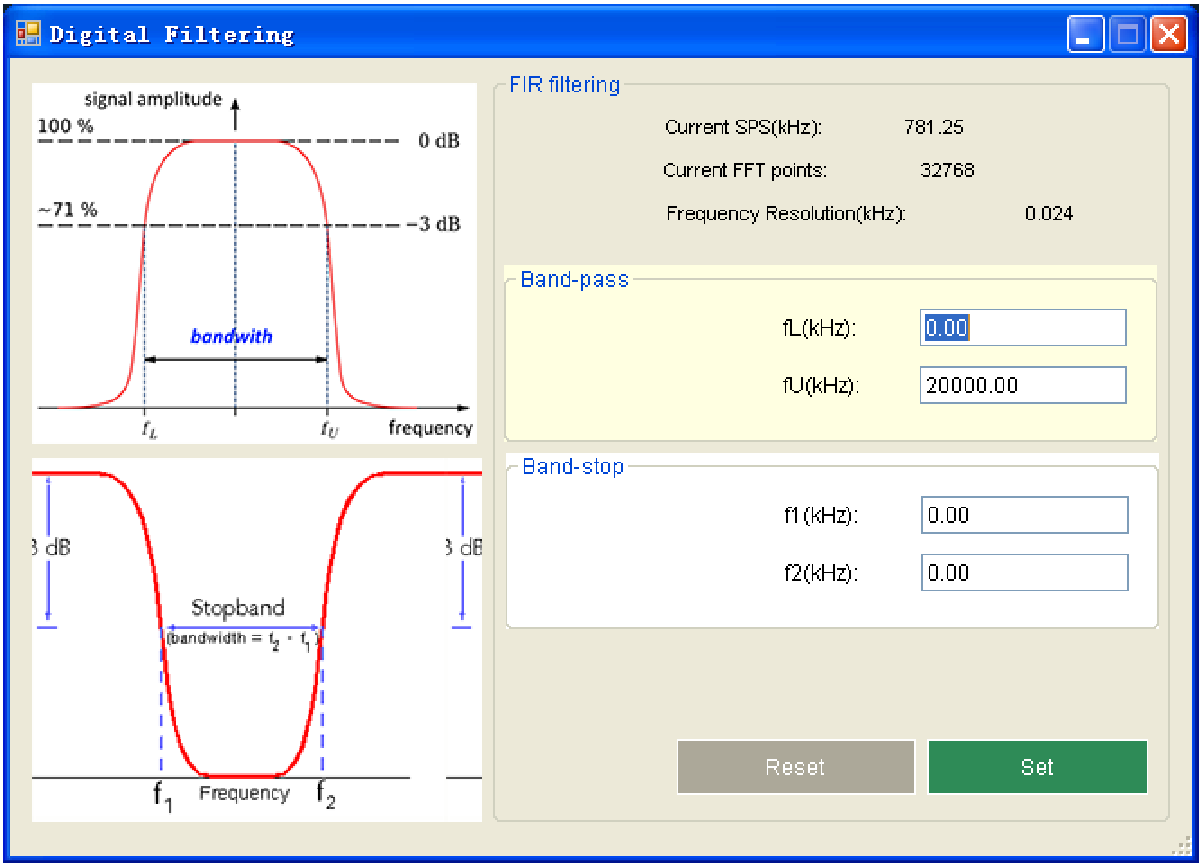
If you want to resume PWM wave sending, just click the PWM Run button.

8.2.28 FIR digital filtering

Click the FIR button to open the FIR digital filter window. This digital filter is based on FFT.



In the pop-up window, you can choose to set Band-pass or Band-stop, or a combination of both. After setting the filter frequency range, click the set button to take effect, and the waveform display area of the oscilloscope will display the filtered waveform.



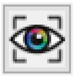
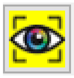
Click the Reset button to restore the default state. Close the window to close the FIR digital filter function.

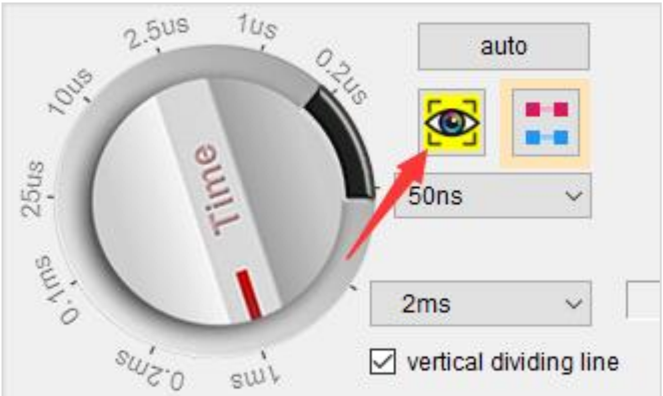
8.2.28.1 Video demonstration：

<LOTO Oscilloscope SW demonstration "FIR digital filter" function>
https://www.youtube.com/watch?v=riojFx8cat4

8.2.29 Eye Diagram

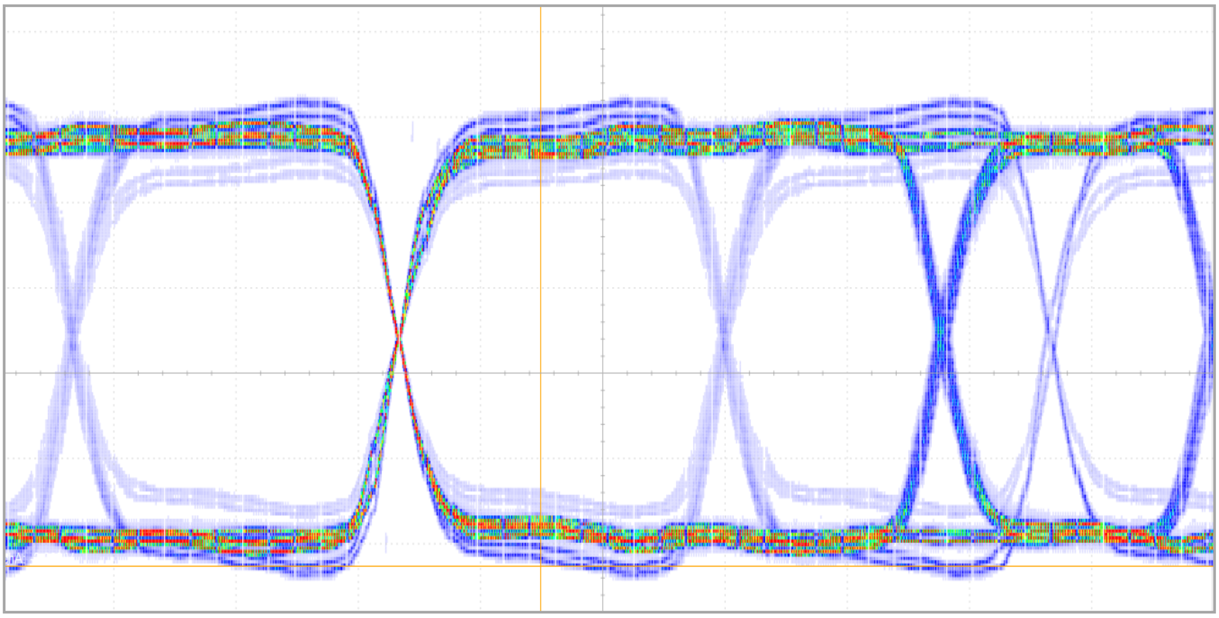
Among LOTO's oscilloscope products, some products support eye diagram analysis. It is used to analyze the influence of inter-symbol interference and noise of signal waveform on system performance, and observe the transmission performance of the system.

Click  the button to turn on the eye diagram analysis function, and click  the button again to turn off the eye diagram function.



According to the performance difference of different customers' computers, the final visual effect of the eye diagram will be different. As the opening time continues to increase, the effect will get better and better. The color gamut level of the eye diagram:

increasing from left to right.



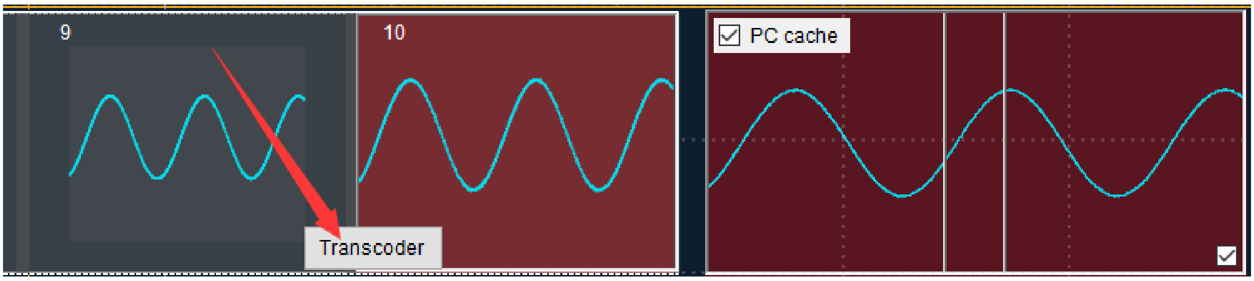
8.2.29.1 Video demonstration：

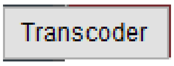
<LOTO oscilloscope --- eye diagram inroduction>
https://www.youtube.com/watch?v=v5UklsXs7Do

8.2.30 Historical data transcoding (PC cache-→TXT/CSV)

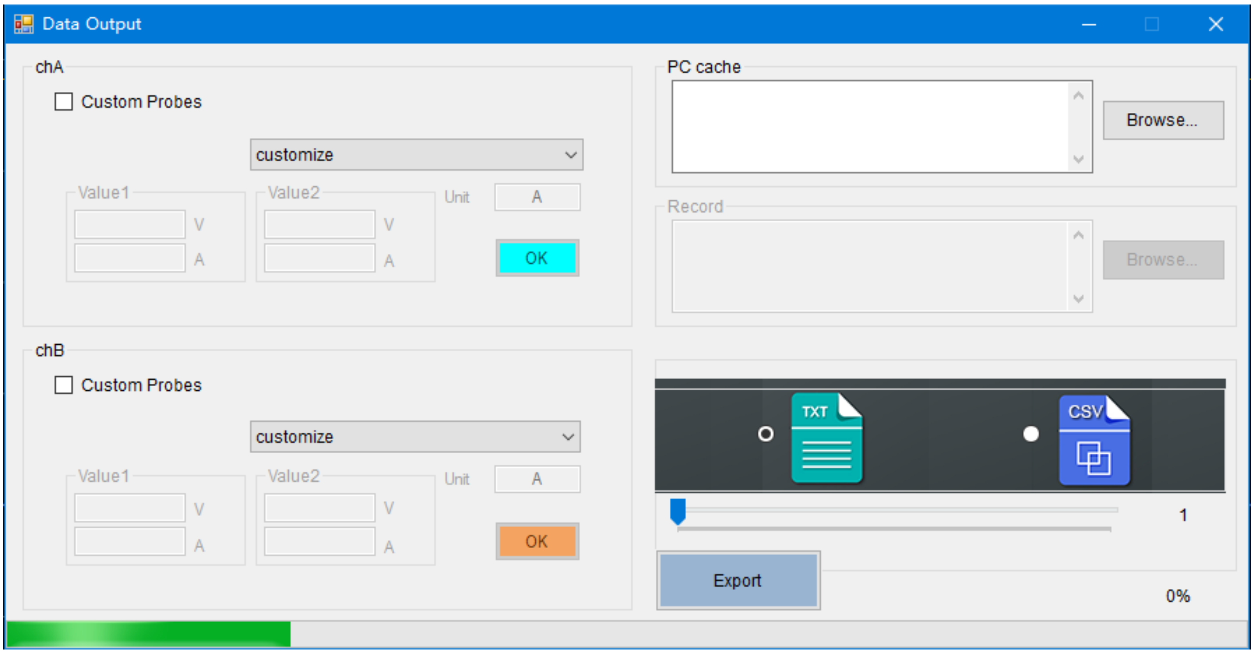
This function can export historical video files or historical PC cache files into voltage format Txt or Csv file sets. The user can use this function to convert the data collected by the oscilloscope to other data formats, which is convenient for reading and analyzing with other data analysis tools .

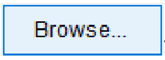
The function open button is located at the bottom right of the multi-frame display window of the PC cache function, as shown in the figure:

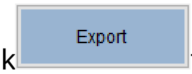


When the 

button is clicked, the transcoding export function window will start and display, as shown in the figure:



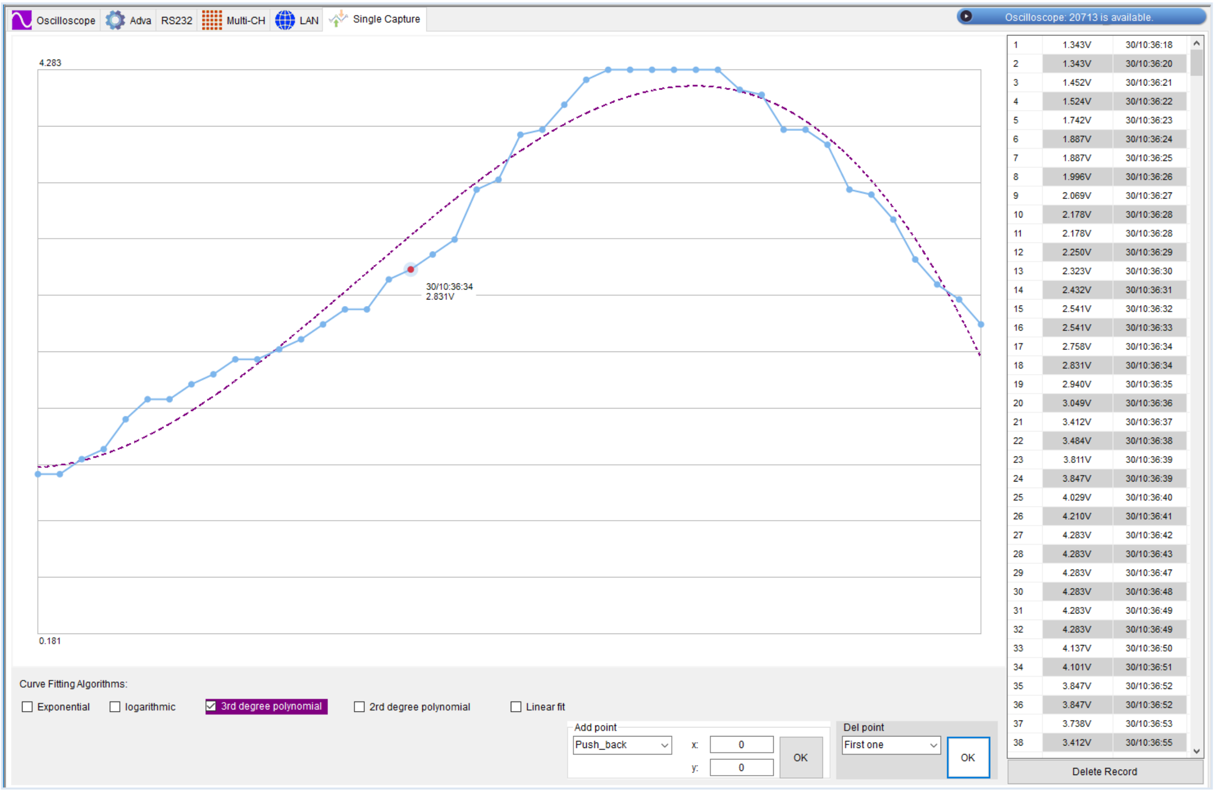
Click 

the button to browse the files saved by the historical oscilloscope. After completing the corresponding settings, click 

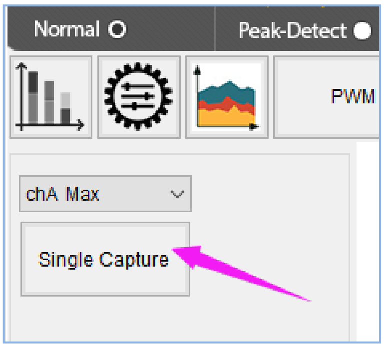
the button to complete the transcoding export.

8.2.31 single acquisition data point curve fitting function

During the running process of the oscilloscope software, when the user is interested in a certain measurement value at a certain moment, the measurement value value at the current moment can be recorded, and the software will draw and display multiple measurement value data points; when necessary At that time, it is possible to perform (exponential, logarithmic, 3rd degree polynomial, 2nd degree polynomial, linear) algorithm fitting on the curve set composed of data points, and calculate a curve with a certain meaning. It is also possible to manually add and delete data point sets.



During the running of the oscilloscope software, click the "Single Capture" button to add data value points to the single acquisition drawing interface.



8.2.31.1 Video demonstration：

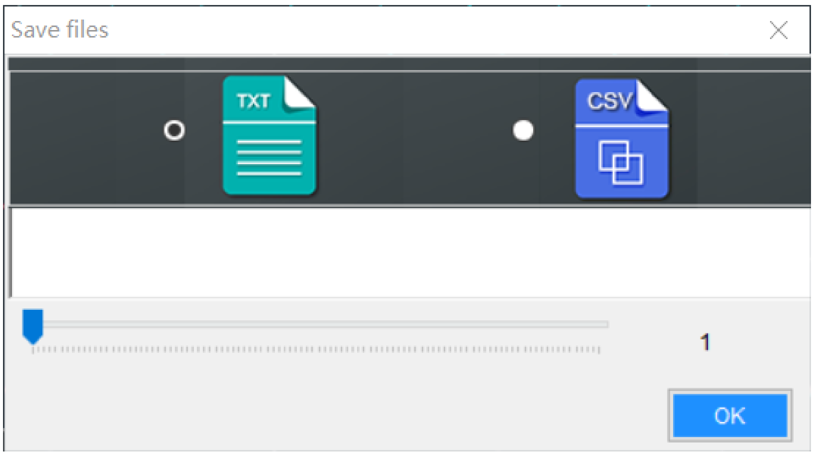
<LOTO Oscilloscope---Single Acquisition and Curve Fitting -- virtual oscilloscope>
https://www.youtube.com/watch?v=7c7LYthM4v0&t=1s

8.2.33 Waveform data file saving （TXT/CSV）

Click the button , the data collected by the oscilloscope will be saved in the user's computer in text format (.txt/.CSV), and the saving path can be customized by the user.

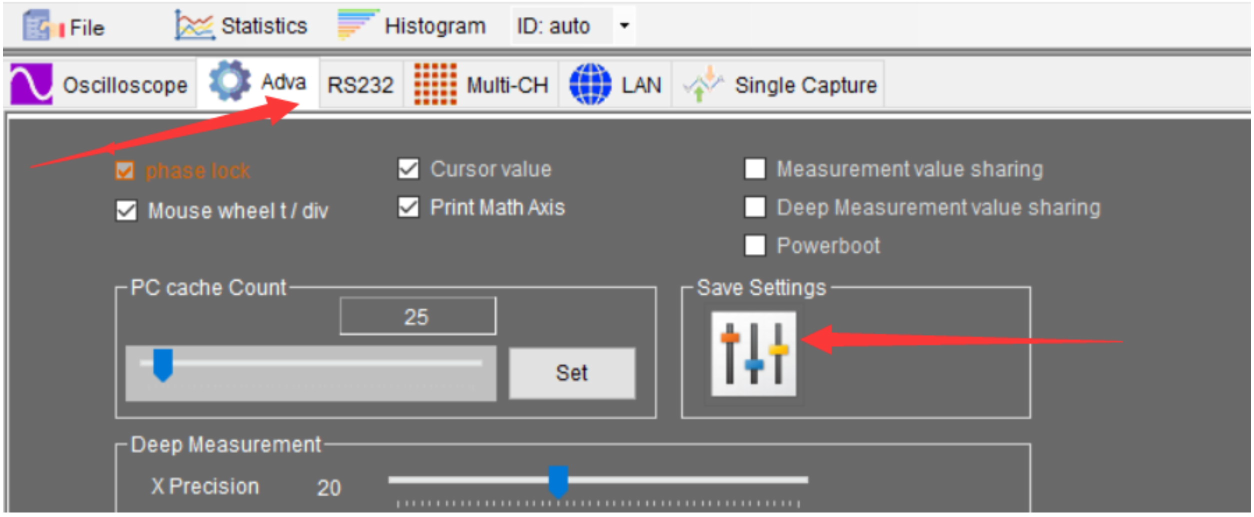
As shown in the figure above, click the save button to save the waveform data of part A displayed in the drawing area, that is, part of the data in the cache, and the waveform data of part B in the lower right corner of the figure, to the computer in the form of a text file superior.

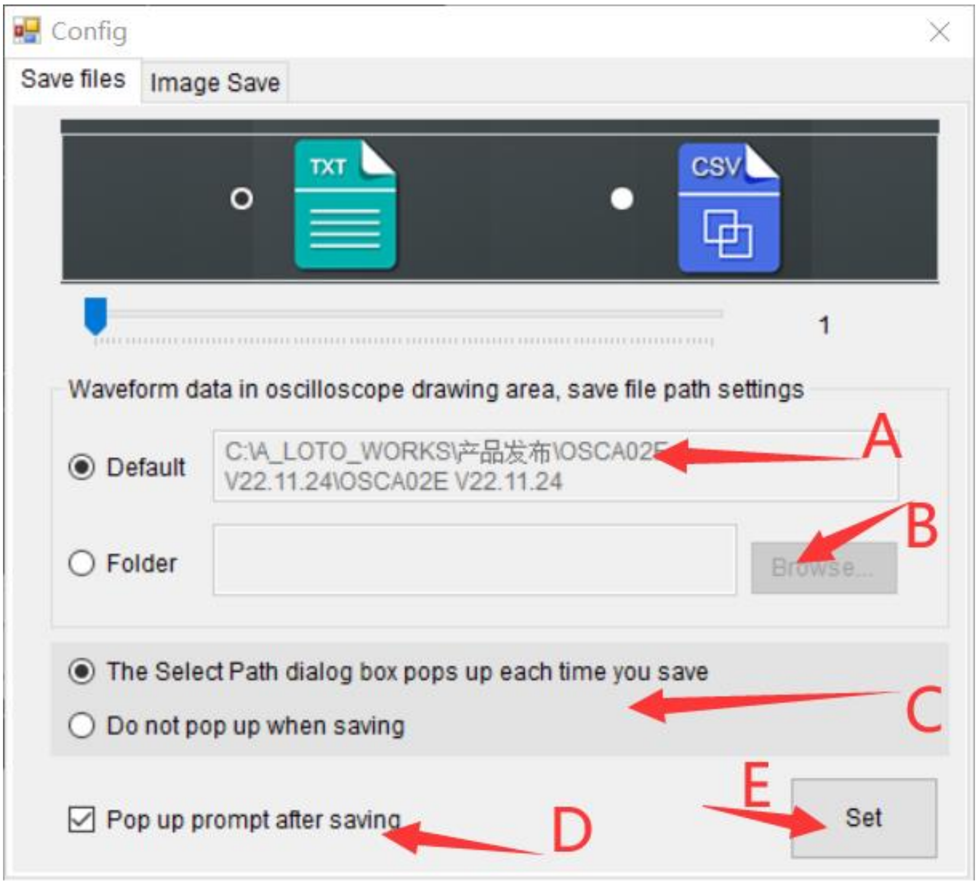
Click the save button, and the following dialog box will pop up by default for the user to choose to save as a TXT file or a CSV file. The former can be opened with the Notepad or notebook that comes with the computer, or it can be opened with a third-party text editor. The latter can be directly opened into an excel table style.



In the figure above, the slider is used to select data points to discard. The default is 1, which means that every data point is saved. If it is 100, it means that when saving, one data point is randomly selected for every 100 data points to be saved. This reduces the file size, but at the same time equates to a several times lower sample rate.

We can also make further settings for file saving, as shown in the figure below, the advanced settings for file saving can be found in the "Advanced" tab:





As shown in the figure above, A is our default save path, which is in the same directory of the oscilloscope PC software, and B is a customer's own save path that can be specified.

Choose one of these two paths, and at C When "Save without popup every time" is selected, the oscilloscope software will store the waveform data displayed on the screen after the customer clicks the save button in this alternative path.

When C is selected as "Every time you save, a dialog box for selecting a path will pop up", the settings in A and B are invalid, and the software will save according to the path selected by the customer.

After the data is saved, whether a prompt dialog box will pop up is set at D. After all the settings are completed, click the setting at E to save the setting.

8.2.34 Waveform color settings

The upper computer software of the LOTO oscilloscope computer can set some colors related to the waveform display. These colors can be saved after setting, and can also be reset to factory default. As shown below:

The opening position of the color setting is at 1 in the figure, and clicking it will open the setting dialog box shown in 2. We can set the corresponding color, and then click the 5 next to it to make the set color take effect immediately. Finally, the color effect has reached the expectation.

Click the "Save" button at 3 to save your color settings to memory and use them by default in future software.

We set a "reset" button at 4, and clicking it will restore the color configuration we set.

8.3 data sharing

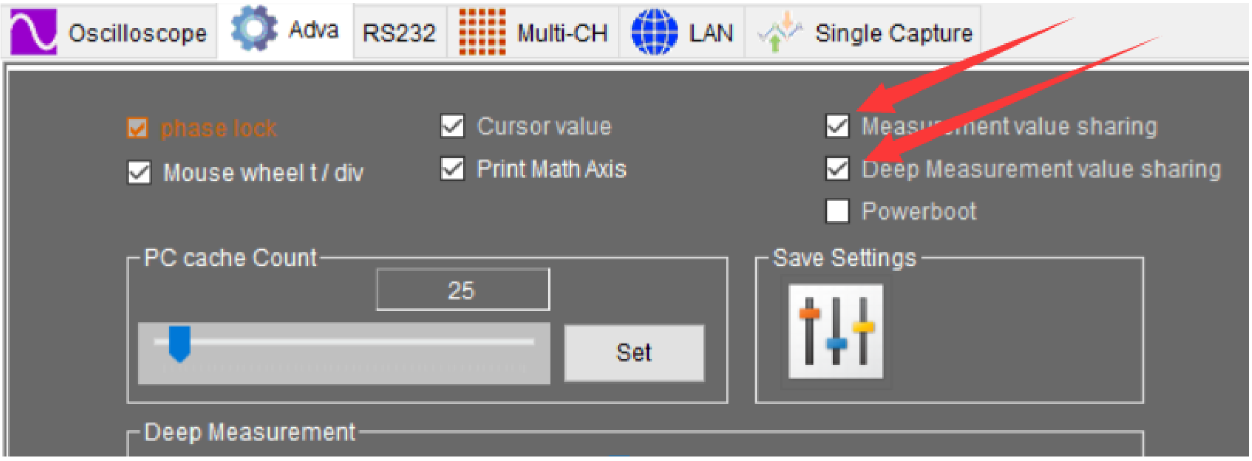
The data sharing function is to send the measured value of the LOTO oscilloscope software in a certain frame format through the serial port of the PC, which is convenient for the customer's third-party equipment, such as PLC, single-chip microcomputer, ARM board, industrial computer, etc., to obtain the waveform of the LOTO oscilloscope Measurements.

Clicking the button pointed by the arrow in the above figure will open the data sharing page, as shown in the figure below:

After opening the Data sharing interface, we check the measured value option, as shown in the figure below:

At this time, the waveform data collected by the oscilloscope in real time locks the various measured values calculated (automatic measurement introduced in 8.2.10).

If we find that the data does not change, it may be because we have not enabled the data in the oscilloscope software by default. Output switch, we are in the "Advanced" tab, make sure to enable the following settings:



These measured values are read from the host computer software of the LOTO oscilloscope in real time. Therefore, it is necessary to ensure that the LOTO oscilloscope software is running normally so that there will be real-time data presentation.

We can set the refresh rate of the data, which can also be understood as how often we go to the oscilloscope software to get the latest measurement data and update it.

We can set the serial port to send these measured values. Generally, we can connect a serial port on the PC through USB to serial port, and then set the serial port to communicate with external third-party devices, as shown in the figure below:

Which measurement values we want to send can be selected in the tree directory on the left. We preset different command codes for each measurement value, so that after receiving the data frame, the third-party device can parse out the command code to distinguish different Measurement category:

The format of the data frame sent is as follows:

Address code (1 byte)	Command code (1 byte	byte count (1 byte)	Data high byte (1 byte)	Data low byte (1 byte)	CRC check high byte	CRC check low byte
It can be set by the interface, the default is 0X02	preset in the software	0X02 (fixed, representing two bytes of data)				

The high byte and low byte of the data form a 16-bit data, which is organized as follows: the lowest bit (bit0) represents the symbol, 0 means that the number is positive, and 1 means that the number is negative number. The penultimate 3rd bit (bit2) and penultimate 2nd bit (bit1) indicate how many digits of the entire data are decimals. for example:

Bit2	Bit1	Detail
0	0	no decimal part
0	1	has 1 decimal place
1	0	has 2 decimal places

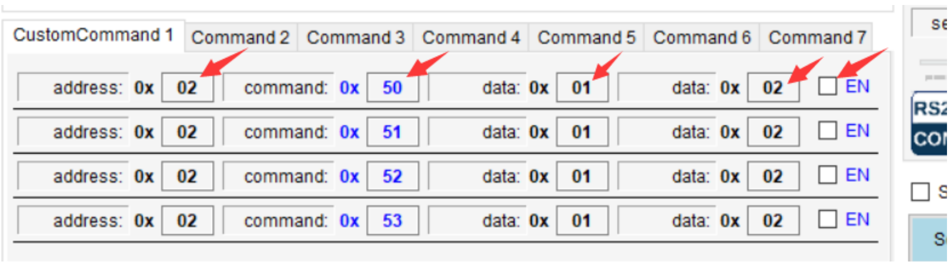
1	1	has 3 decimal places

We consider the remaining 13 bits to be a binary positive integer. After converting it to a decimal integer, we restore the integer to a signed decimal according to the positive and negative bits and the number of decimal places. For examples:

Bit 15	Bit 14	Bit 13	Bit 12	Bit 11	Bit 10	Bit 9	Bit 8	Bit 7	Bit 6	Bit 5	Bit 4	Bit 3	Bit 2	Bit 1	Bit 0
1	0	1	1	0	0	1	0	1	1	1	0	0	**1**	**0**	**1**

The binary data 1011001011100 converted to decimal is: 5724, the lowest bit is 1 to represent a negative number, so it is -5724, the third and second last digits are 10, representing 2 decimal places, so it is -57.24.

If the data sent by the customer requires a special command code or a special data format, you can set a custom frame format for sending, as follows:



8.3.1 Video demonstration:

《LOTO virtual oscilloscope Data sharing function Preview》

https://www.youtube.com/watch?v=Ad31afTcPgc&t=1s

9 Data Acquisition card mode

The LOTO oscilloscope adds the function of the capture card mode to supplement some inflexible application scenarios of the oscilloscope mode.

Since the oscilloscope automatically selects the sampling rate and storage length at a selected time slot, these parameters are fixed, and it is very convenient for the user to view the waveform in some scenarios. Users don't even have to care about or understand concepts and theories like sampling rate and memory depth.

But this also brings another inconvenient application problem. In some applications, the user has already clearly known the situation of the signal to be collected. According to the plan, the user needs to set the sampling rate, and then an exact long-term data. At this time, the original oscilloscope mode is not very convenient. LOTO's acquisition card mode is to supplement the deficiency of the oscilloscope mode and provide a new data acquisition method.

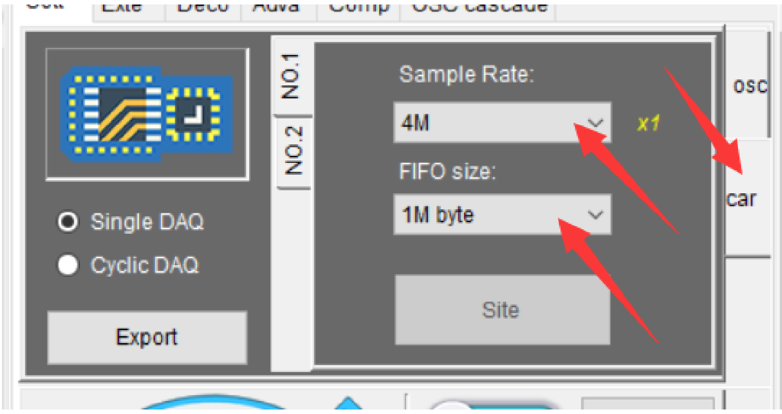
Oscilloscope mode and acquisition card mode have different application scenarios and different operations. They will not be used at the same time. They are either oscilloscope mode or acquisition card mode. If the user selects the "OSC" tab, it is using the oscilloscope mode. If you choose If you open the capture card tab, you are using the capture card mode.

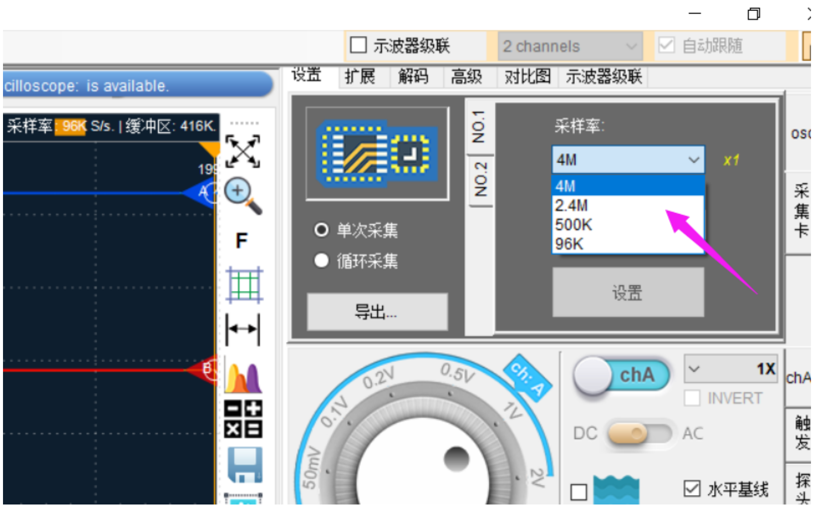
9.1 Basic settings

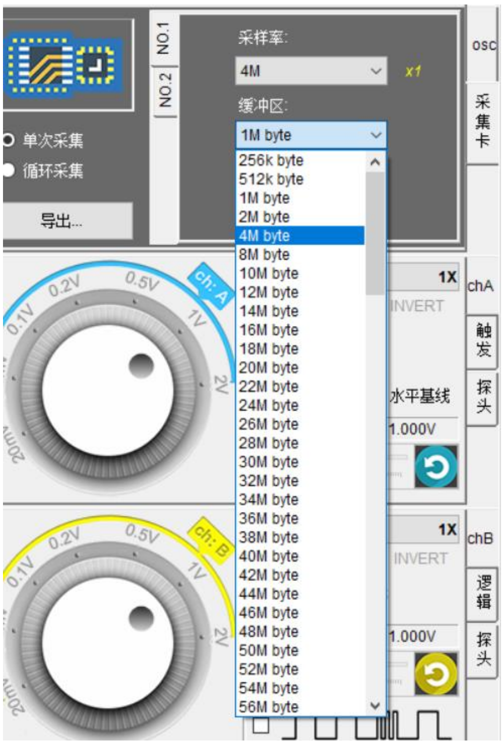
The oscilloscope mode is shown in the figure below, and we can see the time scale knob (roulette), which is in the oscilloscope mode.

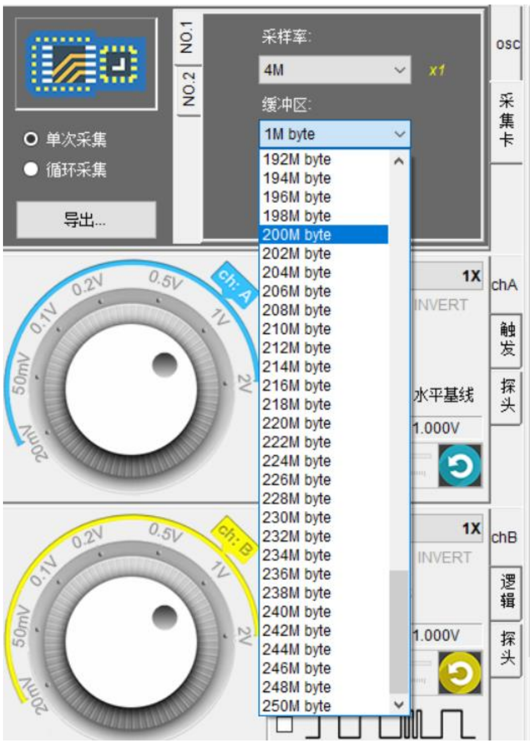
The capture card mode is shown in the figure below, we can't see the time gear knob (roulette), but we can see the capture card icon:

In the setting interface of capture card mode, we can choose the required sampling rate and buffer size. The sampling rate can have several options, and it is not completely possible to enter the settings by yourself. The important thing is that there are many buffer size options, and the maximum buffer size can be 248M bytes.

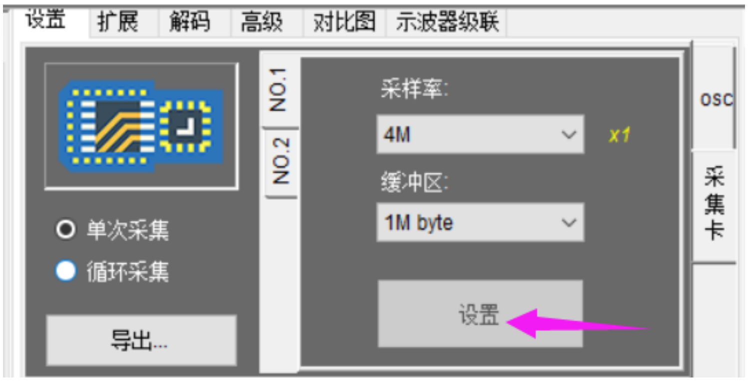








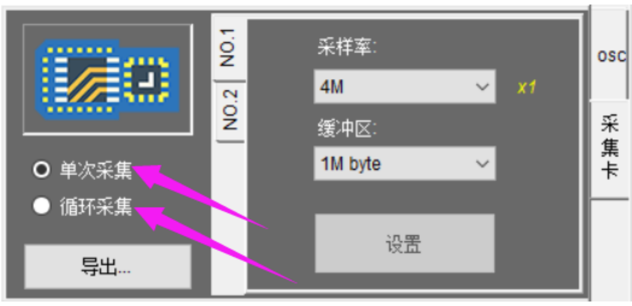
After selecting the sampling rate and buffer size, we need to click the "Settings" button to make the sampling rate and buffer size take effect, as shown in the figure below:



Next, we can use the channel start/stop button to control the acquisition of data waveforms as in the oscilloscope mode, as shown in the figure below:

Other operations are similar to the oscilloscope mode, the following points need to be noted.

In the acquisition card mode, there is no trigger function. After clicking the start button above, the oscilloscope collects the data of the specified buffer size according to the set sampling rate, and the acquisition is completed. Whether the oscilloscope automatically stops after one acquisition or automatically repeats multiple acquisitions depends on our settings, as shown in the figure below:

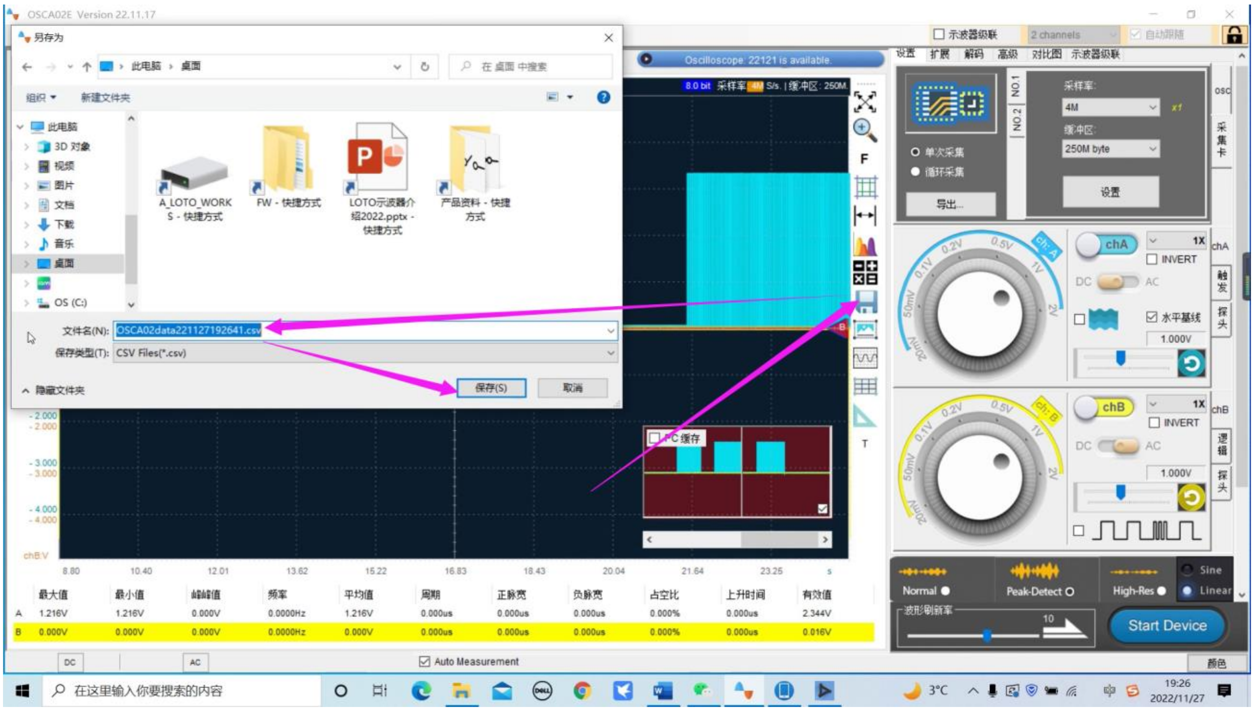


In the capture card mode, we usually set a relatively large buffer, so the time required to complete the entire screen capture is generally relatively long, and the entire time can be calculated, and the capture time = half of the buffer size/sampling rate. Under the setting of long time, the acquisition card mode needs to wait until the data buffer set by the user is completely collected before displaying the waveform. The user needs to wait patiently, which is also different from the oscilloscope mode.

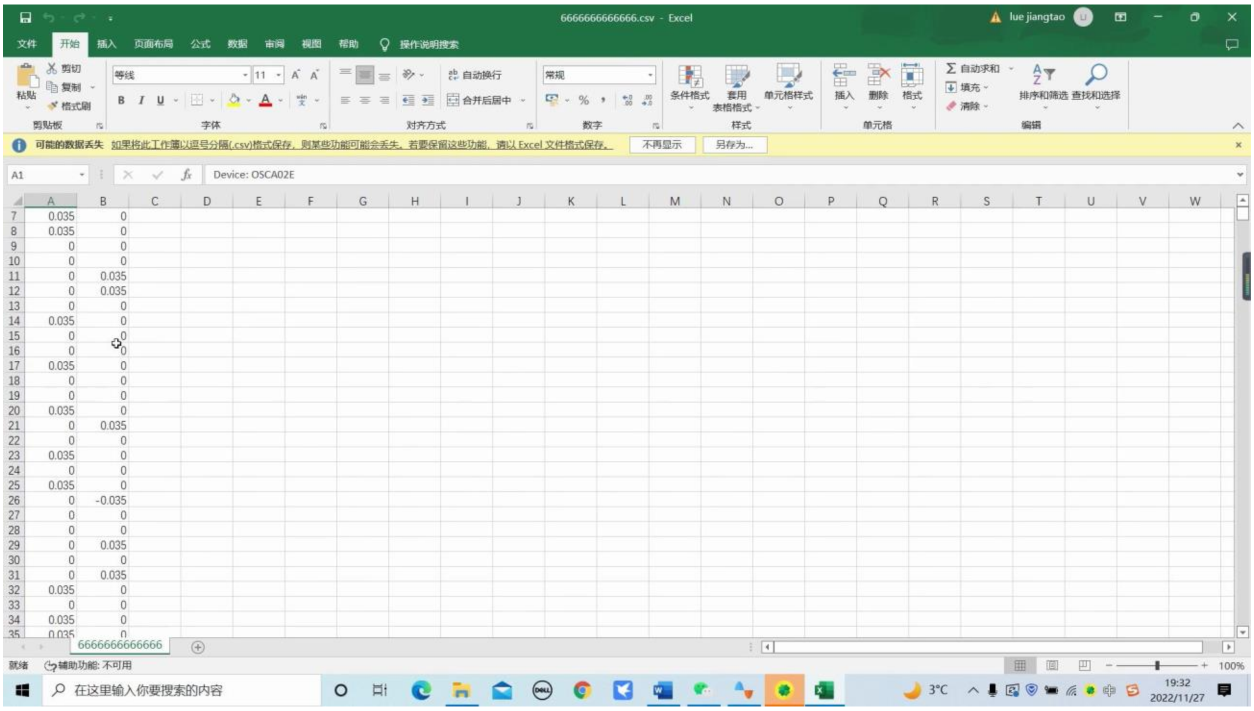
In capture card mode, we will open up an area in the computer memory for buffer use. If the buffer setting is relatively large, the following reminder will pop up. No need to contact us. This is just a reminder for users with low computer performance. It may take up relatively large memory resources. If you can use a small amount of buffer, you don't need a large one, so as not to make the computer freeze.

9.2 Data storage and review in Acquisition card mode

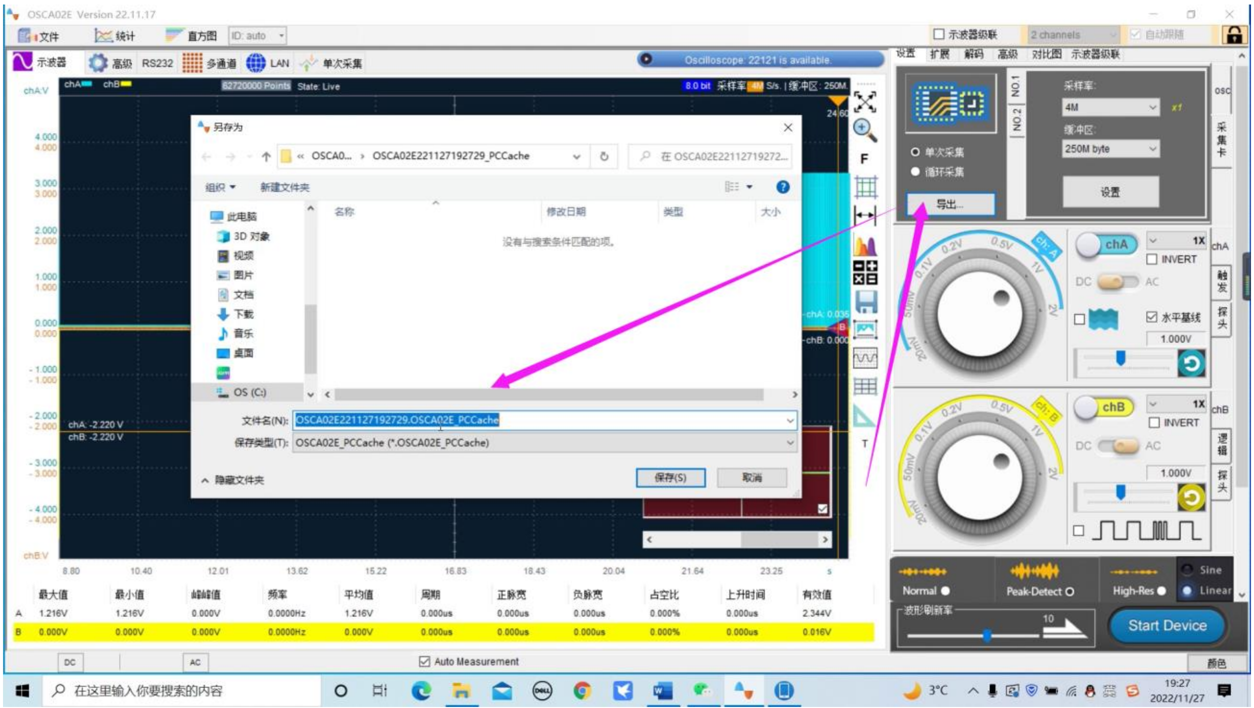
If the collected data is to be saved as a text file, then the acquisition card mode can use the same text save button as the oscilloscope mode to save the collected data as a TXT or CSV format file. As shown below:



The maximum file size of about 480M can be saved, as shown in the figure below after opening:

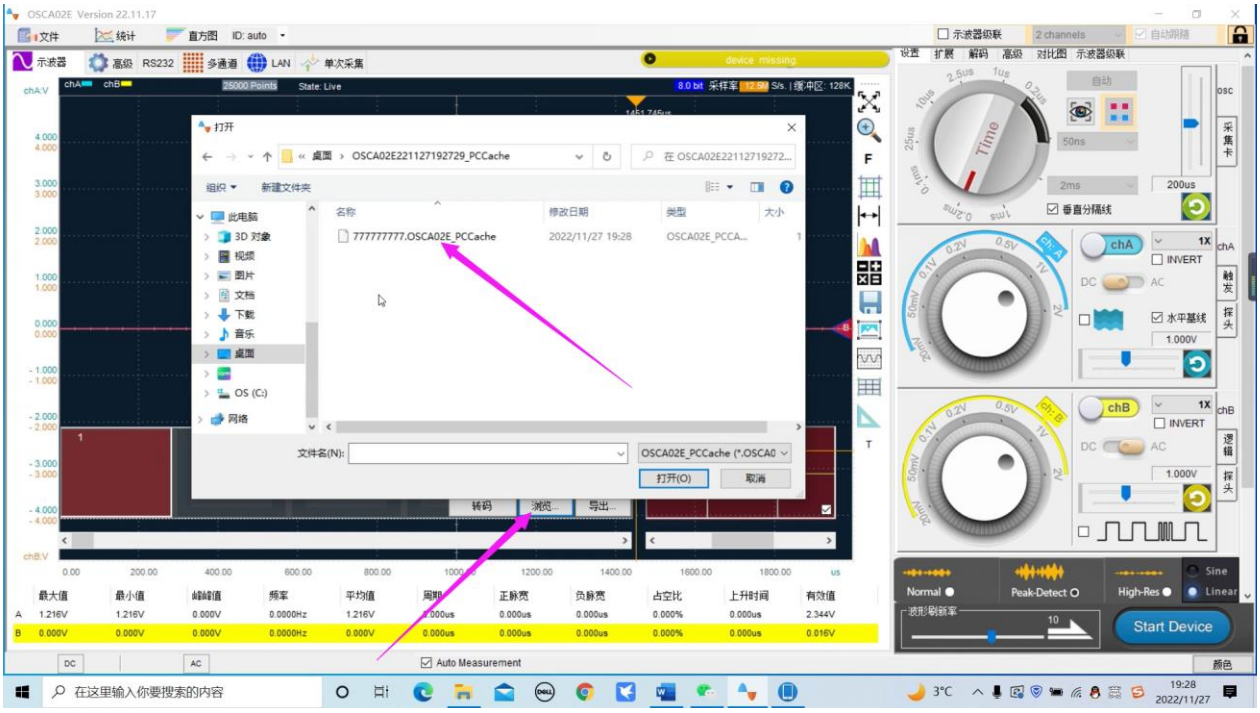


We can also save the collected data as the PC cache file of the oscilloscope. For the use of the PC cache file, please refer to chapter 8.2.5.1. After collecting the required data, we can click the "Export" button in the figure below. Export the data as a PC cache file of the oscilloscope.



We can also save the collected data as the PC cache file of the oscilloscope. For the use of the PC cache file, please refer to chapter 8.2.5.1. After collecting the required data, we can click the "Export" button in the figure below. Export the data as a PC cache file of the oscilloscope.

We can select the "Browse" button in the PC cache setting area in the oscilloscope mode, select the PC cache file stored above, and then import it into the PC cache. As shown below:



9.2.1 Video demonstration:

《Basic introduction of LOTO oscilloscope acquisition card mode》

https://www.youtube.com/watch?v=Qn3VRcIoo_g

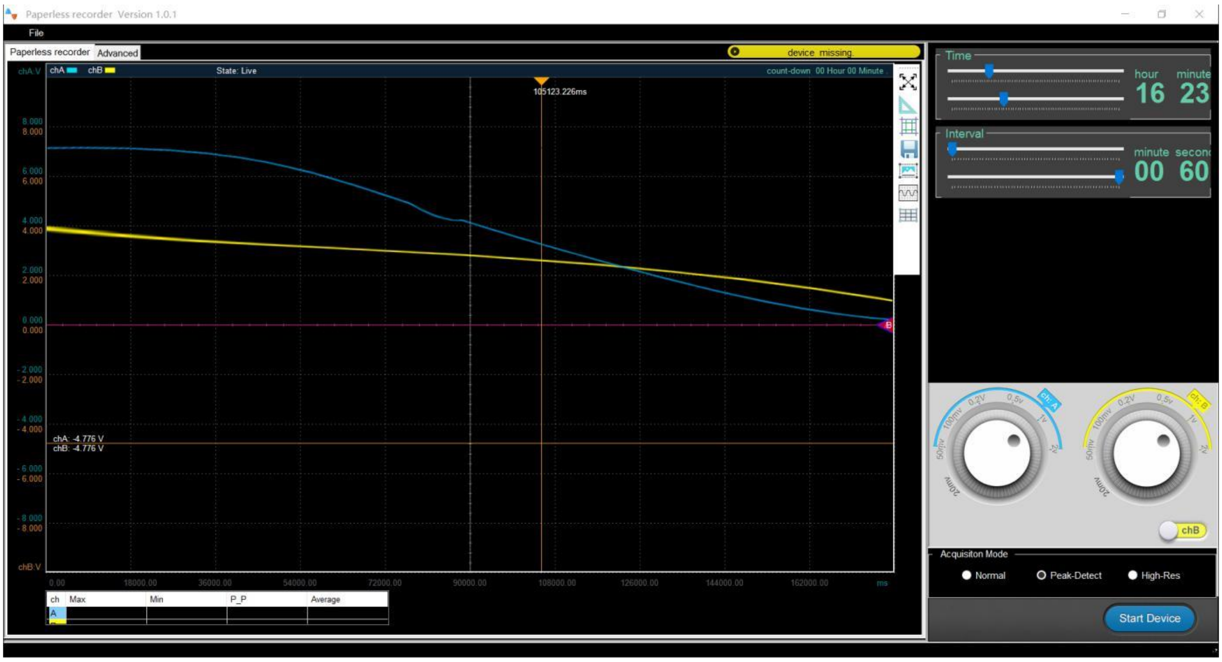
10. Paperless Recorder

This chapter introduces the basic concepts and usage of the paperless recorder, a software functional extension of the virtual oscilloscope hardware.

10.1 Basic Concepts of Paperless Recorders

A paperless recorder is a data logger or data acquisition device used to record measurement data over time. A paperless recorder based on virtual oscilloscope hardware has the ability to record the collected or computed data in a storage system inside the software on a time basis, without consuming any paper, pen or ink. Common collected data are voltage, current, etc.

10.2 Introduction to Paperless Recorder Software



Common software interface

The Recorder software has an interface consistent with the oscilloscope software, so to be familiar to the user.

Equipment monitoring

After the software interface is opened, the OSCxxx device status will be monitored in real time. When the hardware device is connected to the USB port of the PC, the software interface will display a blue background and a message.

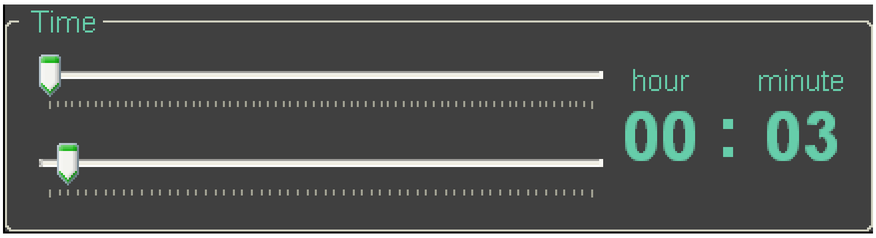
If the hardware get disconnected or the software have no access to the device, then the display background will become yellow, showing a text alert.

The sampling rate of the paperless recorder is not high. It is suitable for scenes with long and slow electrical signal change cycles, such as the detection of a slow discharge curve during the battery discharge process for 24 hours a day. For scenes where the signal changes quickly, it is recommended to use oscilloscope software for inspection and observation.

> **Note**
> The paperless recorder software has exclusive access to the OSCxxx device. Therefore, the device can be used just with one software at a time.

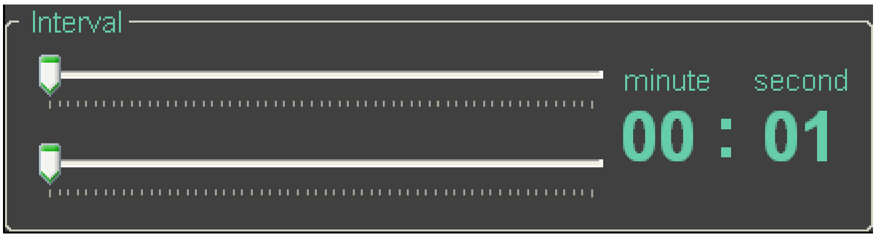
10.2.1 Set the Total Record Duration

This panel allow to set the recording duration; it can be set in the range 1 minute ~ 3 days.



10.2.2 Set the Sampling Interval

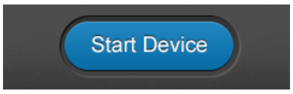
This panel allow to set the sampling interval from 1 second to 1 hour.



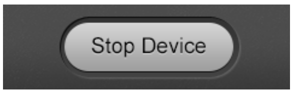
> **Note**
> The sampling interval cannot be greater than the total recording duration, otherwise it is meaningless.

10.2.3 Start and Stop

After the driver has been successfully installed, if the oscilloscope is connected when the software is opened, the software automatically turns on the device. If the device is connected before opening the software, it is required to start the device manually by clicking on the **Start Device** button in the lower right corner of the software.



The acquisition can be stopped by clicking the Stop button. The Start and Stop button are actually the same button that change according the status of the oscilloscope. When the USB cable is suddenly unplugged during the operations, the software will automatically stop.



When the device is not used it is recommended to close the software and then to unplug the device's USB cable.

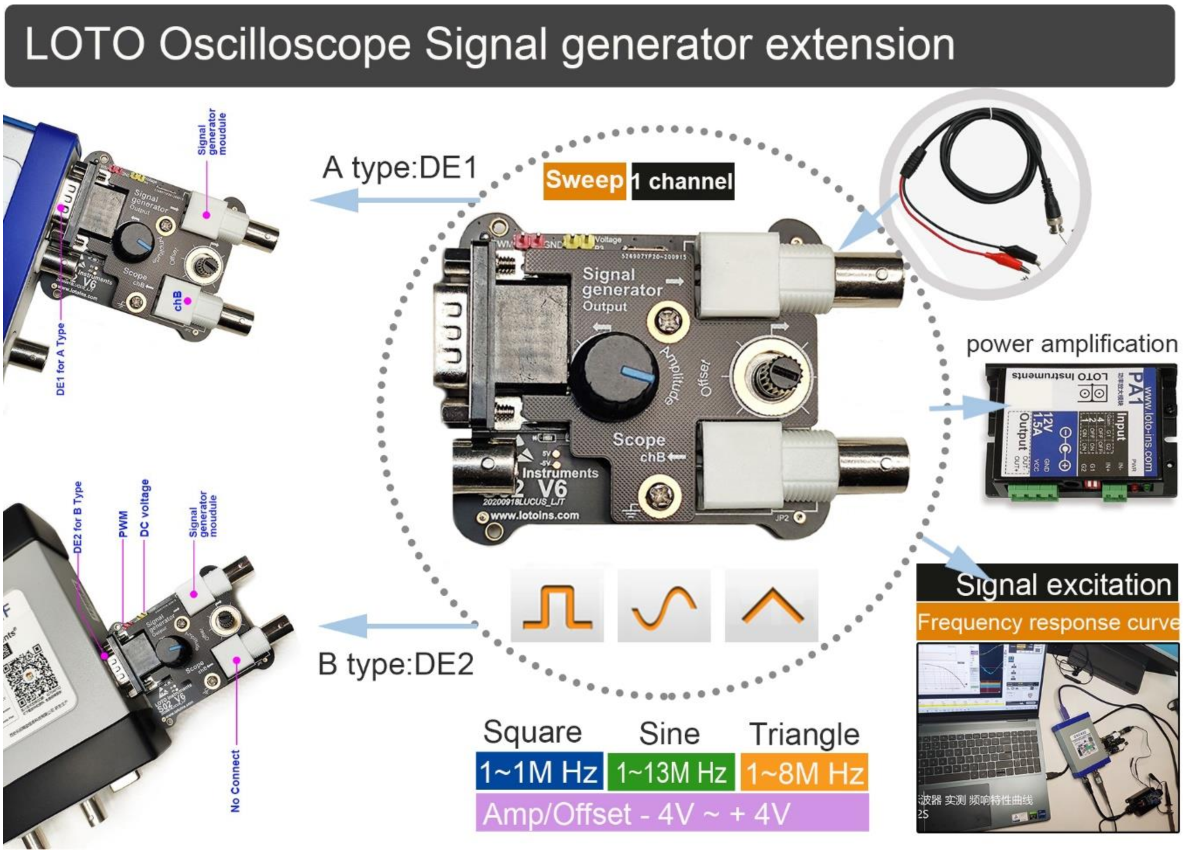
11. Signal generator module(S02)

Signal generator S02 is a single-channel signal source module, which belongs to the expandable module of LOTO oscilloscope. It is a signal source module made around the DDS chip to facilitate customers to quickly generate periodic regular waveforms.

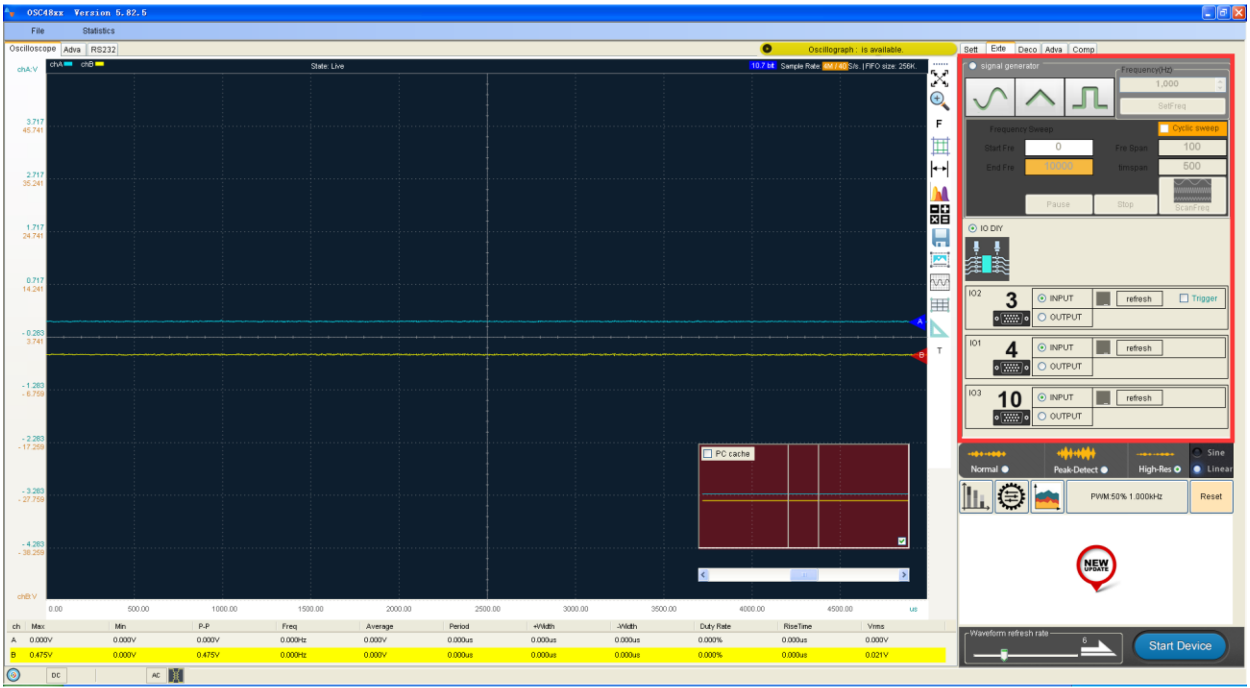
Some types of oscilloscope products support the extended functions of the signal generator module, and you can check the hardware data sheet of the product. If the purchased product does not contain or support the signal generator module, this part of the software function will not produce the expected effect.

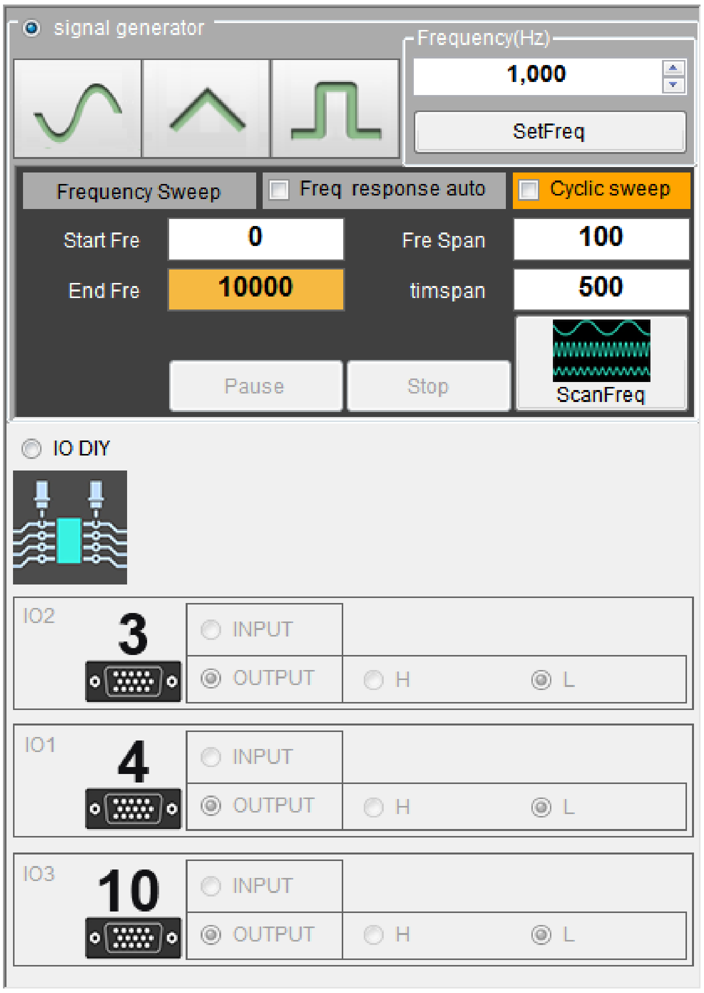
At present, there are many types of oscilloscope products, and the plugging positions of the extended signal generator modules of different types of oscilloscopes are different.

The GPIO function of the OSC482 series is extended at the DE1 port, and the GPIO function of other series of LOTO oscilloscopes is extended at the DE2 port. The specific connection location is as shown in the figure below:



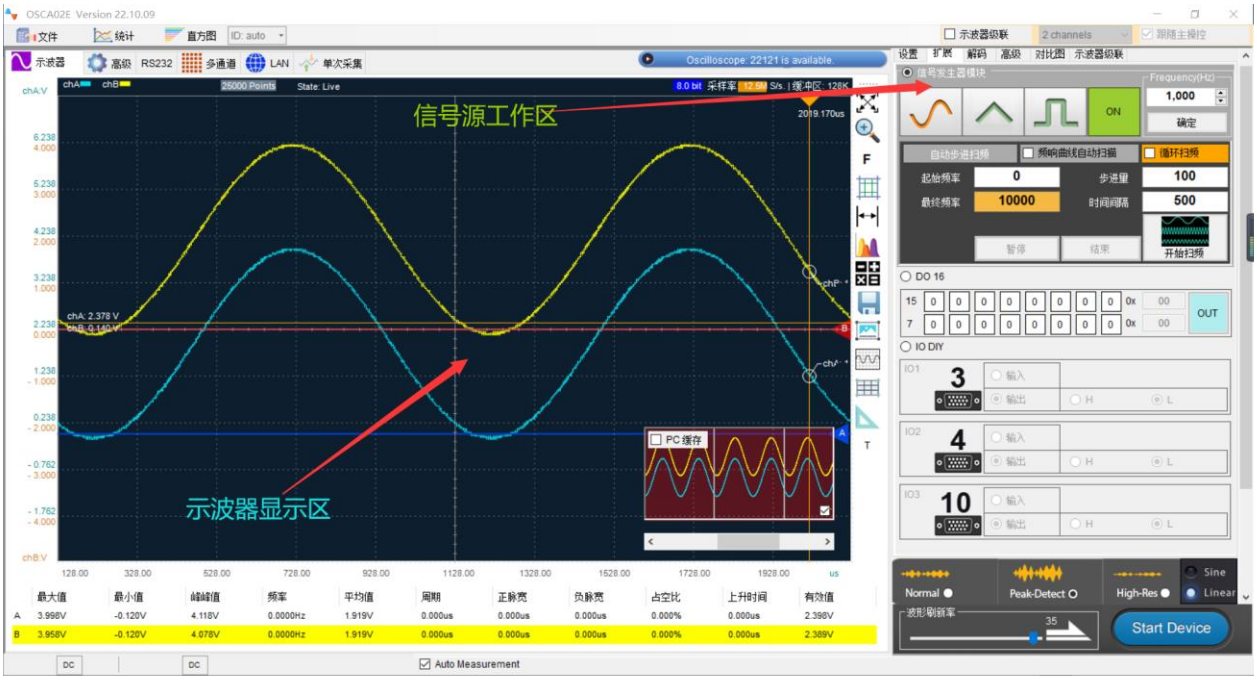
Some devices have the ability to be integrated with signal generator moudle. So if you receive a Signal generator module, you may find the corresponding feature in the host devices' software:



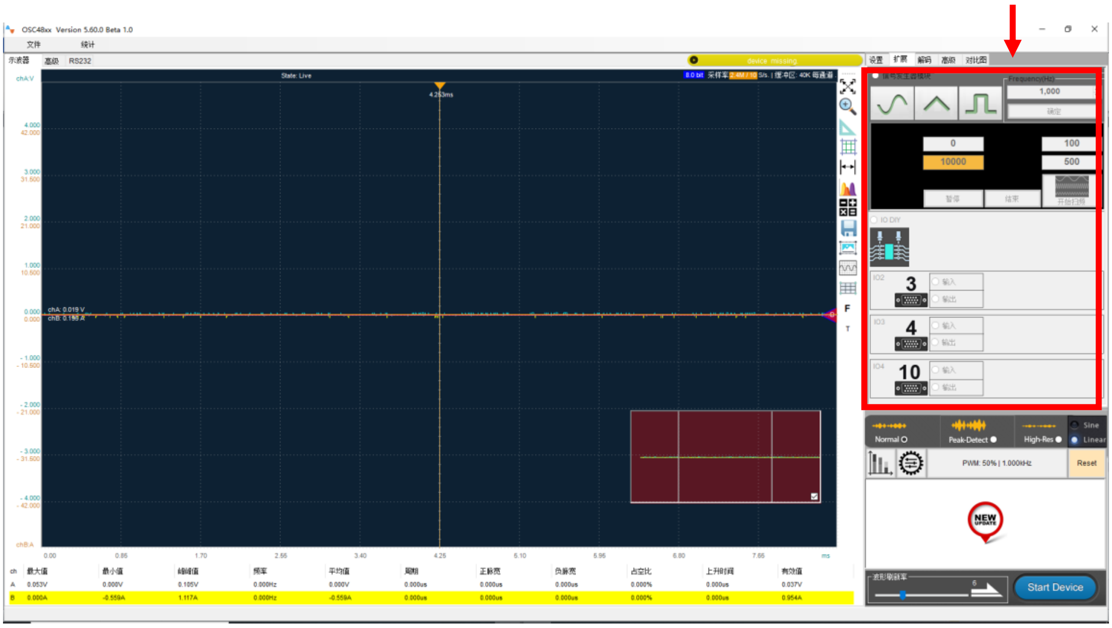


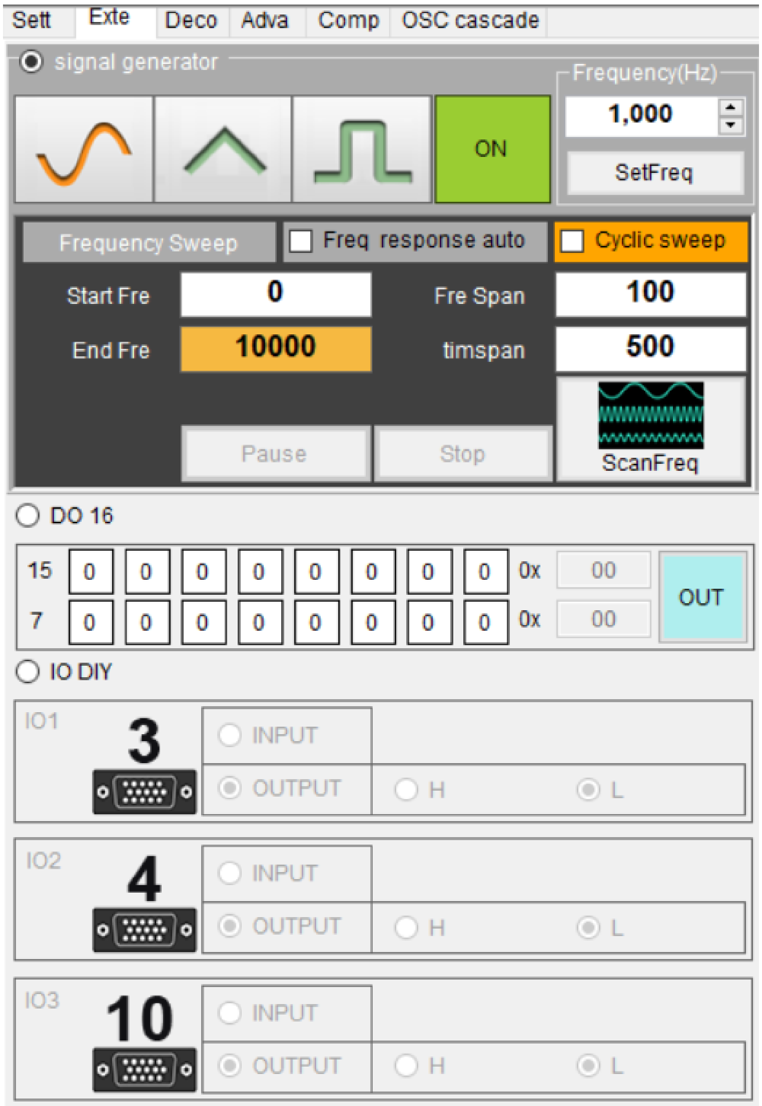
11.1 Signal generator and oscilloscope work simultaneously

The two-channel standard product of LOTO oscilloscope can work with the extended S02 signal source at the same time, so that one analog signal can be output and two input analog signals can be monitored at the same time. On the software, it can also be displayed and set at the same time, as shown in the figure below:



As shown in the figure above, the software setting area corresponding to the signal source module is in the tab page "Extended". Click the "Extend" button of the software, and the extended function interface will cover the "Settings" function interface of the oscilloscope. If the signal generator is not selected in the "Extended" interface, the signal generator module will not start and will not work.

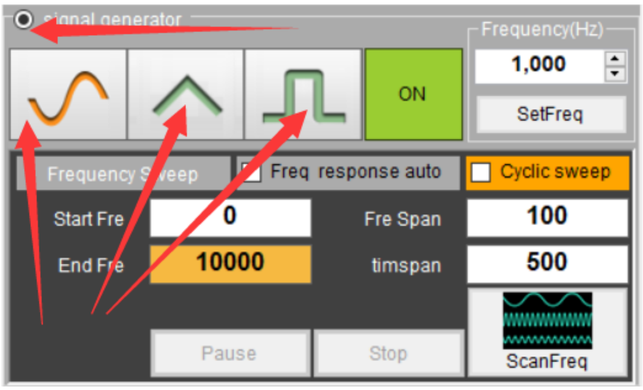




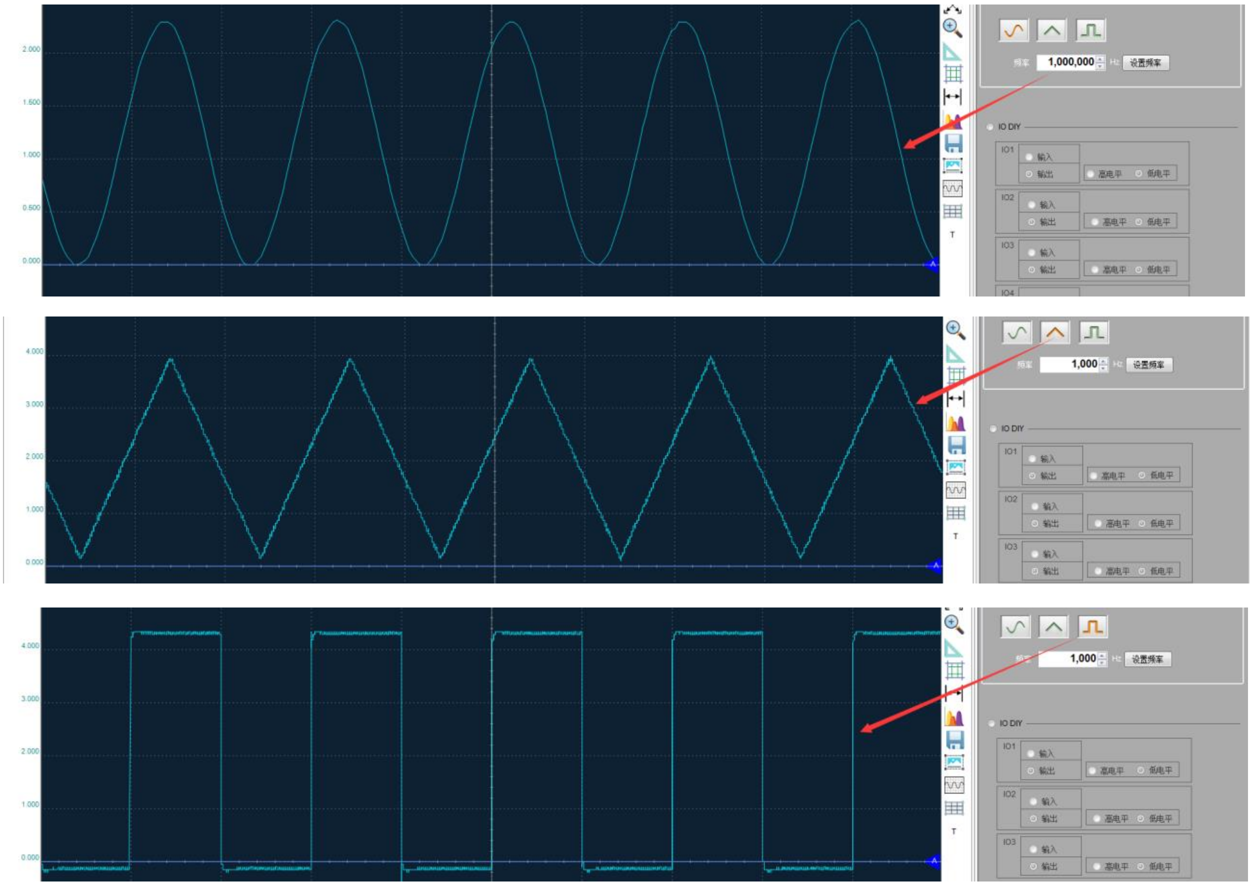
The waveform type (sine, triangle, square wave) and waveform frequency of the signal generator module S02 are set by software, and the amplitude and offset of the waveform are controlled by the hardware knob on the module. Since the LOTO oscilloscope can be used simultaneously with the signal source module, the amplitude and offset of the signal can be observed through self-sending and self-receiving.

11.2 Signal Generation Waveform Settings

Click the "Signal Generator Module" button (marked by the red arrow) to select the signal generator, and the signal generator module will start. Then you can click to select the desired waveform button, the icon on the button will change from green to orange, and the signal generator will generate a waveform to provide a stable signal source.



After getting the signal generator module, in order to quickly check whether the signal generator module is normal. You can use the probe of the oscilloscope to measure the output interface of the signal generator module. If the waveform state, frequency, and amplitude of the signal in the display area of the oscilloscope are consistent with the settings of the signal generator module, it means that the module is functioning normally. (The figure below is the waveform measured by the oscilloscope and set to 1KHZ)



11.3 Signal frequency setting

The signal generator module can set the frequency of the waveform according to the needs, the frequency setting range:

Sine	**1Hz ~ 13M Hz**
Triangle	1Hz ~ 8M Hz
Square	1Hz ~ 1M Hz

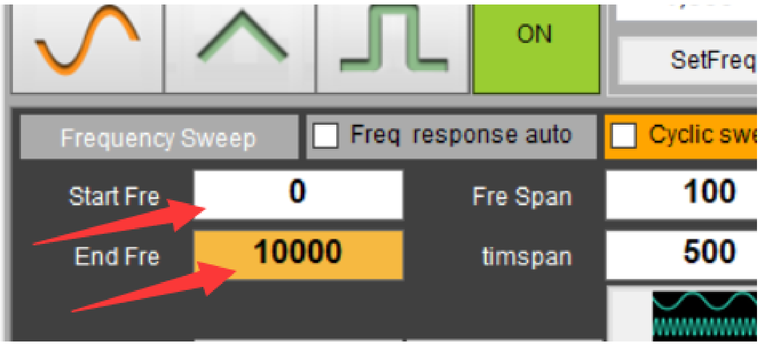
The frequency can be directly input in the input box, and the unit is HZ. You can also adjust the triangle button attached to the input box. Due to the different ranges of the frequency value in the display box, the step value is also different each time the button is clicked. For details, refer to the following table:

Frequency	Click the triangle button (up or down) to change the frequency step by step
1HZ-99HZ	1HZ
100HZ-990HZ	10HZ
1000HZ-9900HZ	100HZ
≥10000HZ	1000HZ

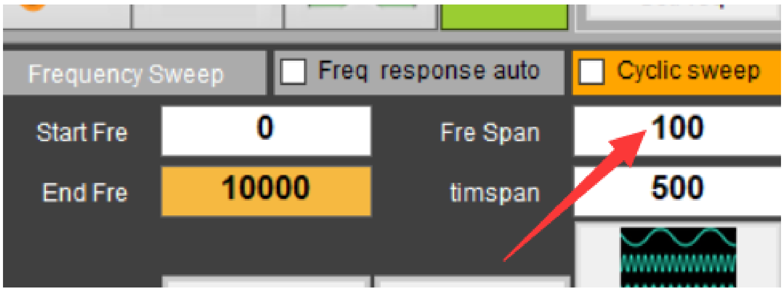
The signal generator module adds a software-controlled shutdown function, which may not be supported by older versions of the oscilloscope hardware. Click the OFF button to cut off the output of the signal waveform, and click the ON button to enable the output of the signal:

11.4 Signal Generator Sweep Function Setting

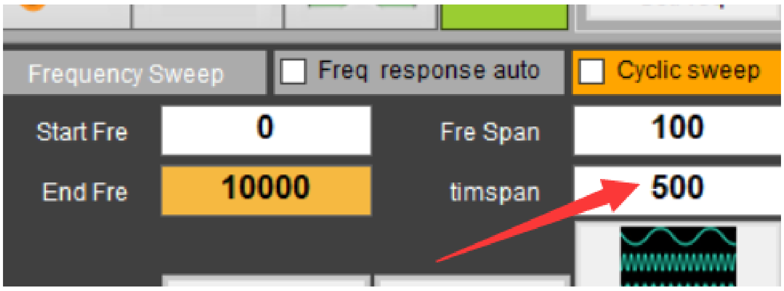
The signal source module S02 supports frequency sweep output in the software. You can set which frequency point to start the frequency sweep and which frequency point to end the frequency sweep. The unit is HZ. Note that the setting range should be within the capabilities of the S02 module.



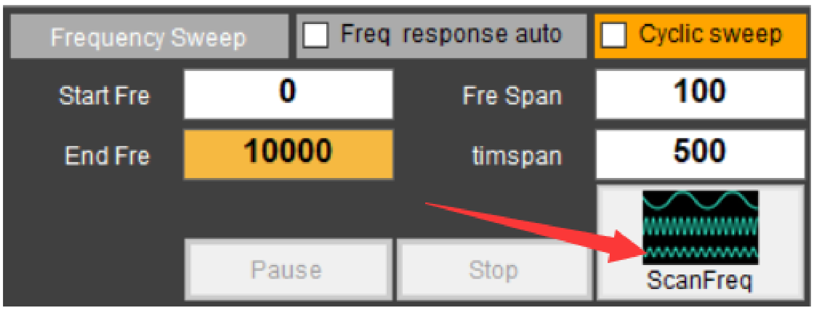
It is also possible to set the step frequency of the sweep, that is to say, how much the frequency should be increased based on the current output signal frequency each time the sweep changes the frequency, and the unit is also HZ.



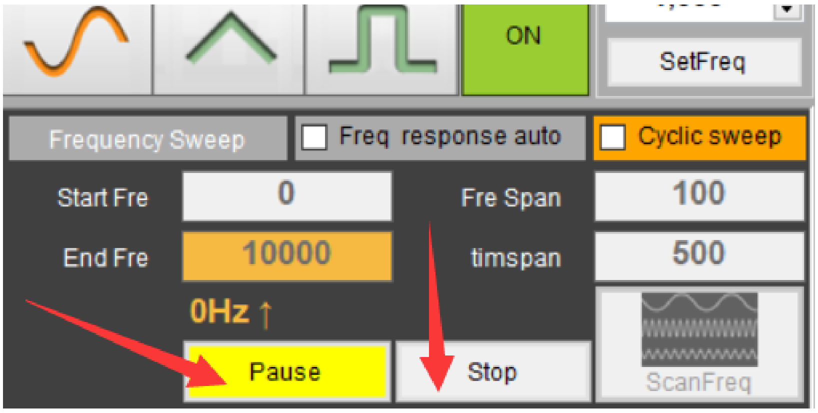
The time interval of the frequency sweep refers to how long the interval is to increase the frequency of the current signal by one step during the frequency sweep process. The unit of this time interval is milliseconds, as shown in the figure below:



After the setting is complete, you can click the frequency sweep button to sweep the frequency:



During the sweeping process, you can pause/resume or stop at any time:



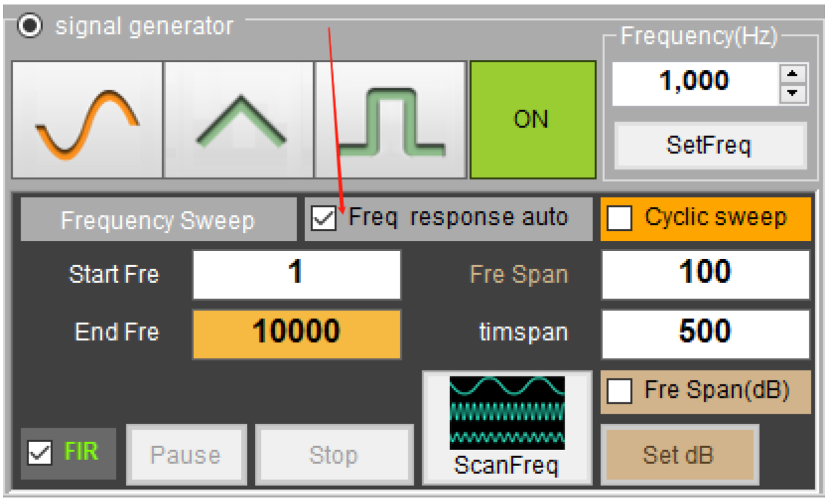
The default frequency sweep is one-way after it starts, that is to say, after sweeping from the initial frequency to the final frequency, the frequency sweep stops, and the frequency is fixed at the final frequency. If you need to perform the entire frequency sweep process cyclically, you can check the "Circular frequency sweep option".

As shown in the figure below, the frequency response curve automatic scanning function is used together with the "frequency response curve drawing function", please read the description in chapter 8.2.27 Frequency Response Curve Function.

The starting value of the fully automatic frequency range supported by different models of this function is different, the OSC482 series is 25Hz, and other models are 2Hz.

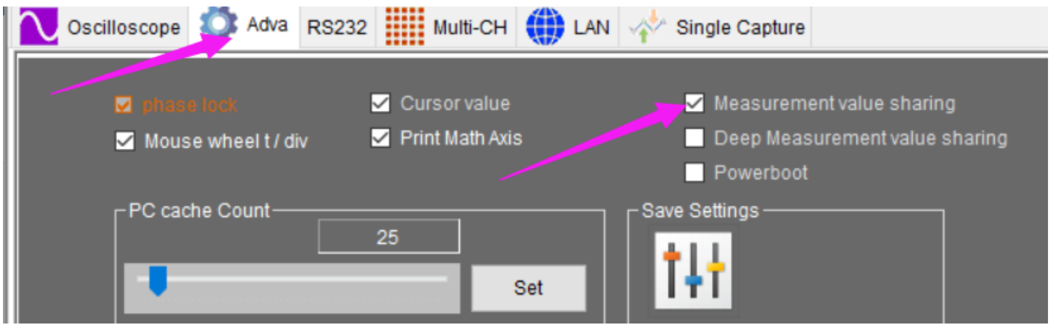
11.4.1 Frequency response auto（20230708）

As shown in the figure below, the frequency response curve automatic scanning function is used together with the "frequency response curve drawing function", please read the description in chapter 8.2.26 Frequency Response Curve Function.



The function of this automatic scan is that during the whole process of the signal source sweeping output, the LOTO oscilloscope automatically adjusts the time scale to adapt to the changing signal frequency. If this function is not selected, the LOTO oscilloscope will be fixed at the time slot set last time. As the frequency of the frequency sweep signal changes, this time slot may become unsuitable for observing the signal under test. Customers need to manually adjust the time slot. Therefore, this function is generally turned on in the test scene of the frequency response curve. The starting value of the fully automatic frequency range supported by different models of this function is different, the OSC482 series is 25Hz, and other models are 2Hz.

When using the automatic frequency sweep function of the frequency response curve, the [measurement value sharing] function needs to be enabled. As shown below:

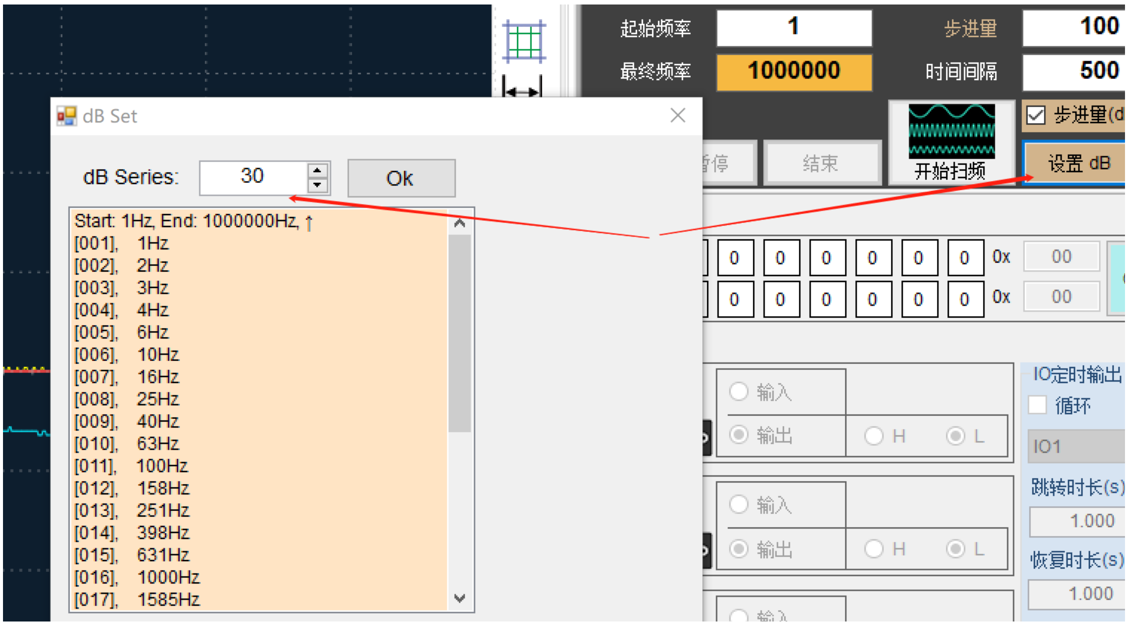


For the introduction of using the signal generator to sweep the frequency response curve, please refer to "Appendix 4: LOTO Oscilloscope Detection Frequency Response Characteristic Curve" and chapter 8.2.6 Frequency Response Curve Drawing Function of this document.

11.4.2 Logarithmic (DB) Sweep Settings（20230708）

In some scenarios where the frequency sweep range is relatively wide, if the normal linear step sweep is used, the entire sweep process will be slower. Especially in the application of measuring the frequency response curve of the circuit unit, we have added the option of DB logarithmic frequency sweep.

As shown in the figure above, after we checked the "step amount DB", we can sweep the frequency in DB logarithmic mode. We can also further set the step number of DB: click "Set DB" to get the logarithmic step number dialog box:



As shown in the figure above, the default is to set 30 logarithmic intervals for the frequency interval of the entire frequency sweep, and customers can set their own intervals according to their needs.

11.4.3 Swept FIR filter settings（20230708）

In the process of drawing the actual frequency response curve, it is recommended to turn on the FIR filter during frequency sweep, as shown in the figure below：

The effect of turning on the FIR filter is to band-pass filter the waveform data collected by the LOTO oscilloscope during the frequency sweep process. The setting of the FIR band-pass filter is: half of the current sweep frequency, and the frequency signal between 1.5 times of the current sweep frequency is reserved, and the component signals of frequencies lower than and higher than this band-pass range are all eliminated and filtered.

This function is very useful in the application of frequency response curve drawing. Ideally, the frequency-swept sine wave signal is added to the circuit under test, and the corresponding sine wave signal output by the circuit under test is standard, but in practical applications, it is possible that the output sine wave signal is distorted or severely disturbed. At this time, the frequency response curve will jitter. Turning on this FIR filter can effectively remove the spectral components of the output signal that are not sweep frequencies, and the waveform will be more standard. The resulting frequency response curve will be more precise.

11.5 Video demonstration：

<LOTO Oscilloscope || Signal generator Module demonstration:>
https://www.youtube.com/watch?v=NmVF-4yOPP4&t=9s

<LOTO oscilloscope frequency response curve sweep and mapping function introduction>

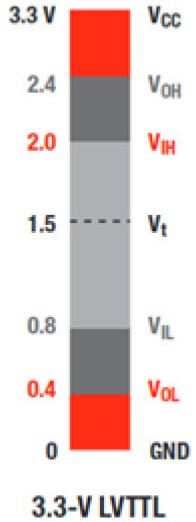
https://www.youtube.com/watch?v=UJpgIUxasKw

<LOTO oscilloscope signal source ||frequency sweep optimization||frequency response curve>

https://www.youtube.com/watch?v=frLPk0uoD_c&t=11s

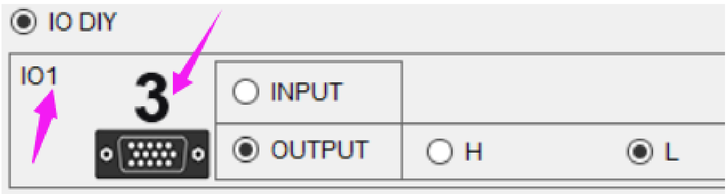
12. GPIO control function

Some devices have this GPIO control function, allow us to implement several IO pins TTL voltage output control or input state detect.These GPIO follow 3.3V TTL level logic.



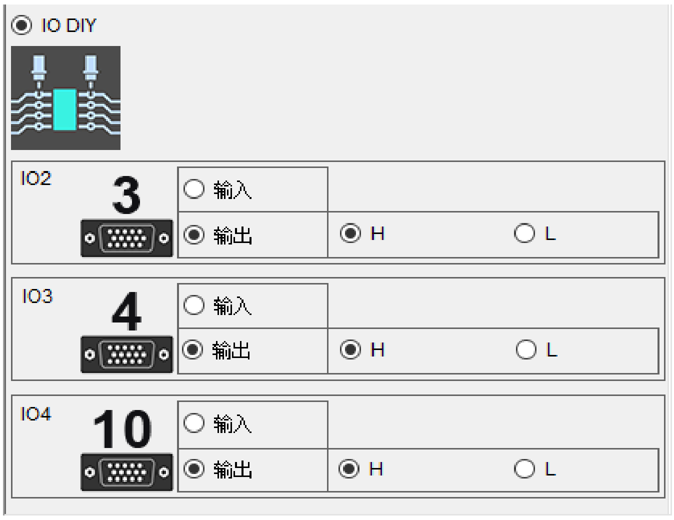
12.1 GPIO Pins

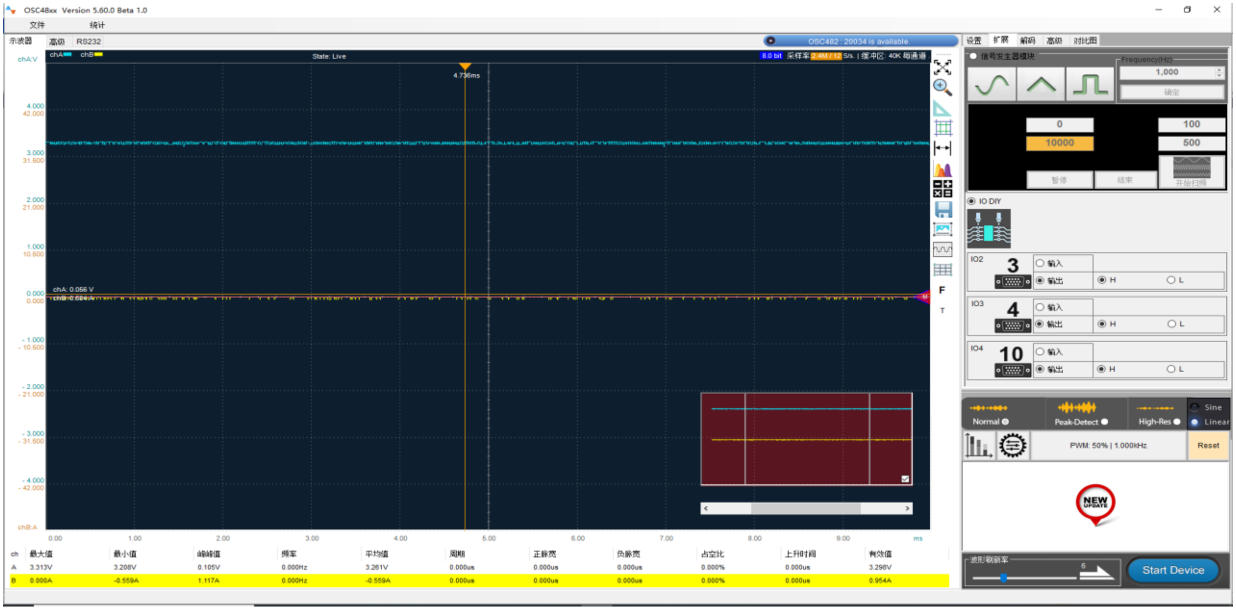
The 3 GPIO pins are named io1, io2, and io3 respectively. The pins of their corresponding physical interfaces are identified in the software interface, for example, io1 is the corresponding pin 3 of the expansion port DE1 or DE2.



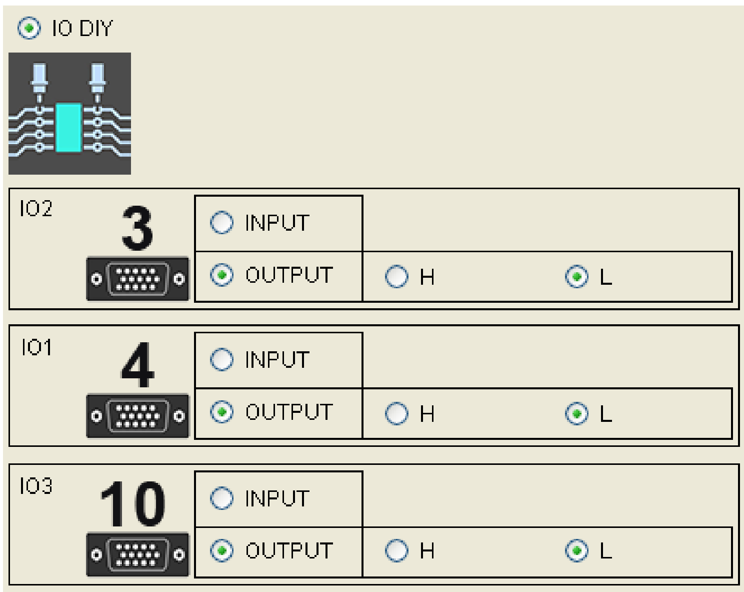
12.2 DO output

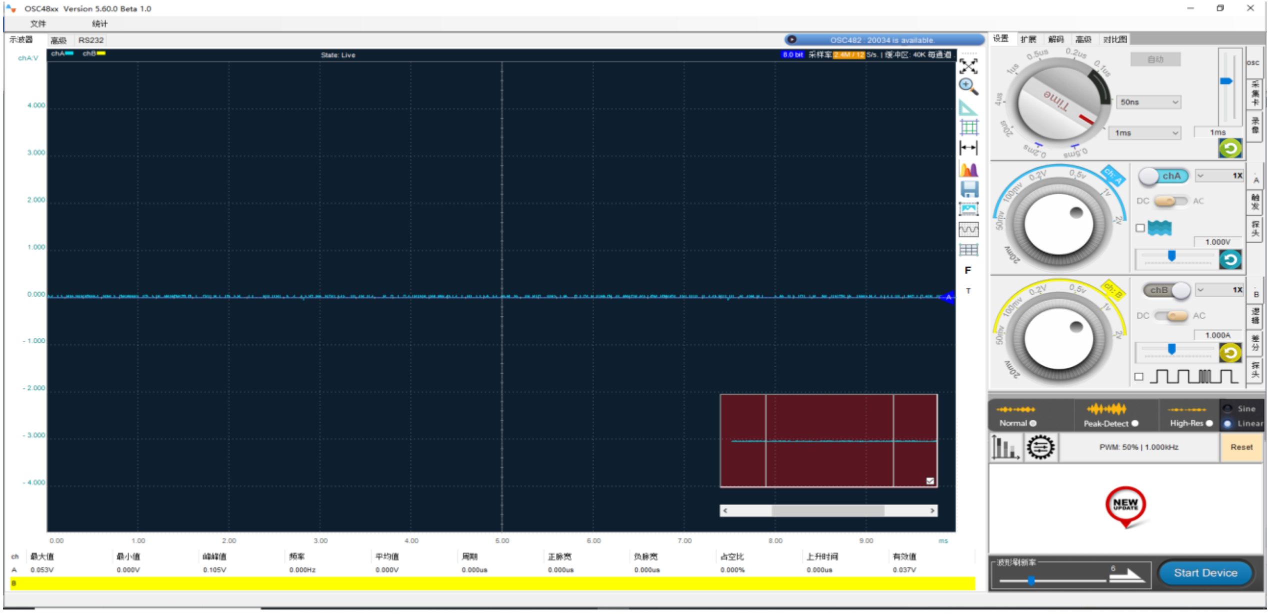
Set the output of the IO to high level respectively, and observe the signal status of the IO port with an oscilloscope:



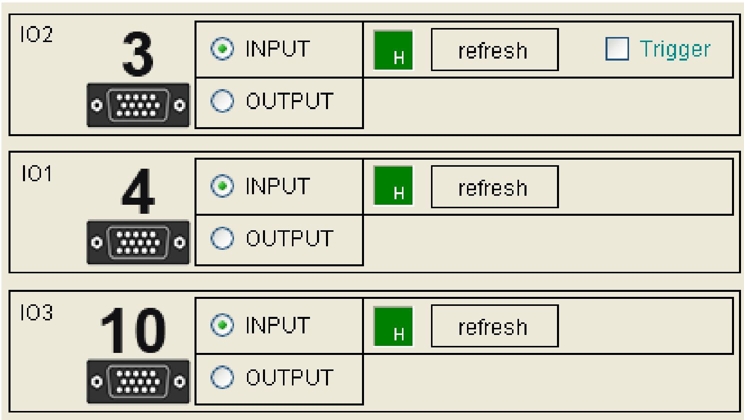


Low level GPIOs output:

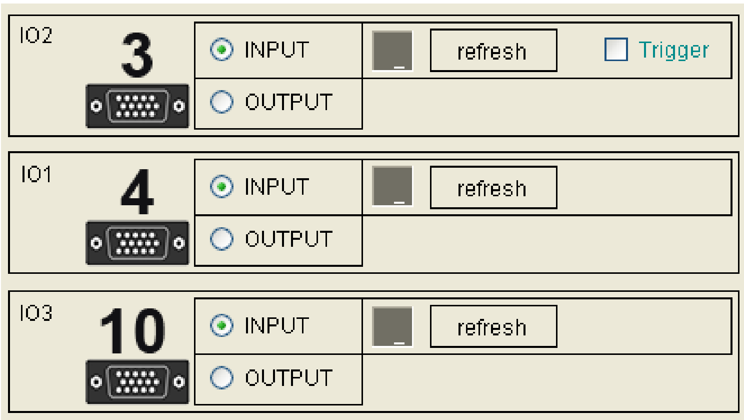




GPIOs detect High level inputs:



GPIOs detect low level inputs:



12.3 DI input

Click the "Input" button to disable the "Output" function, and click the "Refresh" button. When there is no signal connected, the display box defaults to the input high level, and when the signal is connected, click Refresh again, and the current input status is displayed in the display box.

Note
When "IO DIY" is selected, the signal generator module is prohibited to use. Similarly, select the signal generating module, IO DIY is also prohibited.

13 Custom probes

13.1 The role of the custom probe function

Custom probe setting is a very important function of LOTO oscilloscope. Because an oscilloscope is a measuring instrument for voltage data waveforms, it can only measure voltage values.

However, in practical applications, it is necessary to measure various physical quantities, which can be voltage (V), current (A), pressure (Pa), acceleration (g), liquid level (cm), temperature, light intensity, etc.

Usually we use corresponding sensors to convert various measured physical quantities into voltages for measurement. Therefore, each conversion requires a corresponding relationship between the measured physical quantity and the converted voltage value.

Generally, this relationship will be made into a linear relationship, which is convenient for conversion and calculation. Therefore, it is very useful to have a custom probe function to set the linear correspondence between voltage and physical quantity.

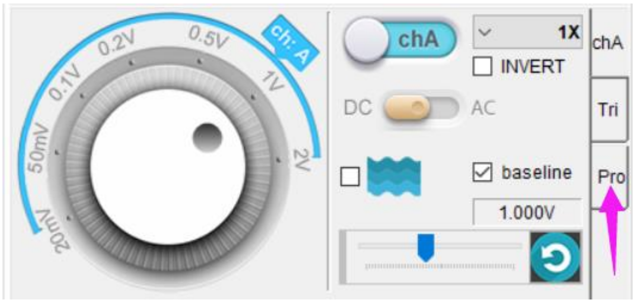
As long as we know the conversion correspondence of various sensors, we can directly display the value of the real physical quantity on the oscilloscope instead of the voltage value.

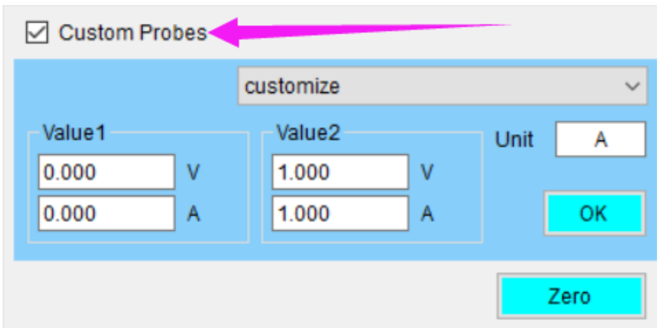
13.2 Enable custom probe function

Most of the standard LOTO oscilloscopes are two-channel, and each channel has a custom probe function.

When customers use other probes or sensors other than the standard voltage probe to measure physical quantity waveform data, they need to use this probe or enable the custom probe function on the corresponding oscilloscope channel of the sensor.

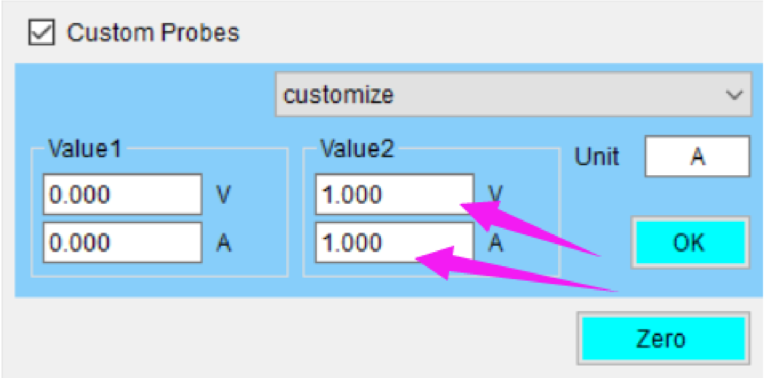
This custom probe function is in the setting area of each channel. For example, the custom probe settings of channel A of the oscilloscope are as follows:





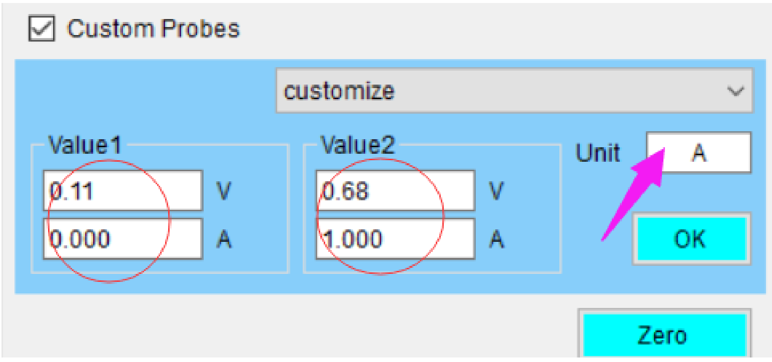
13.3 Settings for Custom Probe Functions

As mentioned above, after we enable the custom probe function of the corresponding channel, there will be two sets of values that we need to fill in.



Two values at value 1 and two values at value 2. They are actually two sets of points with a linear relationship between the physical quantity and the switching voltage. Two sets of points define a straight line.

To give a simple example, if we are using a current probe now, when the current is 0A, the voltage measured by our current probe is 0.11V, and when the current is 1A, the voltage we measure is 0.68V, such Two sets of data are enough to determine the correspondence between the voltage output by the current probe and the current it actually measures, that is the following settings:



Note that the unit is written in the unit of the actual measured physical quantity. After setting in this way, click OK, the data and waveform measured by channel A of the entire oscilloscope will be the actual current data waveform of the current probe, not the data and waveform of the voltage converted by the current probe.

Of course, the same principle and setting method are used for probes and sensors of other physical quantities. As long as we get a probe or a sensor and know what physical quantity and what ratio it converts into voltage, or know how many physical quantities are converted into two sets of voltages, we can directly test the data and waveform of the physical quantity on the LOTO oscilloscope up.

13.4 How to save your own probes (XML)

As mentioned in the previous section, we can already set parameters for any sensor or probe. But the above approach can only be effective at that time.

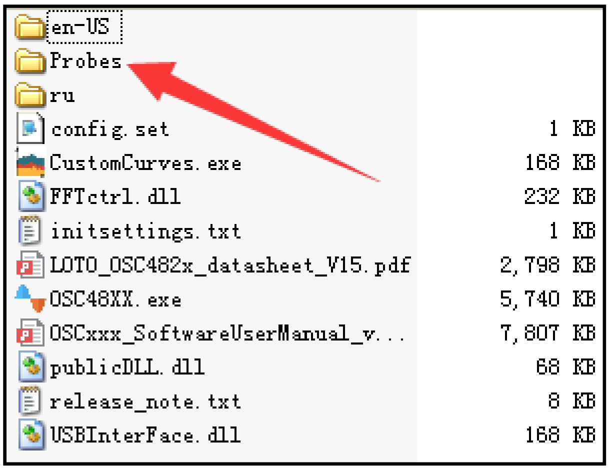
After the software is closed, the parameter setting of this probe/sensor will disappear, and if it is used again, it needs to be set again. It will be a little troublesome. So we need a function to save the parameters of a fixed probe/sensor.

The PC-side software of the oscilloscope supports users to save customization. You can create an XML file in the specified node format under the Probes folder in the application root directory of the virtual oscilloscope software to complete the addition of custom probes.

After adding, as shown in the figure below, directly select the probe you saved and added in the drop-down list, without having to manually set the parameters again.

Let's see how to make a custom probe for your own oscilloscope.

The files of custom probes are all placed in the <Probes> folder under the directory of the oscilloscope host computer software on the computer, so we find this folder and enter:



名称	修改日期	类型	大小
LOTO-C10B.xml	2022/6/8 16:56	XML 文档	1 KB
LOTO-C20B.xml	2022/7/1 18:24	XML 文档	1 KB
LOTO-C50B.xml	2022/6/8 16:57	XML 文档	1 KB
LOTO-C100B.xml	2022/7/1 18:24	XML 文档	1 KB
LOTO-C200B.xml	2022/6/13 16:49	XML 文档	1 KB
LOTO-i01-25mA.xml	2020/11/16 9:17	XML 文档	1 KB
LOTO-i01-40mA.xml	2020/11/16 10:14	XML 文档	1 KB
LOTO-i01-60mA.xml	2021/1/4 9:38	XML 文档	1 KB
LOTO-i01-125mA.xml	2020/11/16 9:17	XML 文档	1 KB
LOTO-IDM02 20V.xml	2020/11/17 15:04	XML 文档	1 KB
LOTO-IDM02 80V.xml	2020/11/17 15:05	XML 文档	1 KB
LOTO-IDM02 200V.xml	2020/11/17 15:05	XML 文档	1 KB
LOTO-IDM02 800V.xml	2020/11/17 15:16	XML 文档	1 KB
LOTO-IDM03 20V.xml	2020/11/17 15:06	XML 文档	1 KB
LOTO-IDM03 80V.xml	2020/11/17 15:06	XML 文档	1 KB

We just copy a ready-made file and change it to our custom probe.

The first is the file name, and the file name will appear in the drop-down selection list of the channel custom probe, so this name is easy to remember and it is enough to identify, preferably the model of the probe, try to avoid using Chinese, so as not to cause garbled file paths in some systems. But do not change the suffix of the file, it must be .xml.

After modifying the file name, which is the name of our probe, you can use the system's notepad or wordpad to open the file and modify the content of the file.

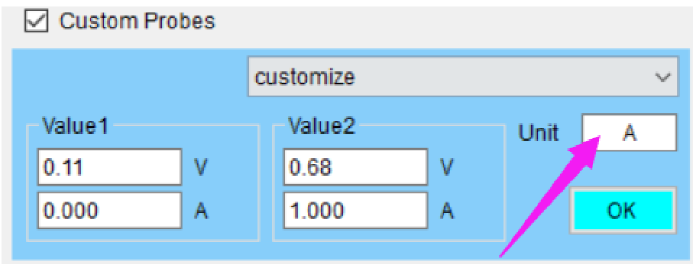
Only a few places need to be modified, and other places do not need to be changed.

The format description is as follows:

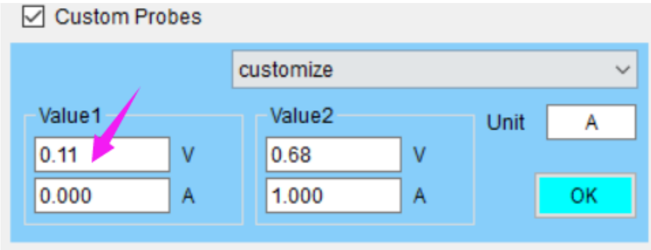
<?xml version="1.0" encoding="utf-8"?> // No modification required, XML file format

<Probe>

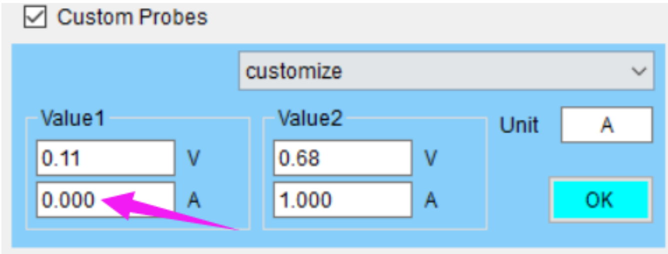
<Unit> A</Unit> //The physical measurement unit of the probe/sensor, such as the current probe is A (ampere), corresponding to the position indicated by the arrow below:



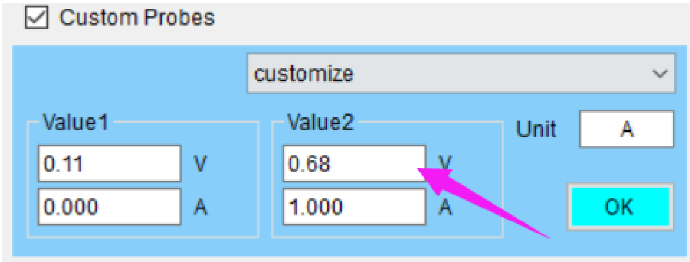
<ParameterV_0> 0.11 </ParameterV_0> // It needs to be modified, corresponding to the voltage value V of value 1, as shown by the arrow in the figure below:



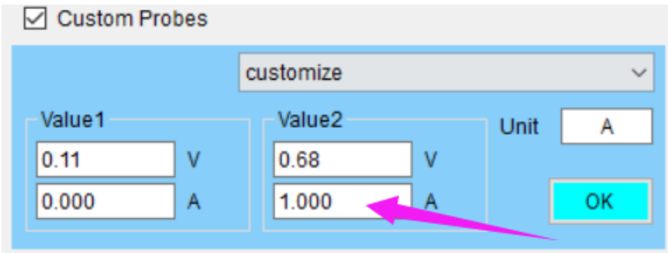
<ParameterU_0> 0.0 </ParameterU_0> // It needs to be modified, corresponding to the physical value of value 1, as shown by the arrow in the figure below:



<ParameterV_1> 0.68 </ParameterV_1> // It needs to be modified, corresponding to the voltage value V of value 2, as shown by the arrow in the figure below:



<ParameterU_1> 1.000</ParameterU_1> // It needs to be modified, corresponding to the physical value of value 2, as shown by the arrow in the figure below:



</Probe>

After the XML file is added and saved, close the oscilloscope software and run the software again, and the newly added custom probe will appear in the custom probe drop-down menu.

13.4.1 Video demonstration:

<LOTO Oscilloscope --- How to add your own customer probe with XML file?>
https://www.youtube.com/watch?v=L3OT4P0F9k0&t=44s

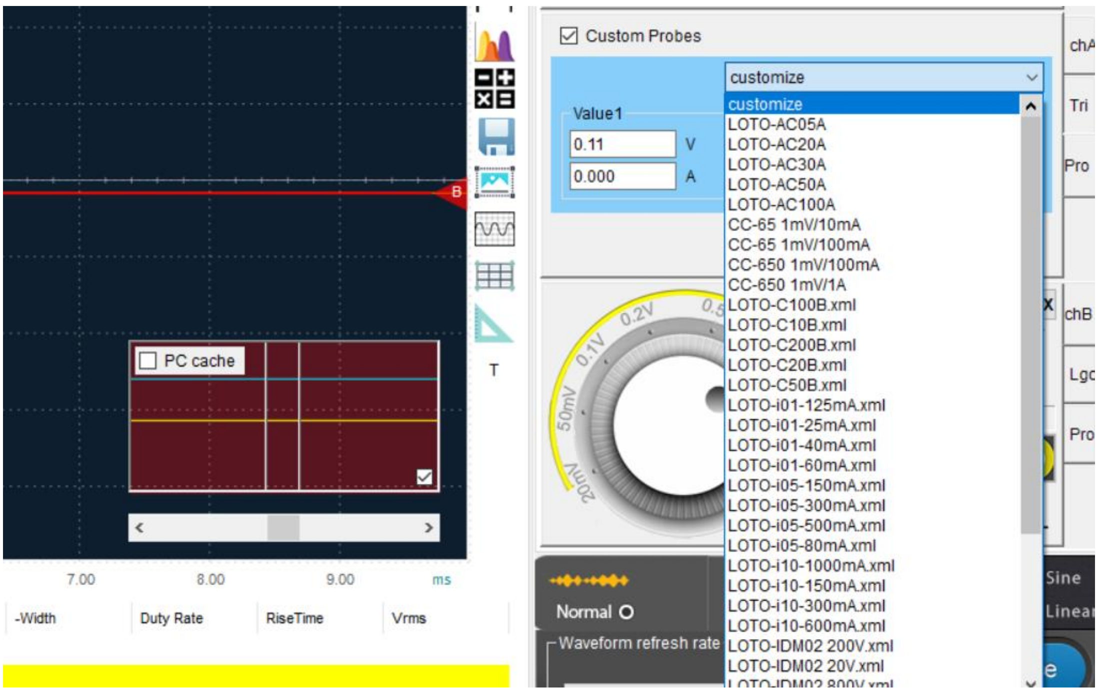
13.5 LOTO Official Probe Support

After understanding the saving of custom probes introduced in the previous section, we can save our own probes. In the same way, LOTO officially makes corresponding XML files for its various current probes, high-voltage differential probes, etc., based on the function of custom probes to save XML files.

In this way, when customers use various probes and sensors of LOTO, they can directly find the corresponding model from the drop-down list of custom probes and use them directly. Don't need to care about conversion parameters anymore.

Customers can also send us the parameters of their own probes, and we can make a file as long as it is stored in the specified directory, and it can directly become an officially supported custom probe.

If the customer feels that the parameters of the previous custom probe are not accurate enough and needs to be adjusted, the method in the previous section can also be used to directly modify the XML file to correct the probe/sensor parameters.



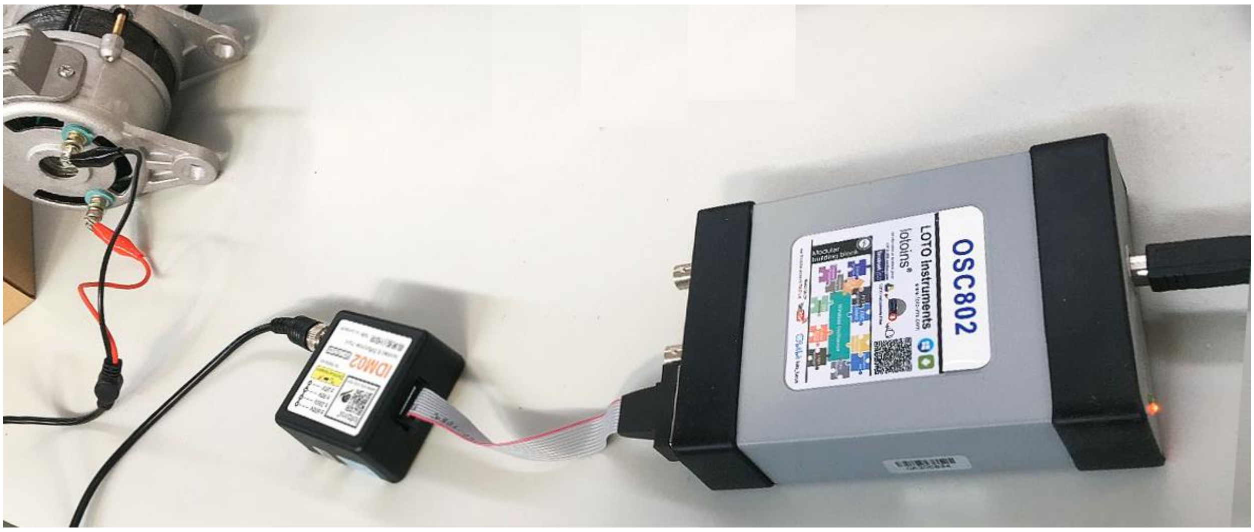
14. Isolated & Differential module IDM0x

At present, all models of LOTO oscilloscope products support the expansion function of the isolated differential expansion module.

For specific models and parameters, please refer to the hardware data sheet of the product. If the purchased product does not include a high-voltage isolation differential module, the software functions in this part will not be effective.

LOTO's high-voltage isolation differential module is realized in software based on the custom probe in the previous chapter, but we have placed the XML file of the corresponding proportional parameter of the module in the factory software, so we can directly choose to use it in the software The corresponding isolation module model and gear can be used. See Section 13.4 for methods and procedures.

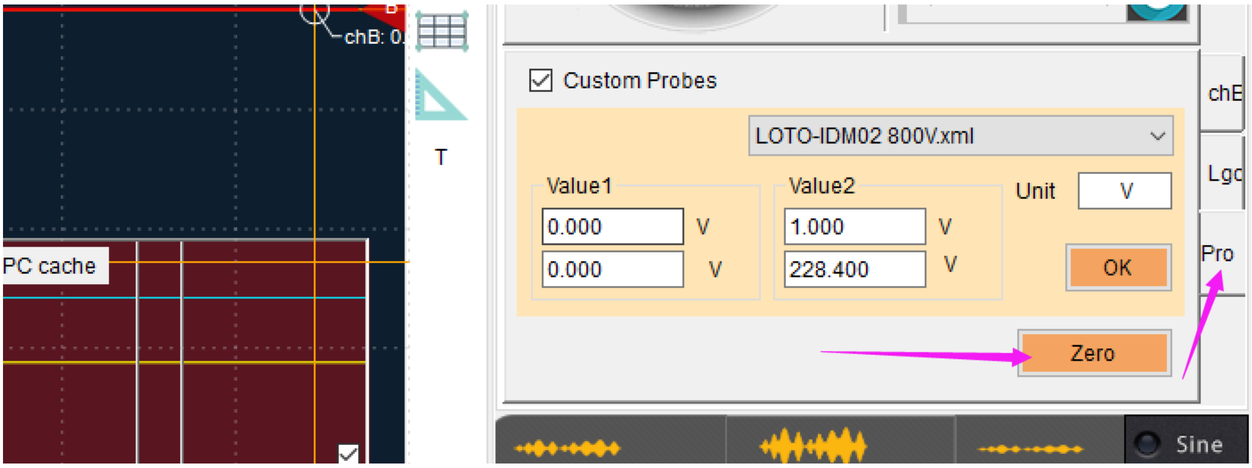
Let's take IDM02 (a high-voltage isolated differential module with a bandwidth of 100K) as an example. For example, we connected the oscilloscope and IDM02 to test the motor waveform as shown in the figure below:



14.1 Zero voltage position calibration

We will calibrate the zero position of the module when making the module. Customers generally do not need to deal with it, but after long-term use, or with seasonal temperature changes or component aging, zero position offset may occur. Therefore, before signal measurement, it is sometimes necessary to calibrate the zero position of the differential module. In order to avoid measurement value errors in the measurement results.

The step of zero position calibration is, after connecting the isolated differential module, observe that the position of the zero noise line of channel chB does not coincide with the position of the reference line of channel chB, and then click the zeroing button to clear. (Note: When calibrating the zero position, the input BNC connector of the module is required, without any external electrical signal)



After calibrating the zero voltage position, it can be used normally. If the user changes any of the following settings for channel chB, the zero position may needs to be recalibrated:

1. The voltage wheel gear position of chB has changed;

2. The coupling mode of the channel chB has changed (AC / DC);

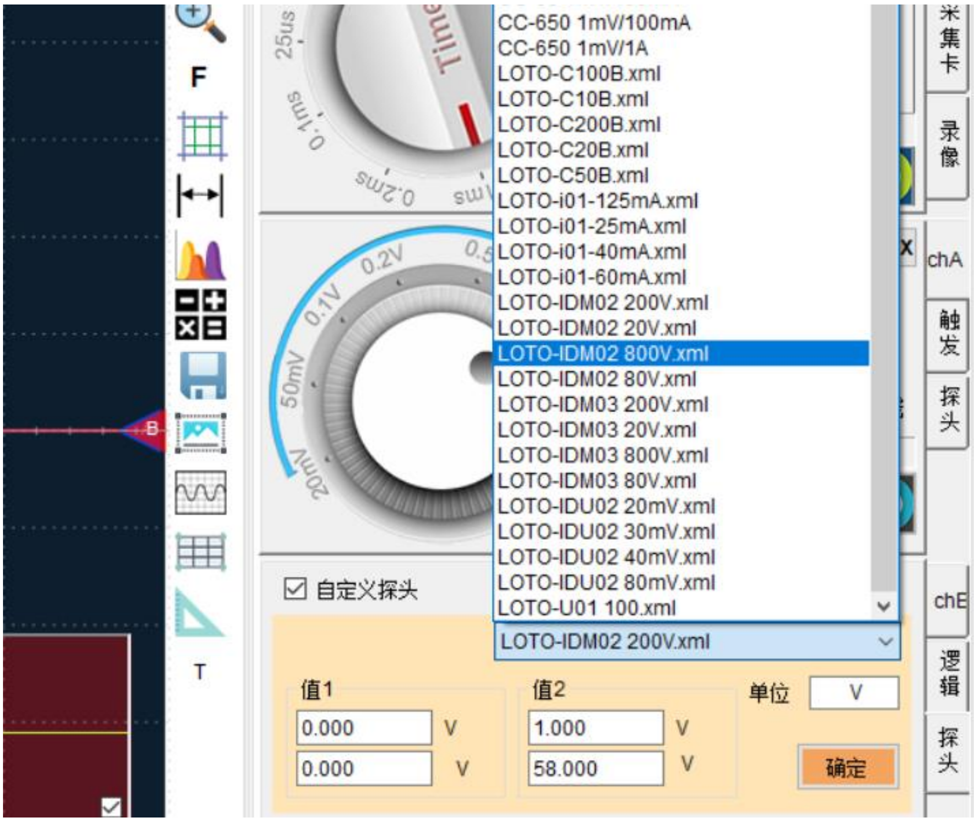
Note: If you are using the channel chB coupling method for AC coupling after isolating the differential module from the connection, you don't need to calibrate the zero position.

14.2 Signal waveform frequency setting

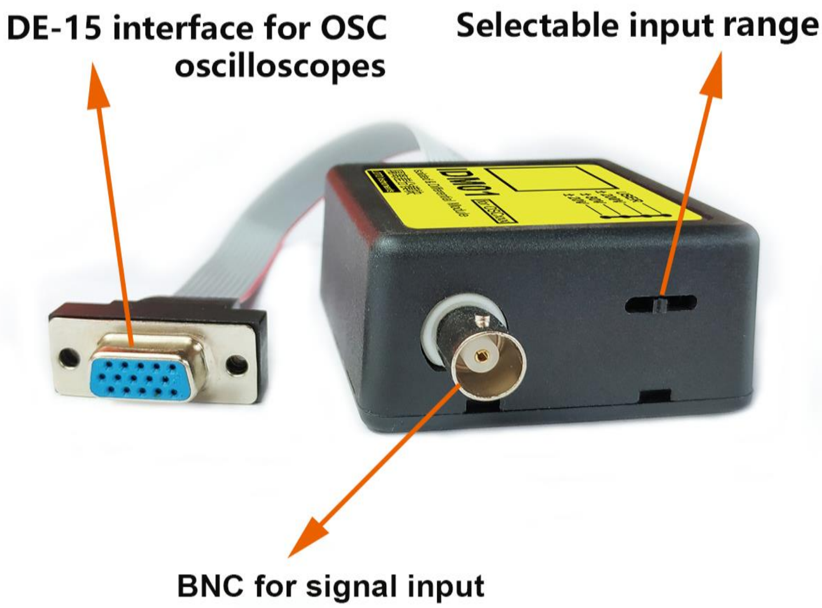
The range of the isolated differential module is parasitic on the channel chB and has its own range. Take one of the isolated differential modules IDM02 as an example:

Gear	Input range
20V	-20V ~ +20V
80V	-80V ~ +80V
200V	-200V ~ +200V
800V	-800V ~ +800V

Module gear setting area in the software:



The gear switch corresponding to the isolated differential module:



14.3 Video demonstration:

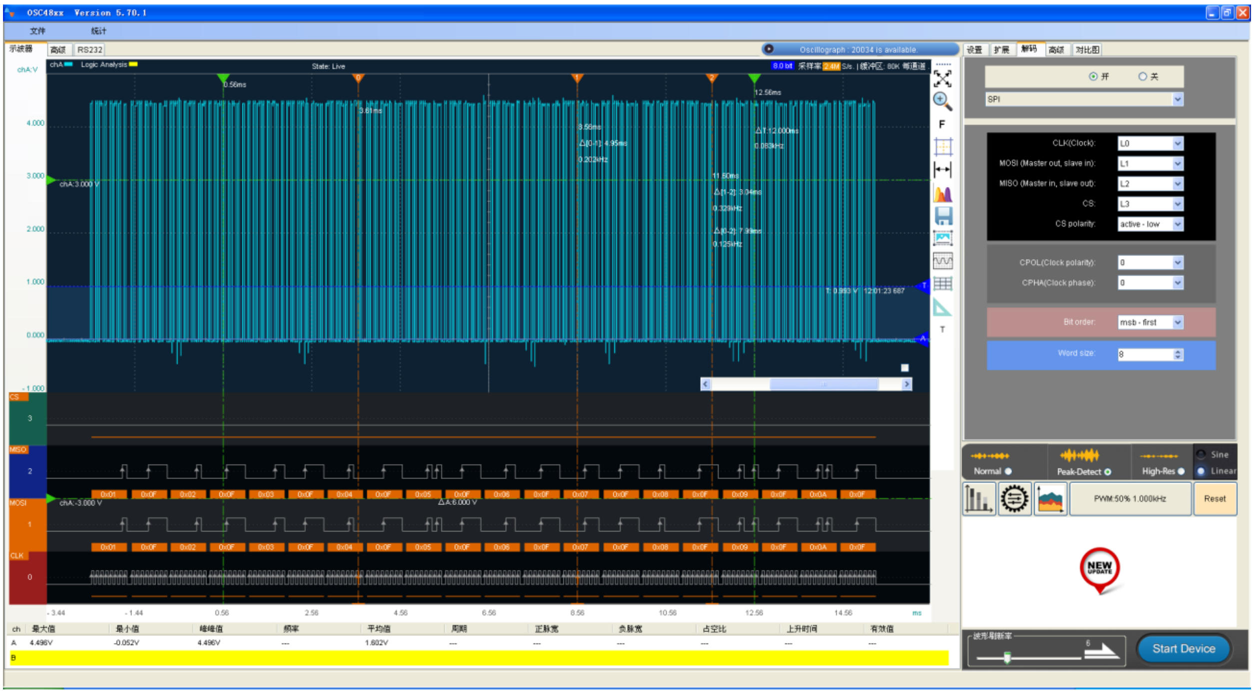
<LOTO Oscilloscope||Two methods to measure high voltage>
https://www.youtube.com/watch?v=LaSExagxOk8&t=2s

Note:

The differential module gear position switch and the software differential module gear position are mechanical operation and software operation, and the two cooperate with each other. It is inevitable to forget to keep their settings consistent at some time. This causes waveforms or measured values, resulting in ploidy errors. This requires more seriousness and care.

15. Logic analyzer function

Some models of products support the extended functions of the logic analyzer module, and you can check the hardware data sheet of the product. If the purchased product does not contain or does not support the logic analyzer module, this part of the software function will not be able to realize the logic analyzer function.

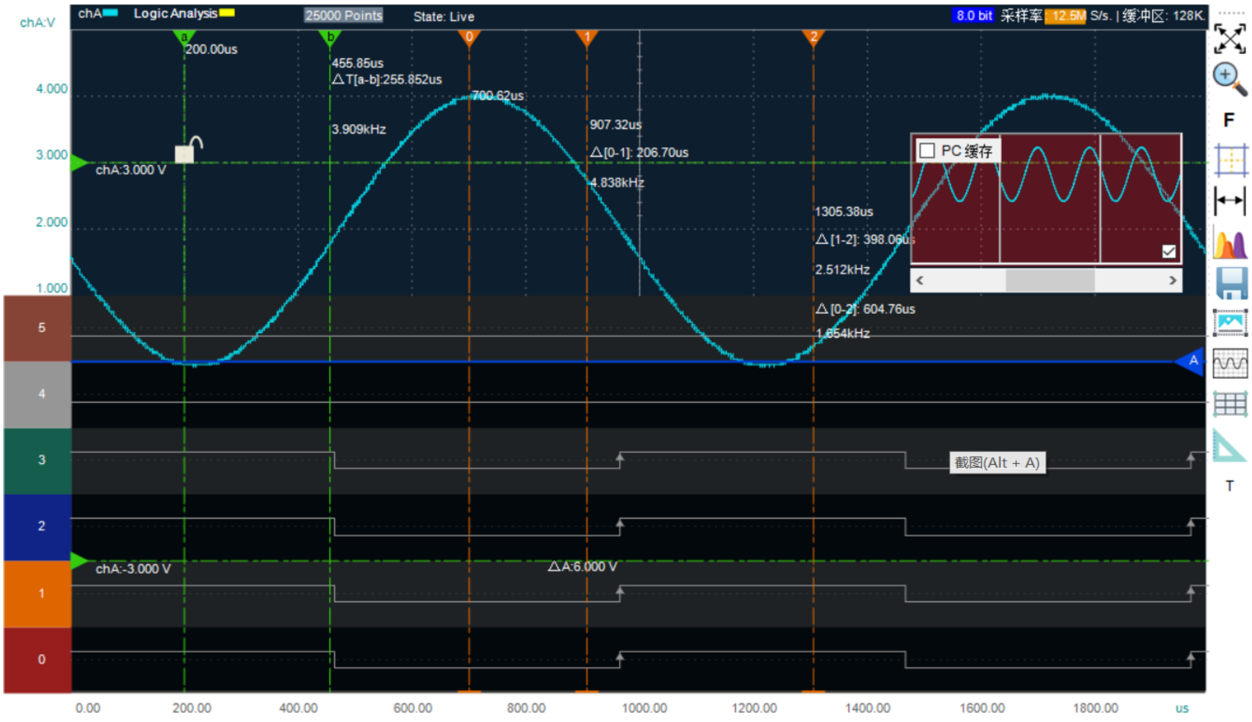


15.1 Video demonstration：

<LOTO Oscilloscope || Logic Analyzer Module demonstration:>
https://www.youtube.com/watch?v=_GJZYLYm8TY

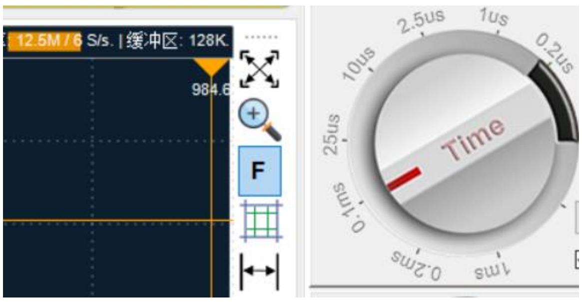
15.2 Simultaneous use of logic analyzer and oscilloscope

The logic analyzer and the channel B of the oscilloscope are mutually exclusive in hardware, choose one of the two. However, the logic analyzer can be used and displayed simultaneously with channel A of the oscilloscope, as shown in the figure below:



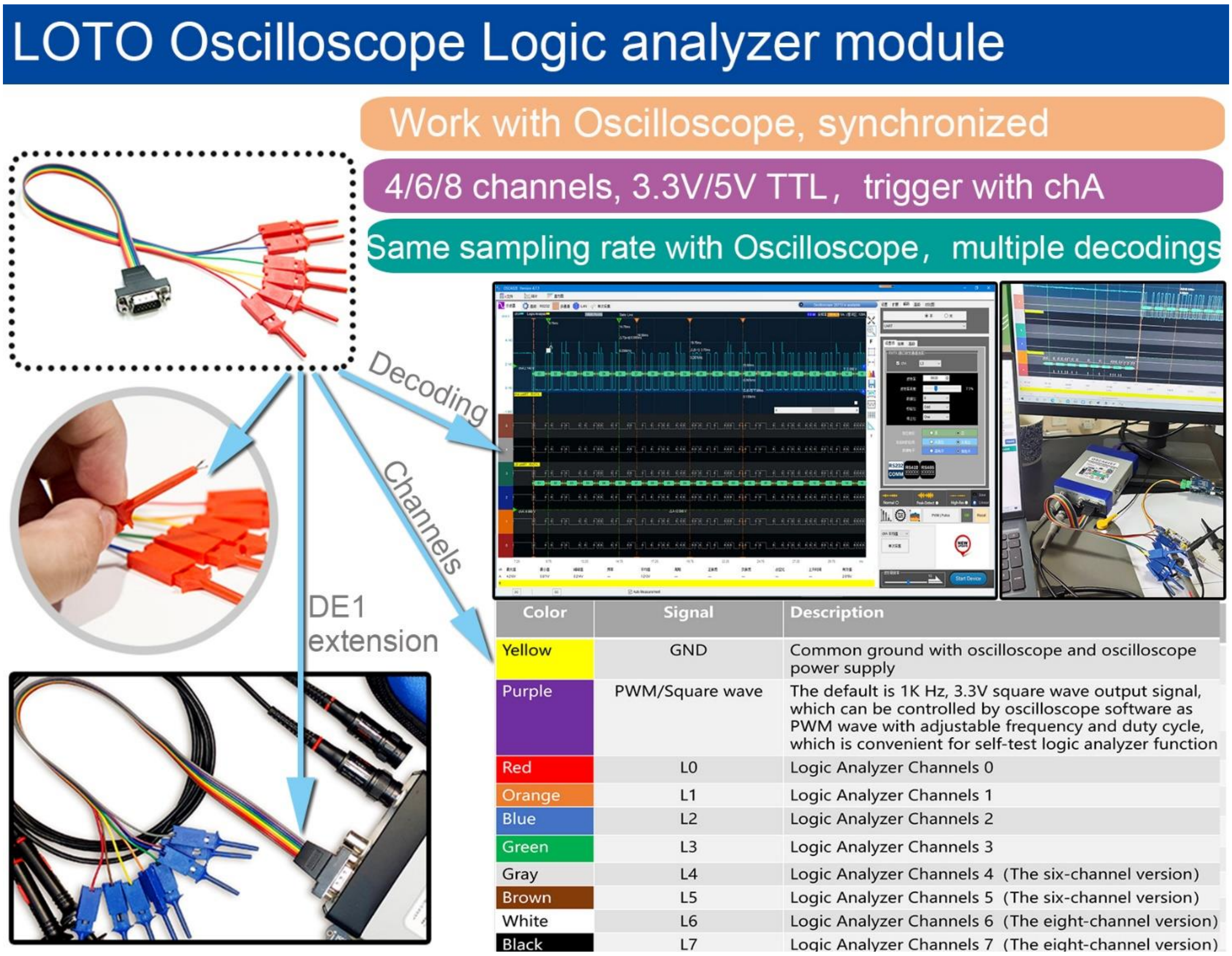
15.3 Time Synchronization for Logic Analyzers

The oscilloscope and logic analyzer are synchronized in time, they have the same sampling rate. Use the same time slot roulette. For example, the highest sampling rate of OSCA02 is 100M, then all channels of the logic analyzer and channel A of the oscilloscope can reach a maximum sampling rate of 100M. Of course, the sampling rate may change with the change of the time wheel, but the change is for the oscilloscope and logic Analyzer is consistent.



15.4 Wiring the Logic Analyzer

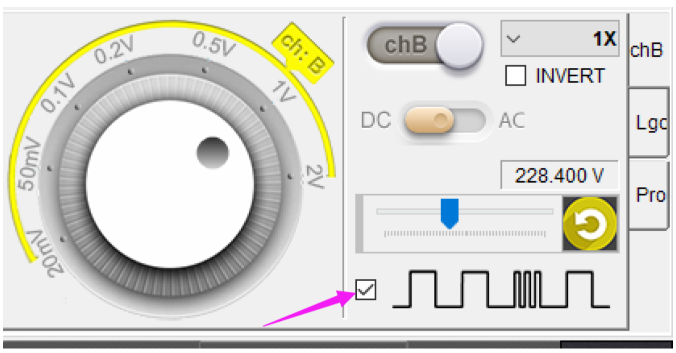
The logic analyzer module and the oscilloscope share the same ground. The yellow wire of the logic analyzer cable is GND. Pay attention to good grounding during testing. The logic analyzer module is compatible with 3.3V TTL and 5V CMOS level standards, and cannot be used to measure voltage logic higher than 5V. The extension port used by the logic analyzer module is the DE1 port of the oscilloscope, as shown in the figure below:



Color	Signal	Description
Yellow	GND	Common ground with oscilloscope and oscilloscope power supply
Purple	PWM/Square wave	The default is 1K Hz, 3.3V square wave output signal, which can be controlled by oscilloscope software as PWM wave with adjustable frequency and duty cycle, which is convenient for self-test logic analyzer function
Red	L0	Logic Analyzer Channels 0
Orange	L1	Logic Analyzer Channels 1
Blue	L2	Logic Analyzer Channels 2
Green	L3	Logic Analyzer Channels 3
Gray	L4	Logic Analyzer Channels 4 (The six-channel version)
Brown	L5	Logic Analyzer Channels 5 (The six-channel version)
White	L6	Logic Analyzer Channels 6 (The eight-channel version)
Black	L7	Logic Analyzer Channels 7 (The eight-channel version)

15.5 Software Switches and Channel Displays for Logic Analyzers

The logic analyzer channel is multiplexed from the ADC data line of the B channel. So we will see that the switch of the logic analyzer function is in the setting area of channel B of the oscilloscope, as shown in the figure below:

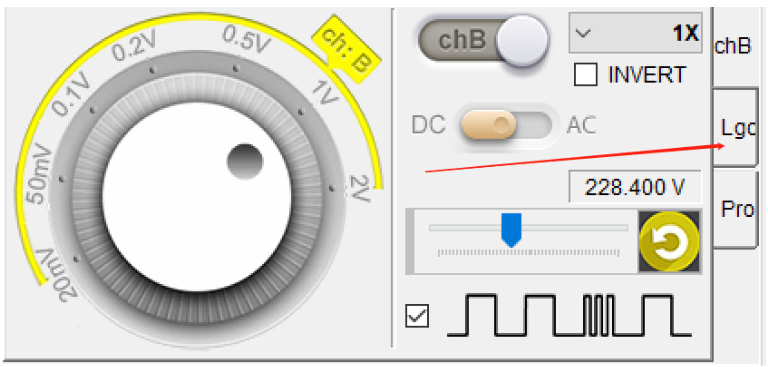


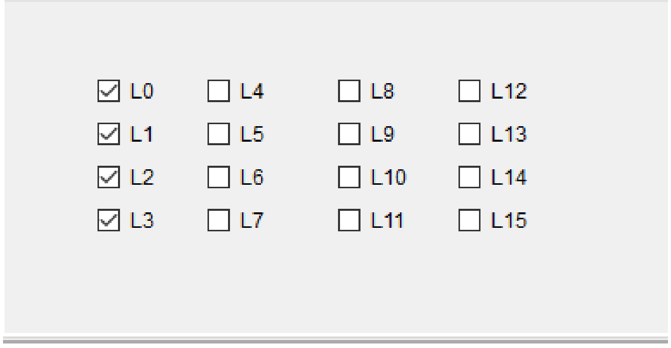
After checking the function switch of the logic analyzer, the display area of the logic analyzer will appear in the lower part of the waveform display area of the oscilloscope PC software:

In the figure, L shows the display area of the logic analyzer, and A shows the waveform of channel A of the oscilloscope. So if the logic analyzer module you purchased has n channels, it is actually equivalent to n+1 channels when used, and channel A is an analog channel, which can see more detailed details than the logic analyzer channel.

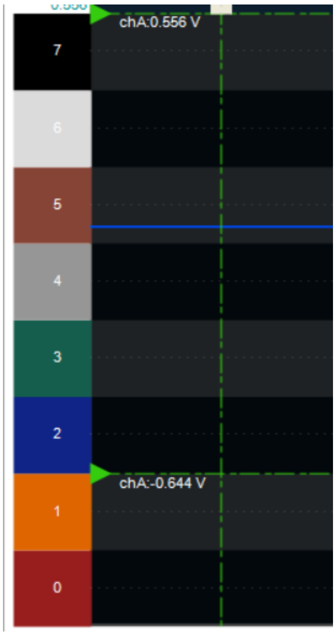
Channel A is the oscilloscope channel, which is also used as the trigger channel of the logic analyzer. For the setting of the trigger function, refer to Section 8.2.14.

The number of channels of the logic analyzer may be different for different types of oscilloscopes, and may be 4 channels, 6 channels or 8 channels. The logic channel to be displayed can be selected on the host computer software. Of course, the actual function needs to match the number of hardware channels of the module. For example, if you buy a 4-channel logic analyzer, it supports up to 4-channel waveform display (L0-L3).

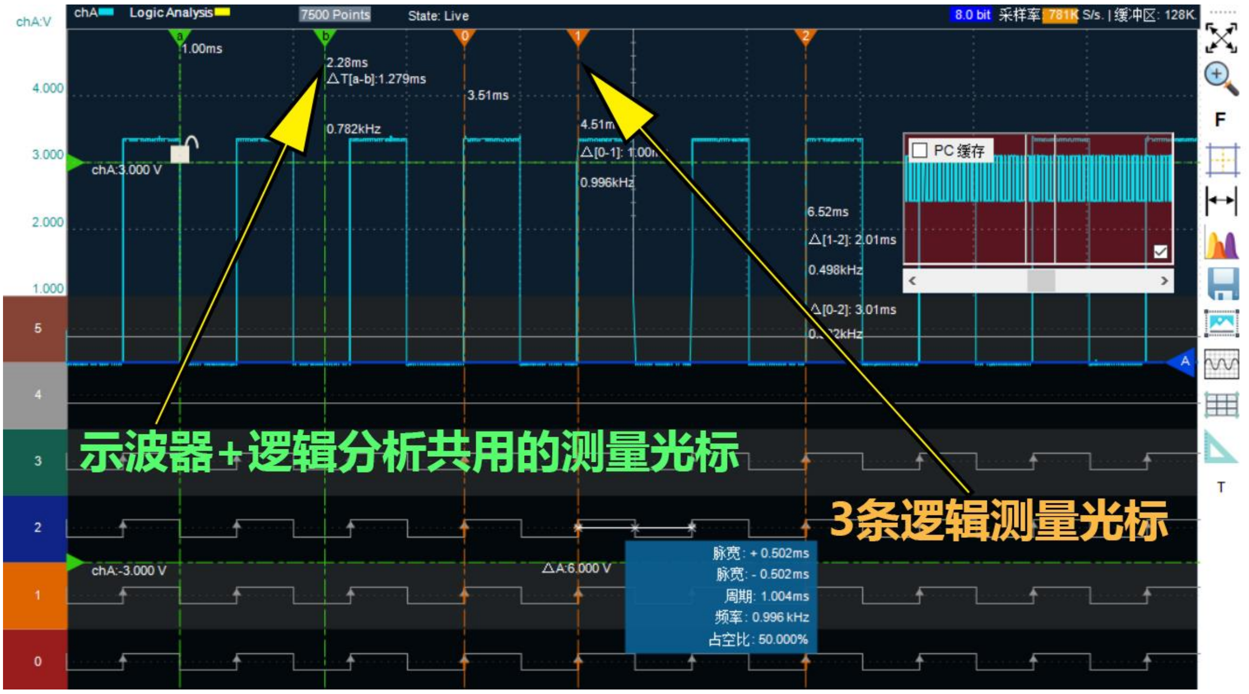




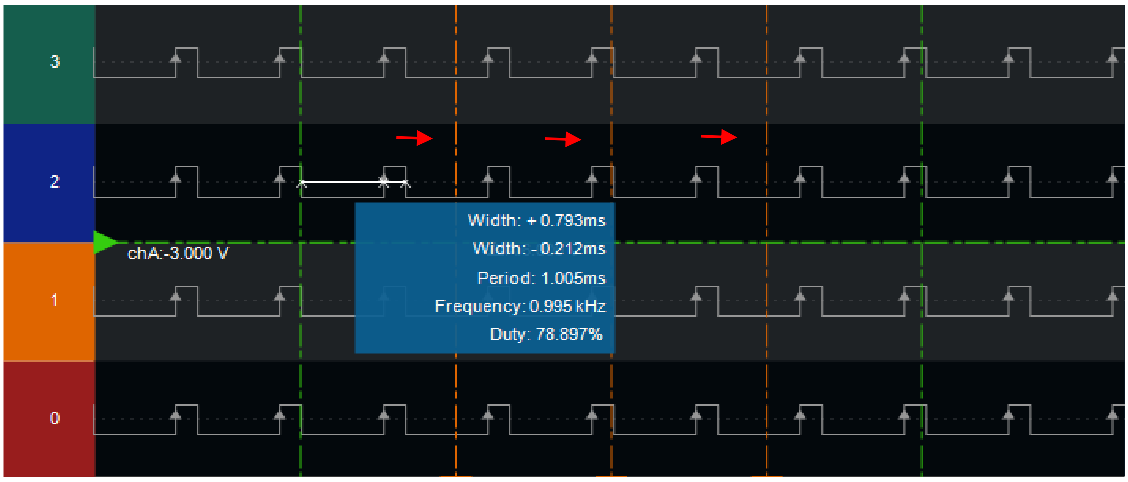
After selecting the displayed logic analyzer channel, it will take effect in the logic analyzer display area. It will be displayed according to the enabled logic channel. The leftmost is the channel number of the logic analyzer, and its color block is the corresponding logic analyzer cable. color. As shown in the figure below, for example, the blue channel number is 2, that is to say, the logical channel of L2, its waveform, on the hardware wiring, corresponds to the blue cable in the cable.



15.6 Cursor Measurements for Logic Analyzer



As shown in the figure above, the logic analyzer and the oscilloscope are used at the same time. The green cursor is the hash cursor of the oscilloscope. As mentioned in the part of the oscilloscope, these oscilloscope cursors can also be used for the logic analyzer at the same time. The logic analyzer will also have orange 3-bar logic cursors. Like the green cursor of the oscilloscope, you can drag the mouse to the desired position, and then the time axis information of the cursor and the time difference information between the cursors will be displayed directly next to the cursor.



In addition to cursor measurement, there is also a way to automatically display information: move the mouse cursor to any position of the waveform, and the pulse width, period , frequency and occupation of the signal at this position will be displayed in real time.

15.7 Decode function of logic analyzer

Because the LOTO oscilloscope and logic analyzer are used at the same time, we recommend using the oscilloscope first to decode the communication signals of one line and two lines. We have described oscilloscope decoding in chapter 8.2.20.

The decoding function of the logic analyzer is the same, and you can also refer to the protocol decoding part of 8.2.20. The difference is that due to the logic level input limitation of the logic analyzer, some decoding cannot be performed on the logic analyzer, such as 232, RS485, RS422, and CAN bus that are not at the TTL level, all of which need to be on the oscilloscope channel decoding.

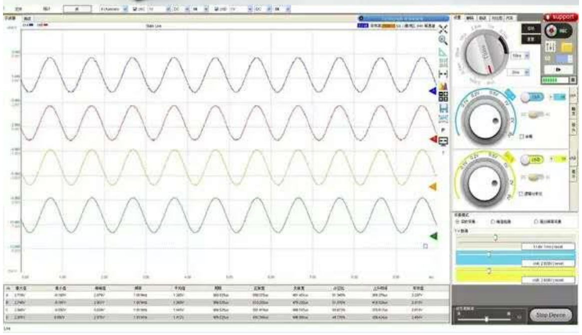
The decoding on the logic analyzer currently supports RS232, iic, and SPI of TTL.

With the above foundation, I took a video of the process of using a USB virtual oscilloscope + logic analyzer to do SPI testing and decoding, which is more intuitive than graphics, as shown below:

16 Four-channel oscilloscope

The LOTO oscilloscope has launched the cascading function, which can cascade multiple 2-channel standard oscilloscopes into a multi-channel oscilloscope, which will be introduced in detail in Chapter 17. This chapter introduces an integrated 4-channel product dedicated to automotive circuit diagnosis. Customers in other application fields can skip this chapter and read the introduction of the next chapter directly.

LOTO oscilloscope has specially launched a 4-channel product for the test application of automotive sensor waveforms: OSC984, which has 4 analog input channels and can measure 4 signals at the same time. As shown below:

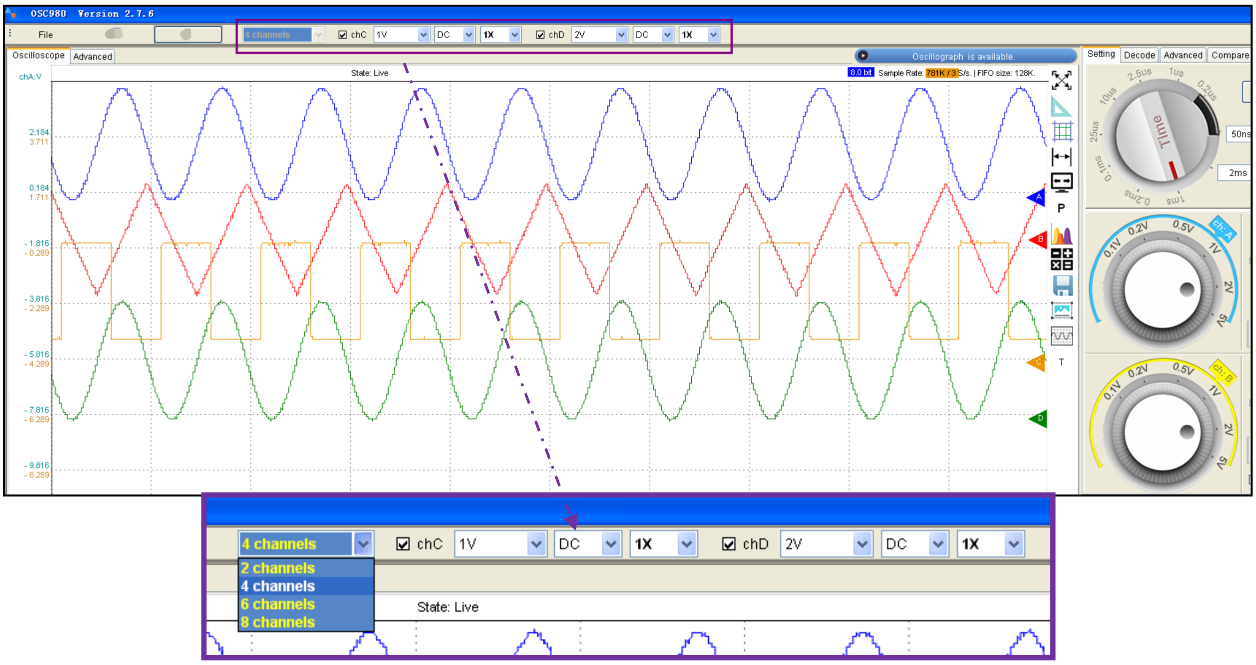


4-channel oscilloscope, USB1 is the master device, and USB2 is the slave device.

USB1 can be used as a single dual-channel oscilloscope.

When using a USB2 device, it must be supported by a USB1 host device.

The supporting software of the 4-channel oscilloscope is the same as that of the 2-channel OSC980. When using two channels, most of the operations are similar to those of other models. You can refer to other chapters of this manual.



16.1 4-channel oscilloscope OSC984 software startup steps:

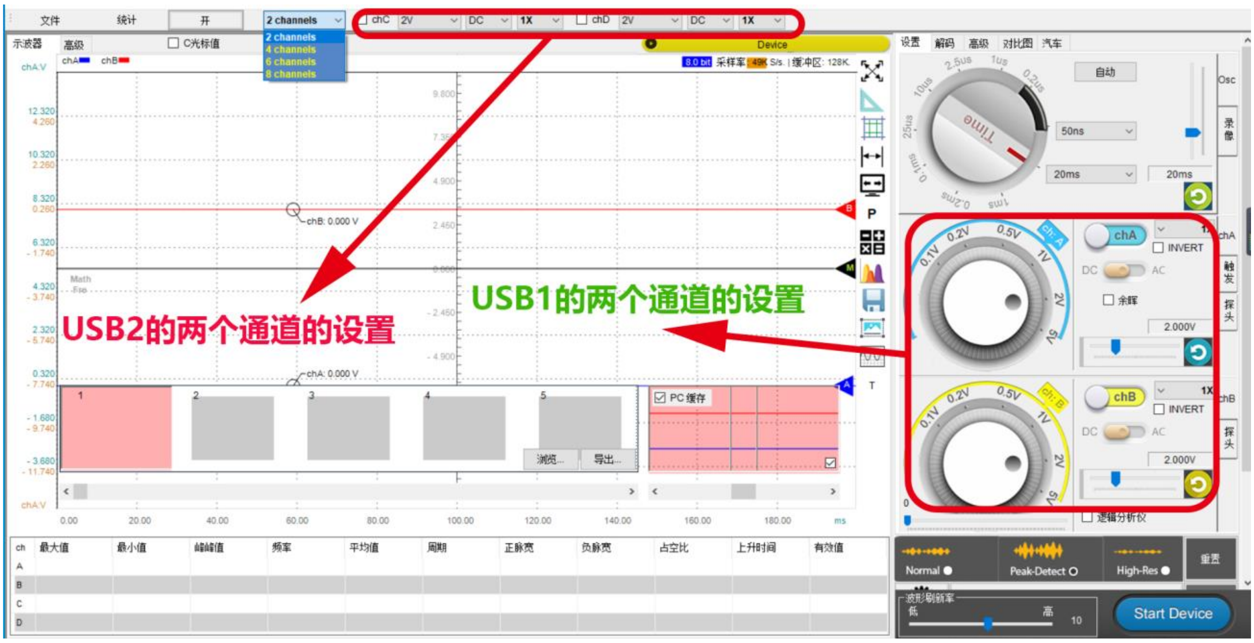
If the oscilloscope software is opened first, then plug in the oscilloscope device. The software will run automatically. The software selects 2 channels by default, a two-channel oscilloscope. At this time, you need to click the Start Device button to stop the oscilloscope software, and in the software menu bar, select 4 channels 4 channels, and then click Start Device to start the oscilloscope software.

If the 4 channels cannot be started normally, please check whether the cables of USB1 and USB2 have been plugged into the USB ports of the computer.

Since the 4-channel is equivalent to two 2-channel oscilloscopes superimposed and packaged together, it is necessary to check whether the drivers of USB1 and USB2 have been successfully installed.

Support 4-channel oscilloscope software, USB2 device (channel switch, channel voltage range, AC-DC coupling, probe multiple), select and set in the menu bar of the software.

The 4 channels are acquired synchronously in time, so they share the same time wheel and use the same sampling rate.



17 Multi-channel oscilloscope

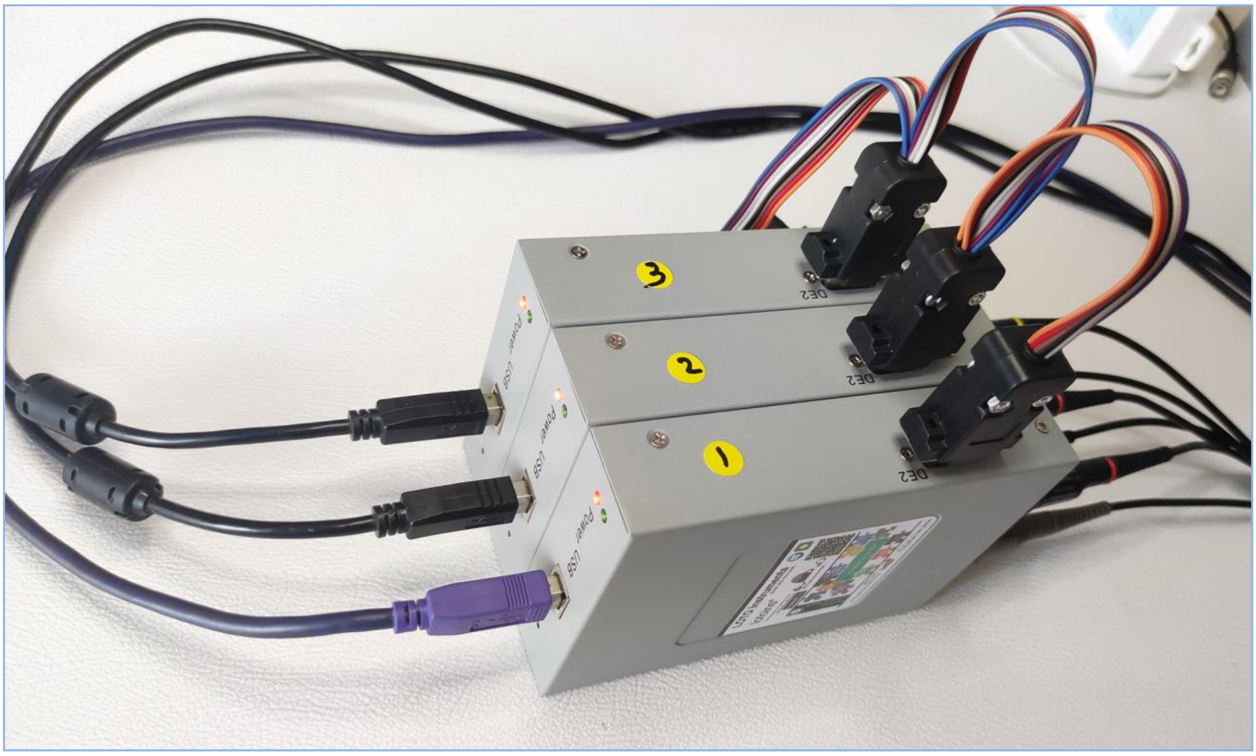
17.1 2 channels cascaded into multiple channels

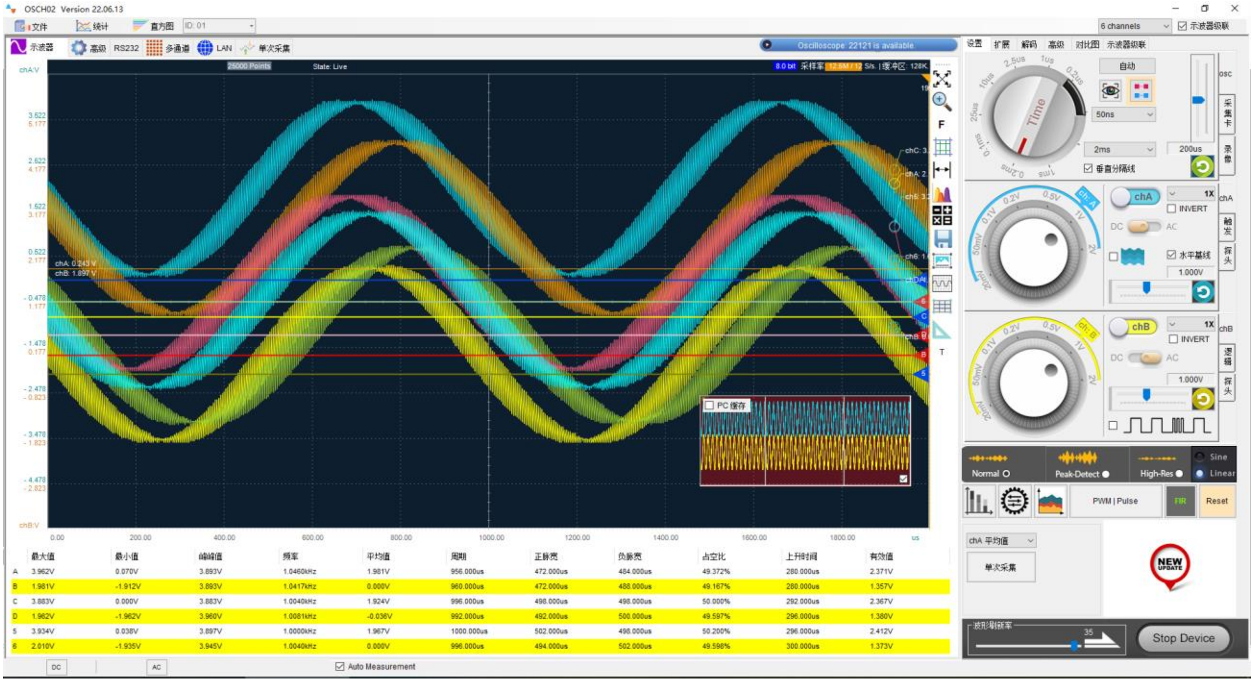
LOTO's standard oscilloscopes are generally 2-channel, except for the 4-channel that is specially introduced for automotive diagnostics mentioned in the above chapter.

Multiple LOTO 2-channel standard products can be cascaded to form a multi-channel oscilloscope. For example, two 2-channel OSCA02 can be cascaded into a 4-channel oscilloscope. 3 sets can form a 6-channel oscilloscope. And when not cascaded, it can be disassembled into multiple independent 2-channel OSCA02 oscilloscopes for use.

The cascaded oscilloscopes need to be connected with cascaded cables through the expansion port DE2 of the oscilloscopes in hardware, for the purpose of time synchronization in acquisition. The most cascaded oscilloscopes we have launched so far can achieve 4 cascaded oscilloscopes into 8 channels. These multi-channel waveforms are displayed in the same software interface, and the same sampling rate is used, and they are collected synchronously in time.

Cascading is just adding cascading lines to the hardware, and some internal synchronous designs, appearance and external wiring are the same as the original standard products.





17.2 multi-channel cascade device number online programming

1. When multiple oscilloscopes are cascaded, the software enforces the hardware ID of each device. Although we can program the hardware ID of the oscilloscope online, we need to program the specified number, otherwise the cascading will fail.

ID. 1, the main equipment number, (main control) oscilloscope. The channels are chA and chB on the software.

ID. 2, auxiliary equipment number, (auxiliary cascade) oscilloscope. The channels are chC and chD on the software.

ID. 3, auxiliary equipment number, (auxiliary cascade) oscilloscope. In software, the channels are ch5 and ch6.

ID. 4, auxiliary equipment number, (auxiliary cascade) oscilloscope. Channels are ch7 and ch8 in software.

If you want to cascade 2 oscilloscopes to form a 4-channel oscilloscope, the programming ID should be 1 and 2. Similarly, if you want to cascade 3 oscilloscopes to form a 6-channel oscilloscope, the device ID to be programmed should be 1/2/3.

2. When multiple oscilloscopes are cascaded, the software requires the hardware type, and each device needs to program the hardware device type.

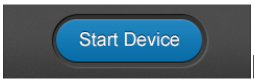
ID. 1, (main control) oscilloscope. The device type must be written as "Host", the master device.

Other ID, (auxiliary cascade) oscilloscope. The device type must be written as "Guests", auxiliary device.

HW ID	Host/guest	channels
1	Host	chA, chB
2	guest	chC, chD
3	guest	Ch5, ch6
4	guest	Ch7, ch8

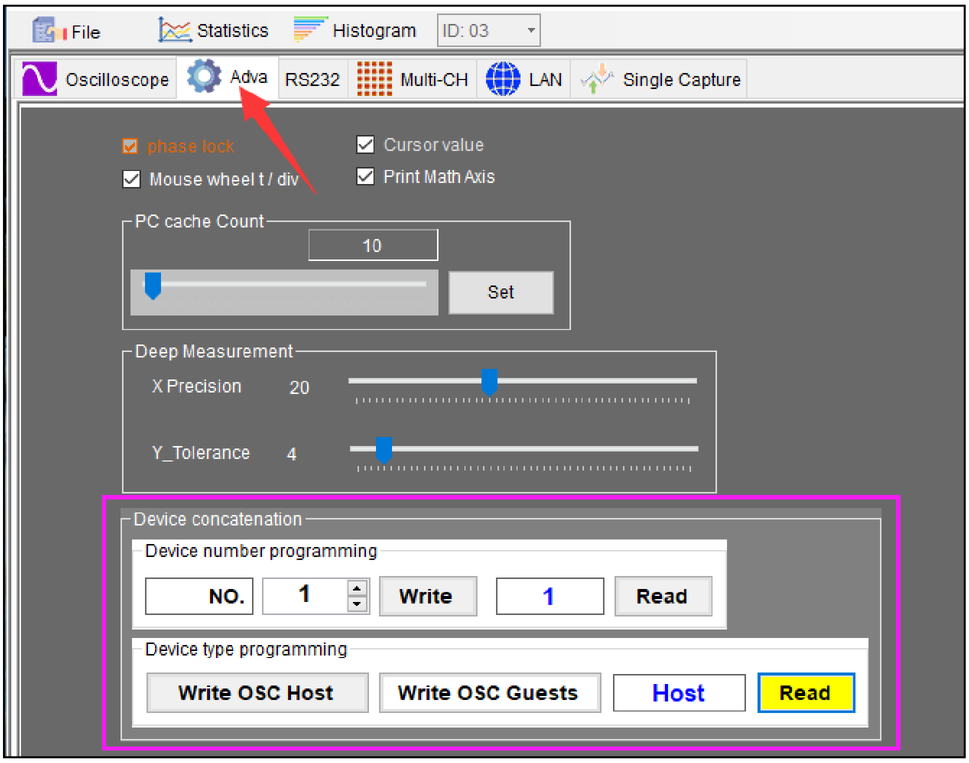
17.3 Device ID/master-slave programming

1. Connect the oscilloscope to be programmed with the ID to the USB port of the PC. Note that only one oscilloscope can be connected to each programming, to prevent connecting multiple oscilloscopes to open the device and programming confusion.

2. After the USB of the PC is plugged into the oscilloscope device, the same as the usual oscilloscope startup steps, click the 

button to start and connect the oscilloscope.

3. Enter the "Advanced" option page of the oscilloscope and find the "Device concatenation" group operation area.



4. Program the ID and program the device type.

For the main control oscilloscope, write "1" for the number and "Host" for the type.

Auxiliary cascade oscilloscope, the number is written 2/3/4.... in order, and the type is written "Guests".

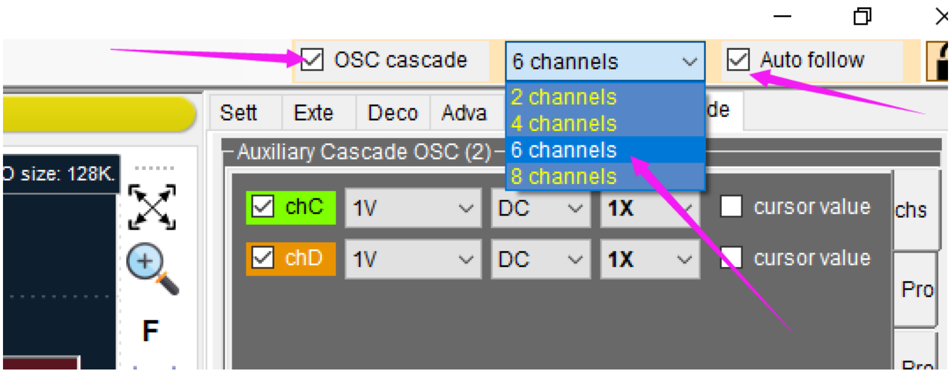
Note, do not jump between IDs. There has to be a number 2 in order to have a number 3.

When the number of the oscilloscope is programmed as 2/3/4.., not after the "1" number. Plugging these oscilloscopes into a PC alone will not work as a single two-channel oscilloscope.

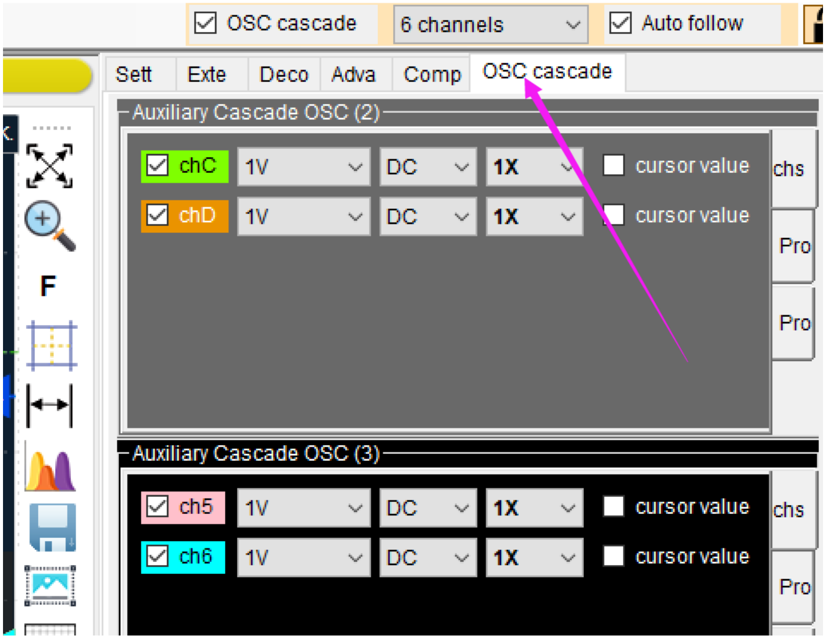
If you need to use it alone, you need to write its number as "1" and the device type as "Host".

17.4 Software Operation of Multi-channel Oscilloscope

Plug all the oscilloscopes with ID and device types into the USB of the PC and open the oscilloscope software. First, check "Oscilloscope Cascading", select the number of channels, and then click (Start Device) to start the oscilloscope software.



The auxiliary cascaded oscilloscope can be controlled in conjunction with the main oscilloscope (voltage and channel), or it can be controlled independently. The location of the controls is in the Oscilloscope Cascading tab:



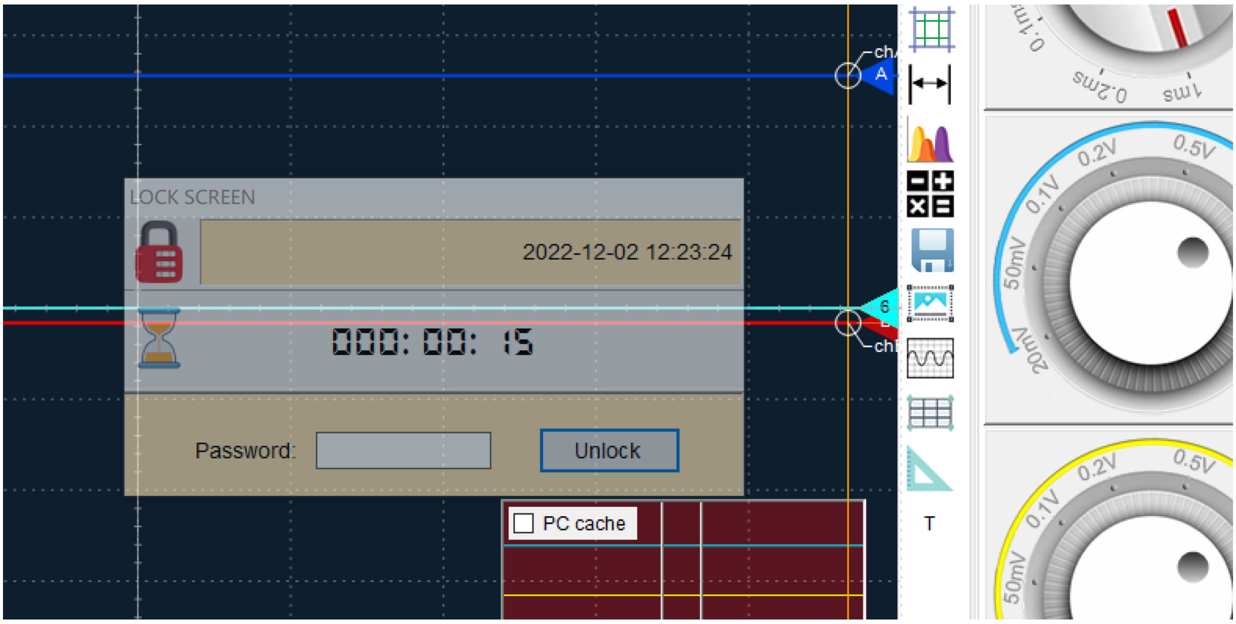
Note that when you want to switch the number of channels or adjust and change the number of cascaded oscilloscopes, you need to suspend the oscilloscope software to adjust the number of channels.

18 Screen lock function (software lock screen)

In order to prevent the oscilloscope software interface we set up from being unintentionally damaged by others, such as moving and pressing the mouse, or accidentally tapping the keyboard, etc., changing the settings of the oscilloscope, resulting in changes to the running data and waveforms and associated operations, LOTO oscilloscope has added the function of locking the screen.

As shown below:

After clicking this button, the software will pop up a dialog box under the current setting and display, as shown in the figure below:

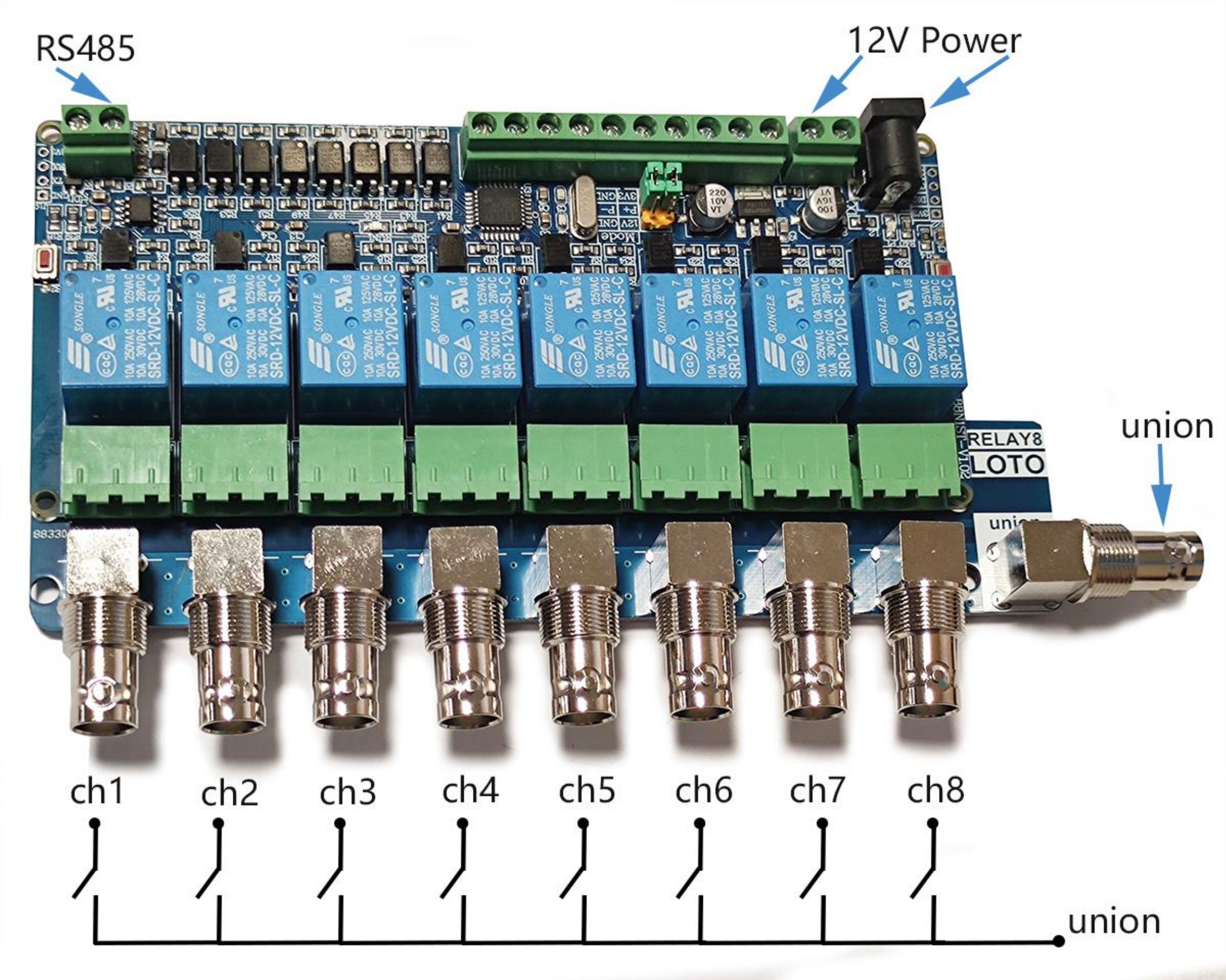


This dialog will display the current date and time, and how much time has elapsed since the screen was locked, and it will block any mouse and keyboard operations unless the lock is unlocked by entering the unlock code, or someone forcefully quits the current oscilloscope software.

The password to unlock the screen is 11235.

19 Eight Channel Switching Relay Board RELAY8

LOTO oscilloscope has launched a 8-to-1 relay switching board, which is convenient for customers to make software-controlled switching selections for 8-way input signals. In this way, 1 channel of the oscilloscope is used to collect 8 signals in time division, as shown in the figure below:



The relay switching board is RS485 interface, so the customer needs to connect to the communication interface of the switching board through the USB to RS485 module of the computer, and then on the software of the LOTO oscilloscope, we have corresponding control functions, as shown in the figure below:

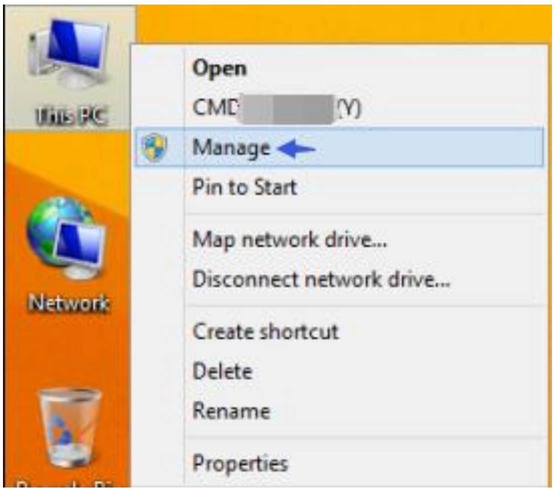
Set the serial port parameter settings shown in Figure 2 above on the interface of the "RS232" tab of the software, that is, a serial port that will be virtualized by the USB to RS485 conversion of the computer. As shown in Figure 3 above, any of the 8 channels can be selected to be connected.

If you need other ways of software control, you can contact us for customization.

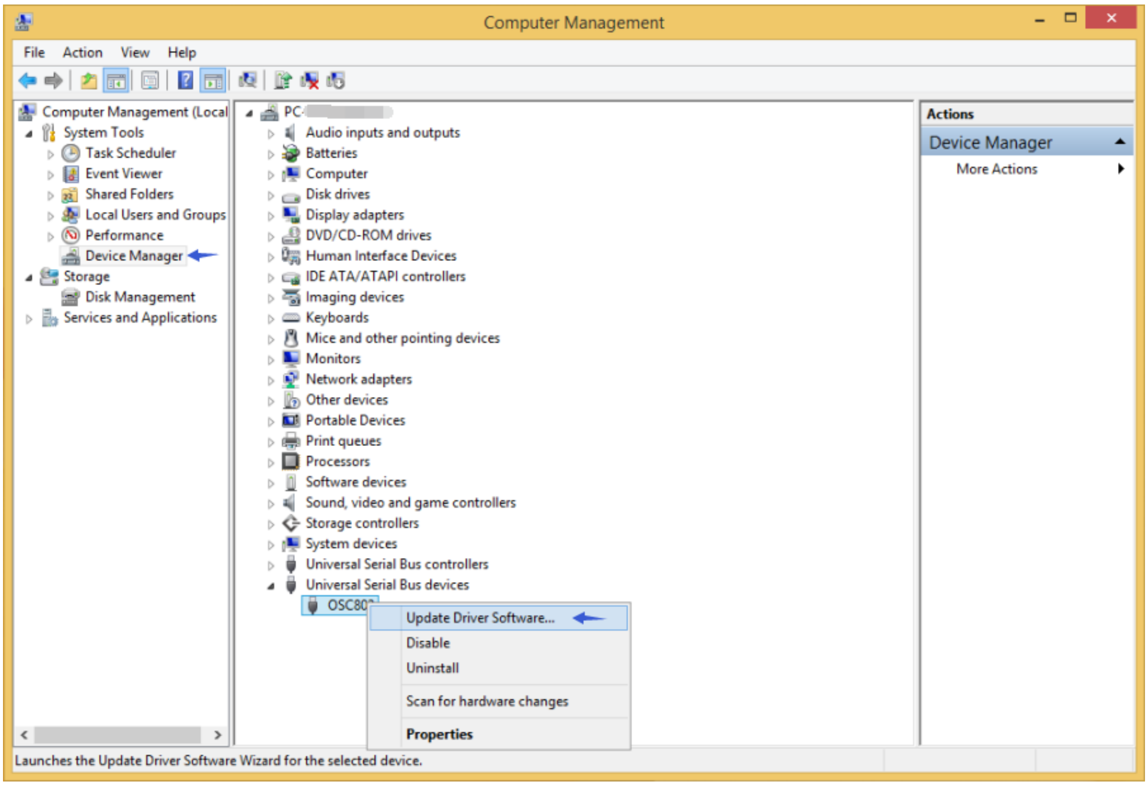
Appendix I: Driver installation under exceptional circumstances (Complete and detailed driver installation steps)

Manual install of the driver**:**

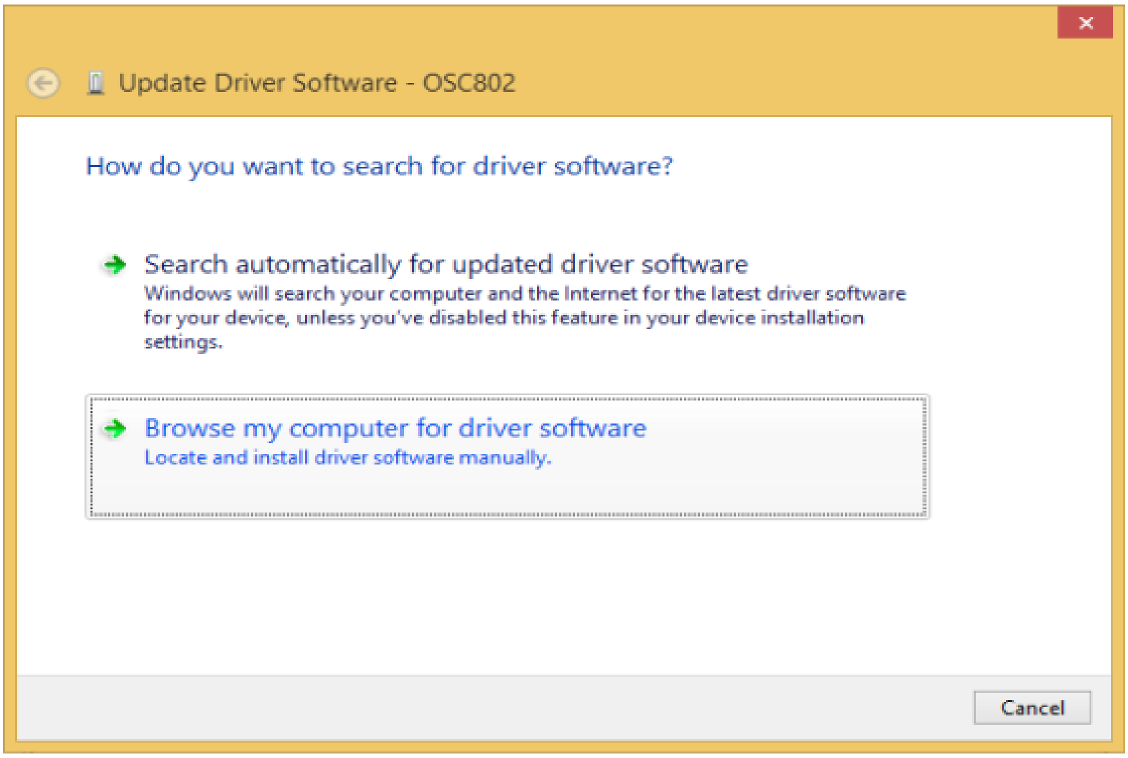
1. Right click on **Computer** and select **Manage**.



2. Enter in the **Device Manager**, and search for the device called **OSCxxx** inside the section **Other Devices** (if not found, unplug and plug again the device and wait), then right click on the device and select **Update Driver Software**:

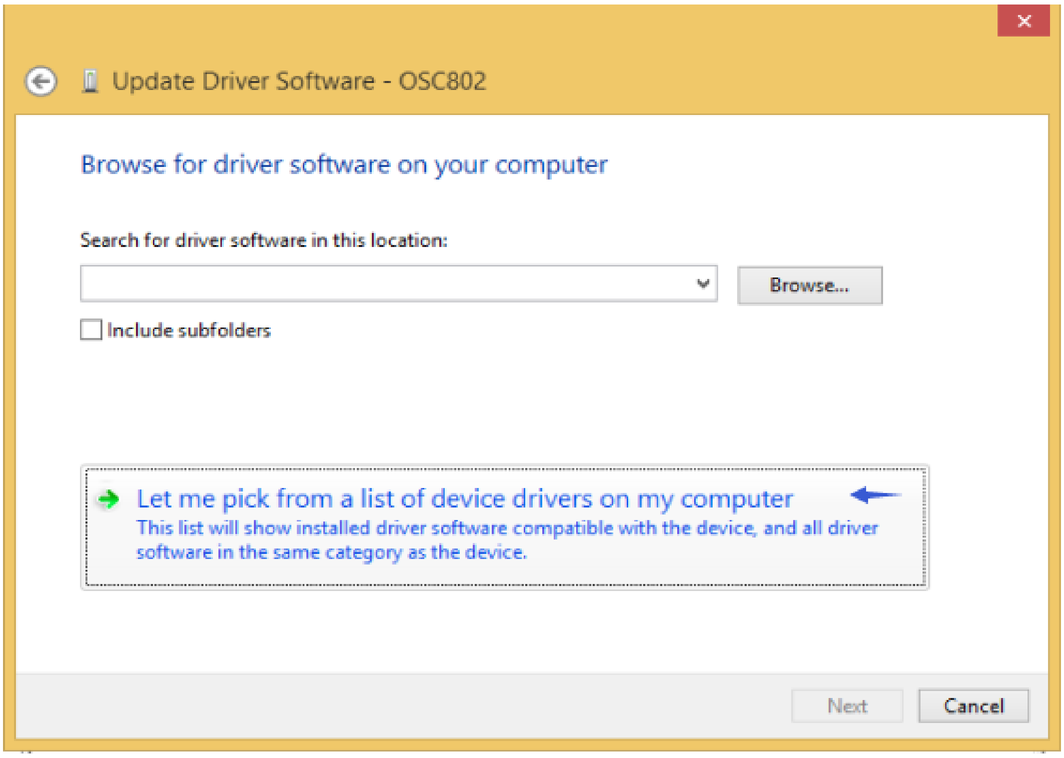


3. In the next dialog box select “ Browse my computer for driver software ”:

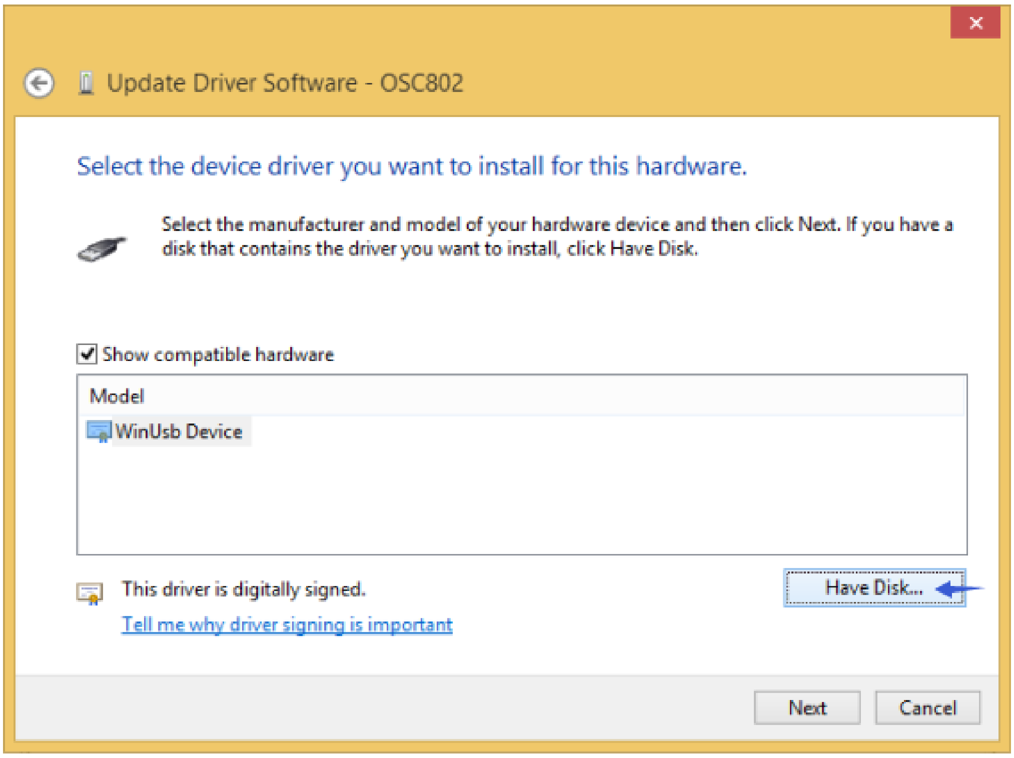


4. In the next dialog box, select " Let me pick from a list of device drivers... ":

Note
In many cases, in post-patched systems or systems with strict permissions, this second option is easier to succeed.



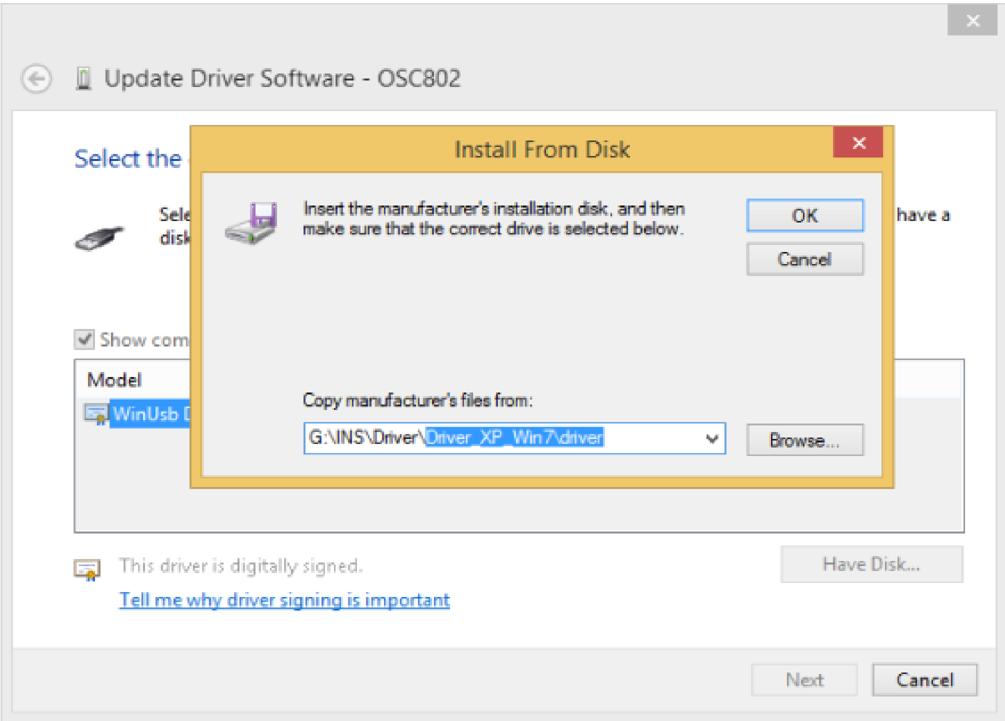
5. In the next dialog box, select "Have Disk..."



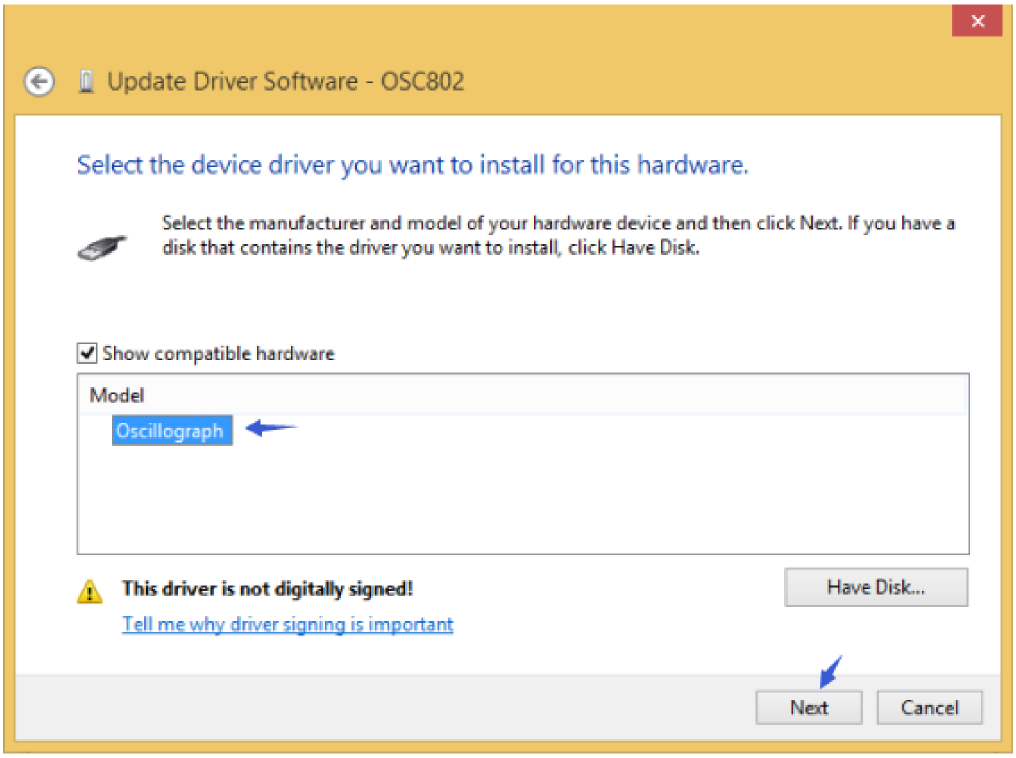
6. In the next dialog box, select **Browse** to select the driver path like " Driver_XP_Win7\driver\UsbLjtMS.inf " and click **OK**.

Note：

The driver XP_Win7 (Driver_XP_Win7\driver\UsbLjtMS.inf) is compatible with all other Windows versions.

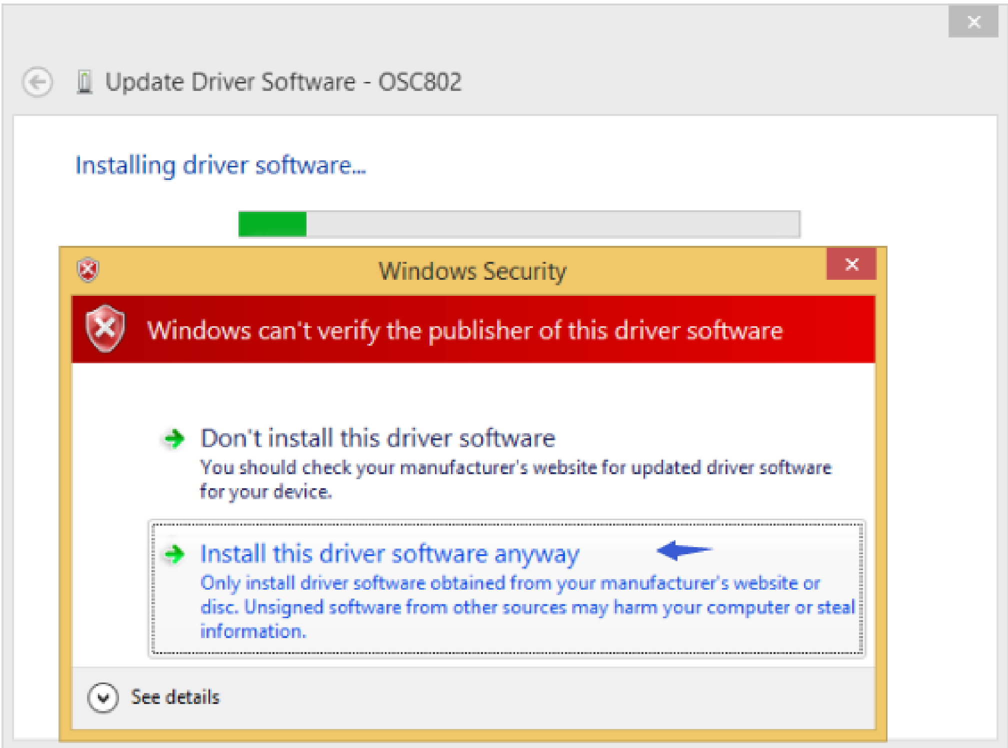


7. In the list of compatible hardware, select **Oscillograph** and then click on **Next**.



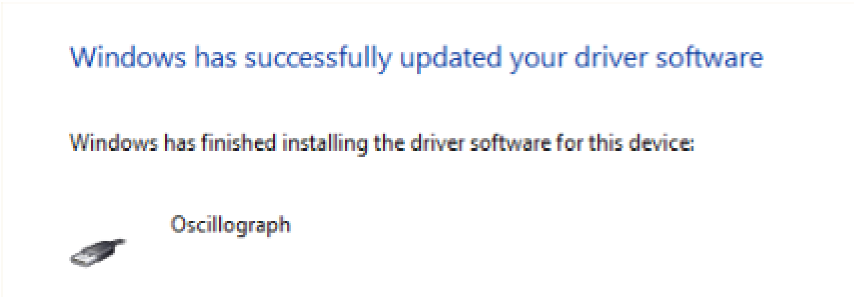
8. Select "Install this driver software anyway" :

> **Note:**
>
> In systems with more restrictions a warning window will appear. However, you can install the driver anyway since it is safe.



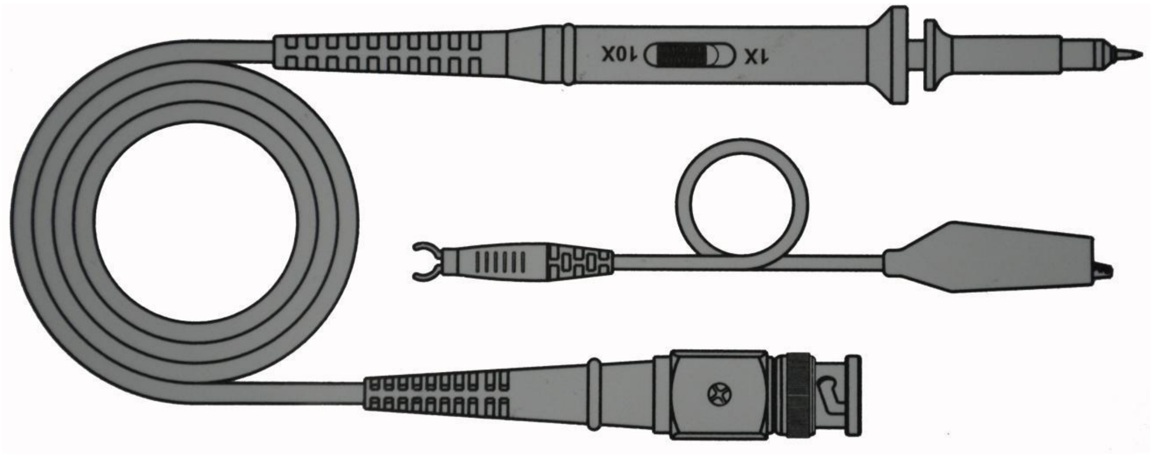
(Installation finished)

> **Note**
> In some systems, our products will be shown as "WinUsb Device", but after the correct driver installation it will be shown with the correct name.



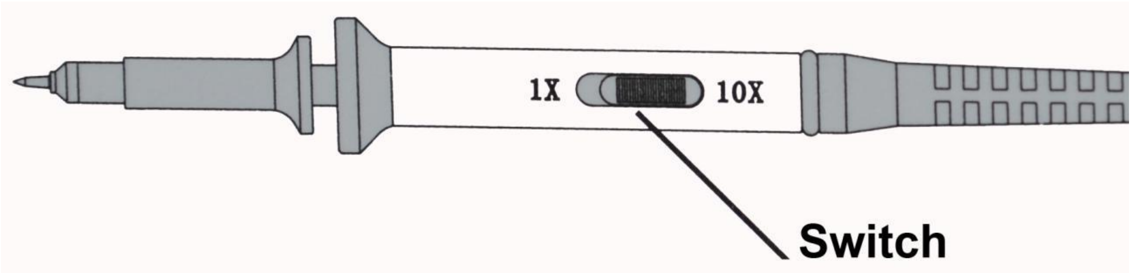
Appendix II: Probes

The device is provided with two 40MHz probes with 1X/10X attenuation selector, as shown below:

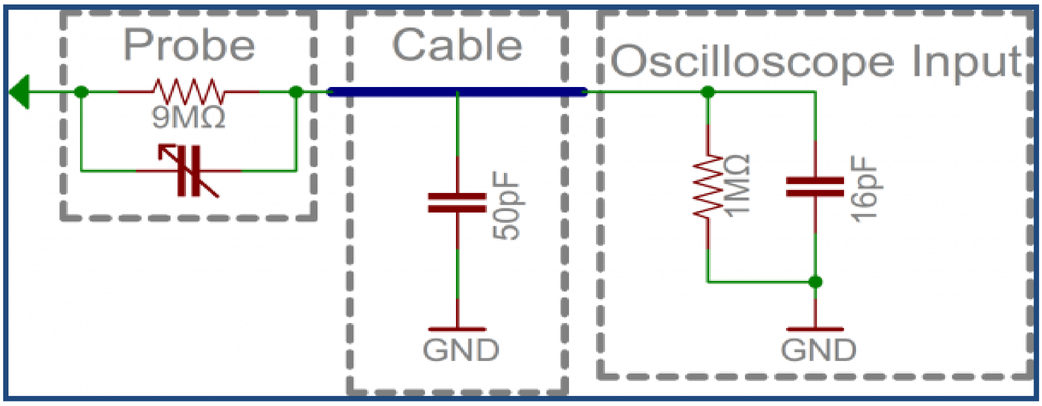


1. Attenuation Selector

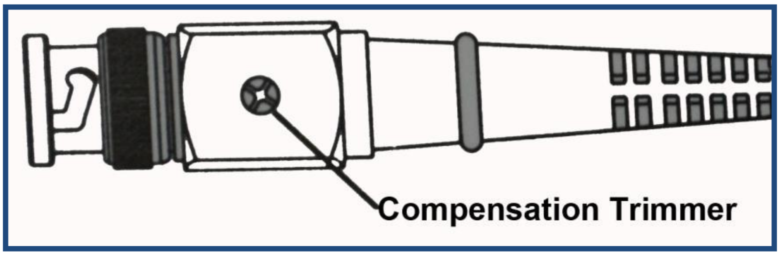
When the attenuation selector at the probe position is toggled to the 1X position as shown below, the input signal connected by the probe tip is sent to the oscilloscope without attenuation. Since the input range of the device is ±5V, then just a signal in this range can be measured. When the attenuation selector is moved to the 10X position, the input signal is sent to the device attenuated by a factor 10. In this case you can measure signals within the range of ±50V.



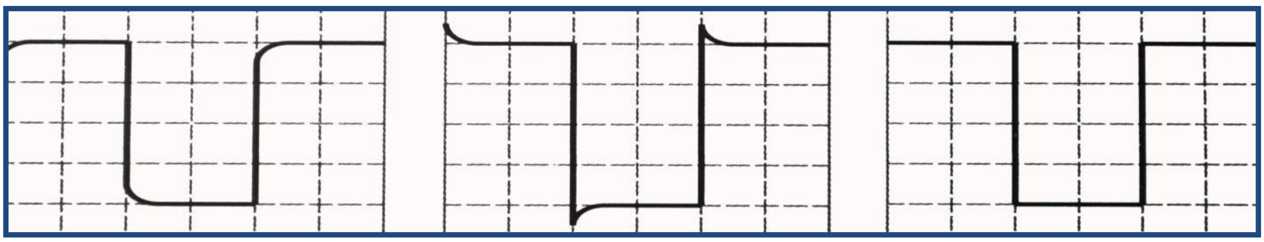
The 10X selector has better frequency characteristics and wider bandwidth, so you can use it when dealing with bandwidth and frequency limitations, for better measurements.



2. Compensation Trimmer

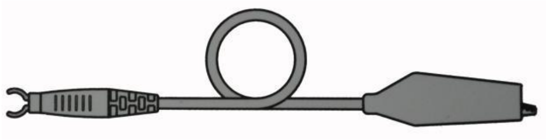


When using the 10X attenuator, the frequency characteristic of the probe can be corrected by adjusting the compensation capacitor on the probe. For this calibration set the probe to the 10X position and connect the probe to the reference signal generated by the device (square wave). Adjust the compensation trimmer until you can see on the screen a proper square waveform.



X1 position cannot be used for this probe calibration.

3. Probe Ground Clip

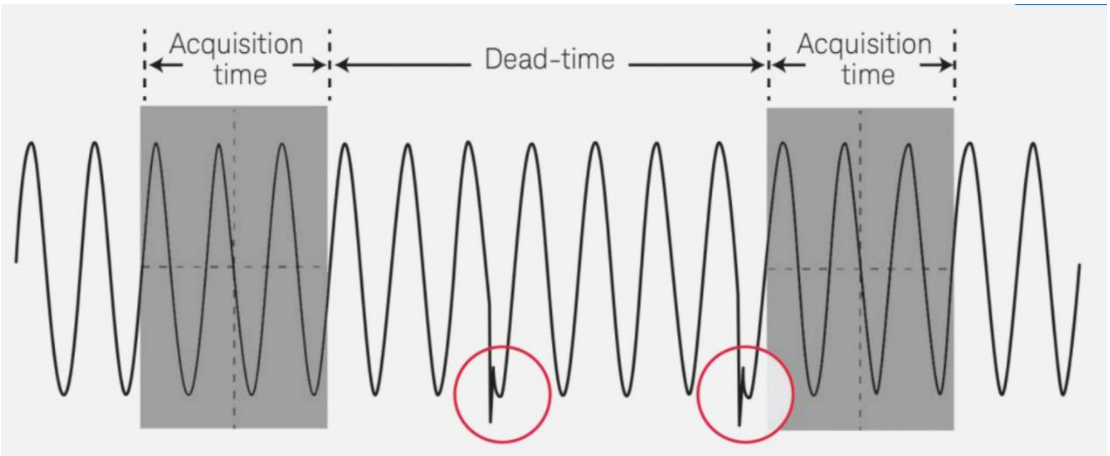


The probe ground clip is directly connected to the ground of the virtual oscilloscope circuit and connected to the PC ground via USB cable. When using a PC powered by the power outlet, the PC ground is connected to the ground through the three-core power jack. Make sure that the ground of the probe is connected to the ground of the circuit under test and to the ground used by the PC; otherwise you can have current leakage, short circuits or measurement errors. In severe circumstances, the device may also be damaged. See *Chapter 4* for details.

Appendix Ⅲ: oscilloscope variable length memory depth and segmented memory

Customers often consult and do not understand why the storage depth of the LOTO oscilloscope is longer, and they also express that they do not understand the segmented storage function of the LOTO oscilloscope. This article makes a complete review of the storage mechanism of the LOTO oscilloscope to help our customers better use an oscilloscope.

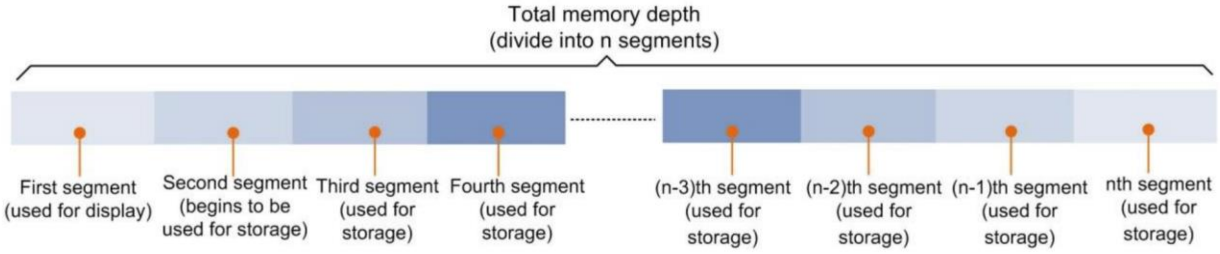
Digital oscilloscopes all have dead time (Dead-time), as shown in the figure below, between two acquisition time periods (Acquisition-time), there must be a dead time that has not been collected. The signal waveform inside is not collected and displayed



Therefore, oscilloscope manufacturers are trying their best to shorten the dead time period and lengthen the acquisition time period (gray part in the above figure). However, the dead time period cannot be reduced indefinitely, and the acquisition time period cannot be increased indefinitely, both of which need to be weighed in the actual use of the product.

Decreasing the sampling rate or increasing the storage depth are two common methods to lengthen the acquisition time period. A large memory depth can ensure that the oscilloscope can use the highest possible sampling rate to acquire waveforms for as long as possible at a time, but it cannot eliminate dead time periods. Larger memory will reduce the processing and waveform capture rate. This reduces the oscilloscope's response rate and increases the dead time between each acquisition. The method of reducing the sampling rate reduces the number of stored data points, which is beneficial to improve the capture rate and waveform refresh rate of the waveform.

High-end oscilloscopes also have a third method: the segmented memory function. The segmented storage feature divides storage into smaller segments. The user can specify that the storage should be divided into segments, each segment of equal length.



When the oscilloscope observes the first trigger event, it begins storing samples in the first segment of the acquisition memory until the first segment memory is full. The oscilloscope will then re-trigger and start looking for the next trigger event to occur.

When a trigger event occurs, the oscilloscope stores the sample point in the next memory segment. The oscilloscope repeats this process until all memory segments are full. The segmented memory mode is especially useful when capturing burst signals with long dead times. Many serial bus and communication signals fall into this category.

With segmented storage, the oscilloscope can maintain a high sample rate while capturing minutes, hours or days.

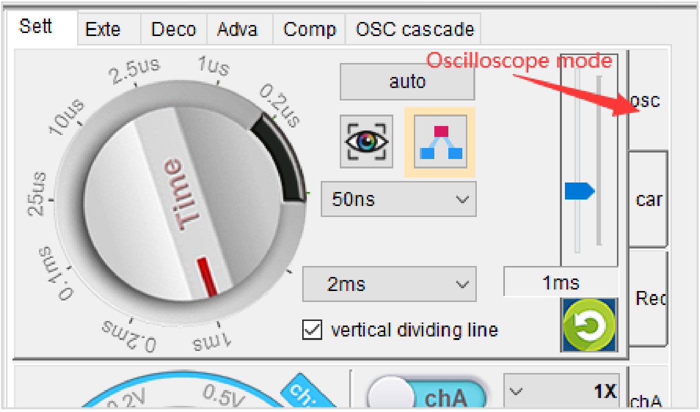
The LOTO oscilloscope provides a variety of flexible storage methods for OSCA02 and higher models, and has made a trade-off and consideration for the performance of the product. It can not only refresh quickly with a small storage depth, maintain a high waveform update rate, but also set a long time. The acquisition mode is continuous acquisition, and segmented storage is also possible.

● Memory depth of each frame (byte /Ch):	64K	≤50 ms/div
	258K	200 ms/div
	645K	500 ms/div
	2M	1 s/div
	8M	2 s/div
	10M	10 s/div
	20M	20 s/div
	30M	30 s/div
	40M	40 s/div
	50M	50 s/div
	60M	60 s/div
	250M	Stream mode
● PC buffer:	Up to 500 frames of PC online data waveform buffer.	

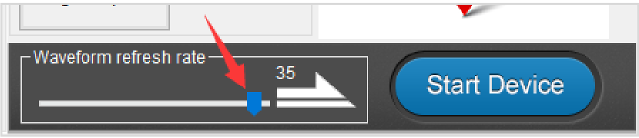
Let's introduce the functions one by one:

1 Oscilloscope mode:

In most application scenarios of most customers, the oscilloscope mode of the LOTO oscilloscope should be selected, which is also the default mode of our software. This mode preferentially uses a fixed storage depth of 128K to ensure fast waveform update, display and calculation.

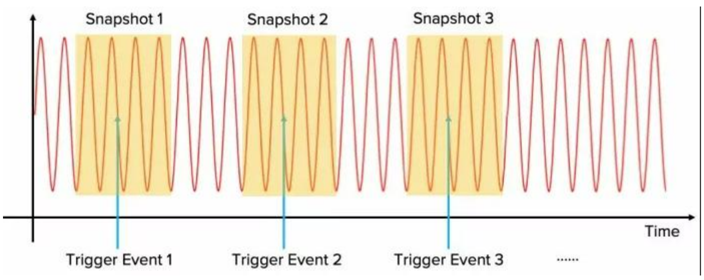


In this mode, customers can adjust the frame rate of waveform refresh by themselves. You can also set the trigger mode and position freely, and get fast waveform response.



In the oscilloscope mode, the customer does not need to consider the sampling rate and storage depth, but only needs to consider the time span of the current screen display, that is, the time scale. The oscilloscope will automatically adjust the sampling rate and storage depth according to the current time scale to achieve the most appropriate effect, taking into account both efficiency and accuracy. Therefore, the storage depth of the LOTO oscilloscope becomes longer.

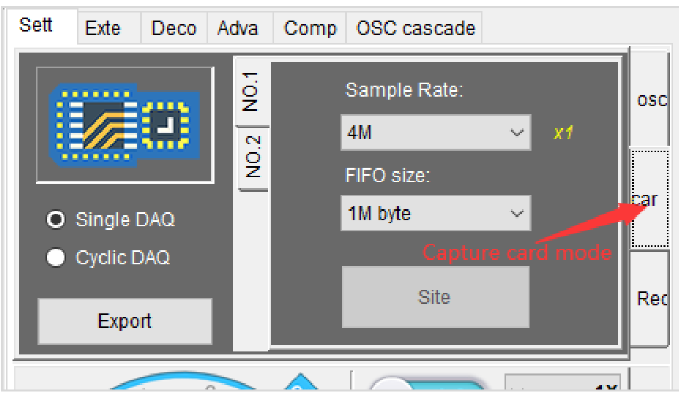
In this mode, the LOTO oscilloscope also provides a PC buffering function of 500 frames, and also provides a persistence function. These two functions record data and waveforms of multiple frames, and display them horizontally and vertically. The storage is equivalent to 500 current storage depths. For example, if the current storage depth is 128K, turning on the PC cache function is equivalent to a depth of 60M. And these data can be exported to a computer file, or imported into the software of the LOTO oscilloscope to reproduce the waveform for analysis, or exported to a text file or spreadsheet file frame by frame.



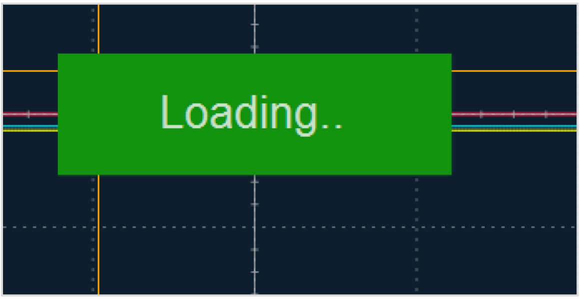
There is still a dead time between these 500 memory depths. When a high waveform refresh rate is used, this function can greatly improve the discovery probability of abnormal waveforms. This is a bit like the next mode segmented storage mode, which we will introduce later.

2 Capture card mode (traditional long storage mode):

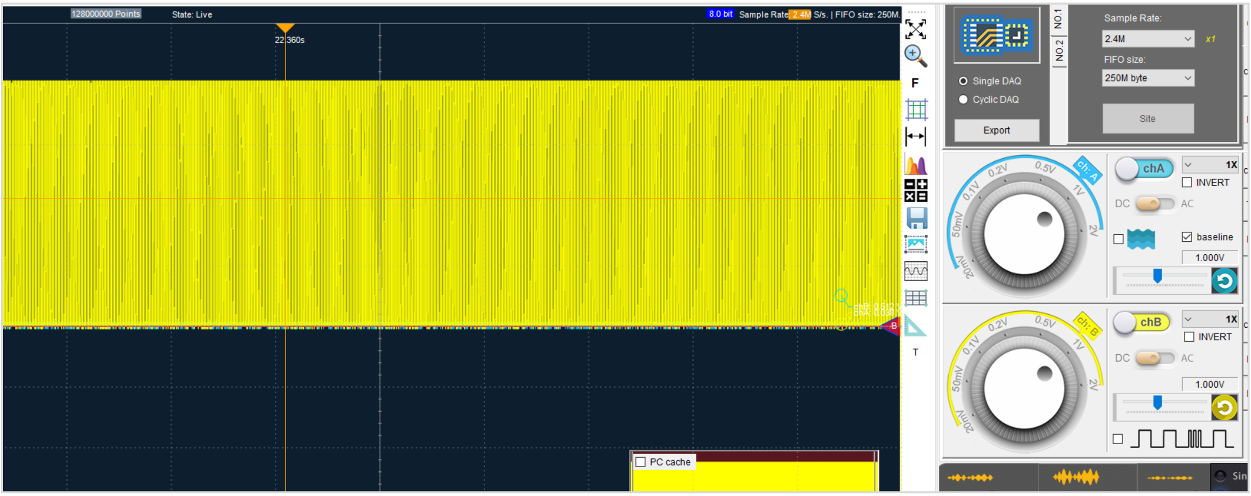
The LOTO oscilloscope can choose to switch the oscilloscope mode to the capture card mode, as shown in the following figure:



In this mode, the customer manually selects the sampling rate and selects a suitable buffer (equivalent to the storage depth) size for acquisition. After the selection is made, click on the acquisition, and the acquisition will continue for a long time until the entire selected buffer is filled. The buffer can be set to a maximum of 250 megabytes. If it is a dual-channel acquisition, the time required for the entire acquisition process is the buffer size divided by 2 and then divided by the sampling rate. For example, if the sampling rate is 2.4M, the buffer is 250M, and the sine wave of 1KHZ is collected, the waveform data will be collected continuously for 60 seconds, about 1 minute, and drawn on the screen. During the collection process, it will appear very stuck. Set a long time. If the waveform is not updated, there will be an in-progress indicator:

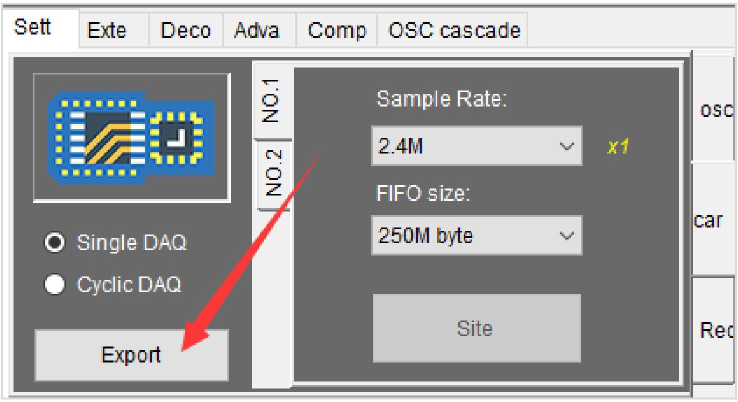


You need to wait patiently for the collection to end, as shown in the figure below.

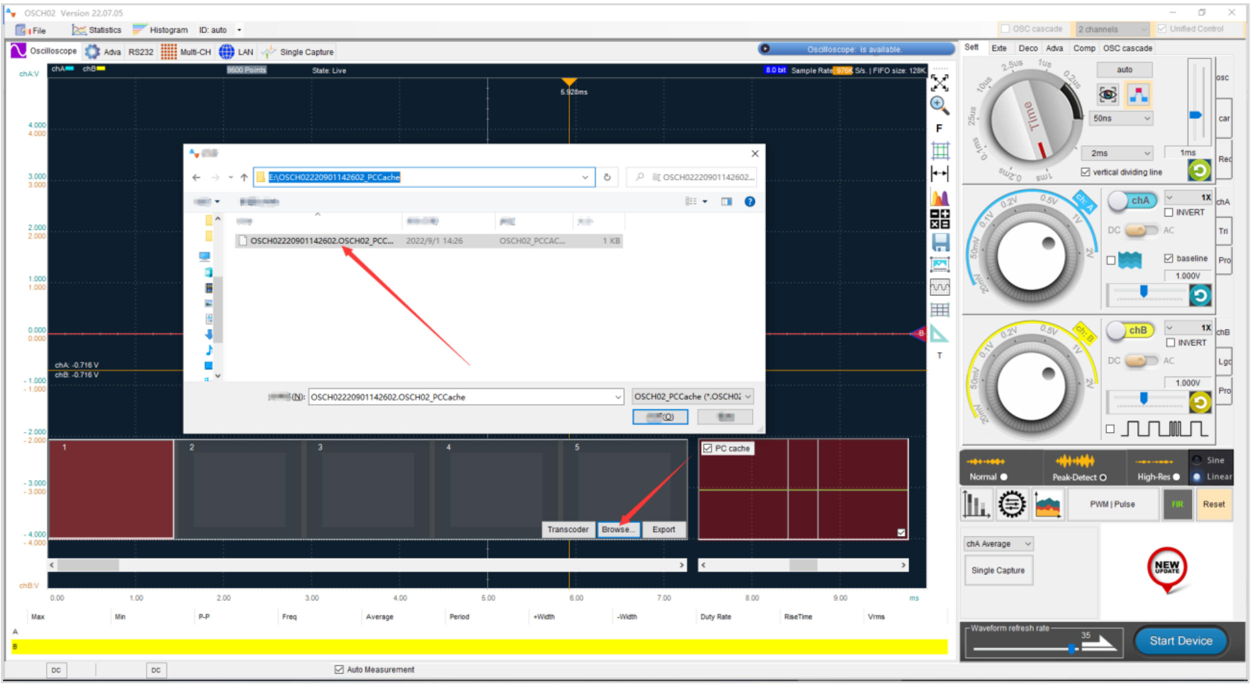


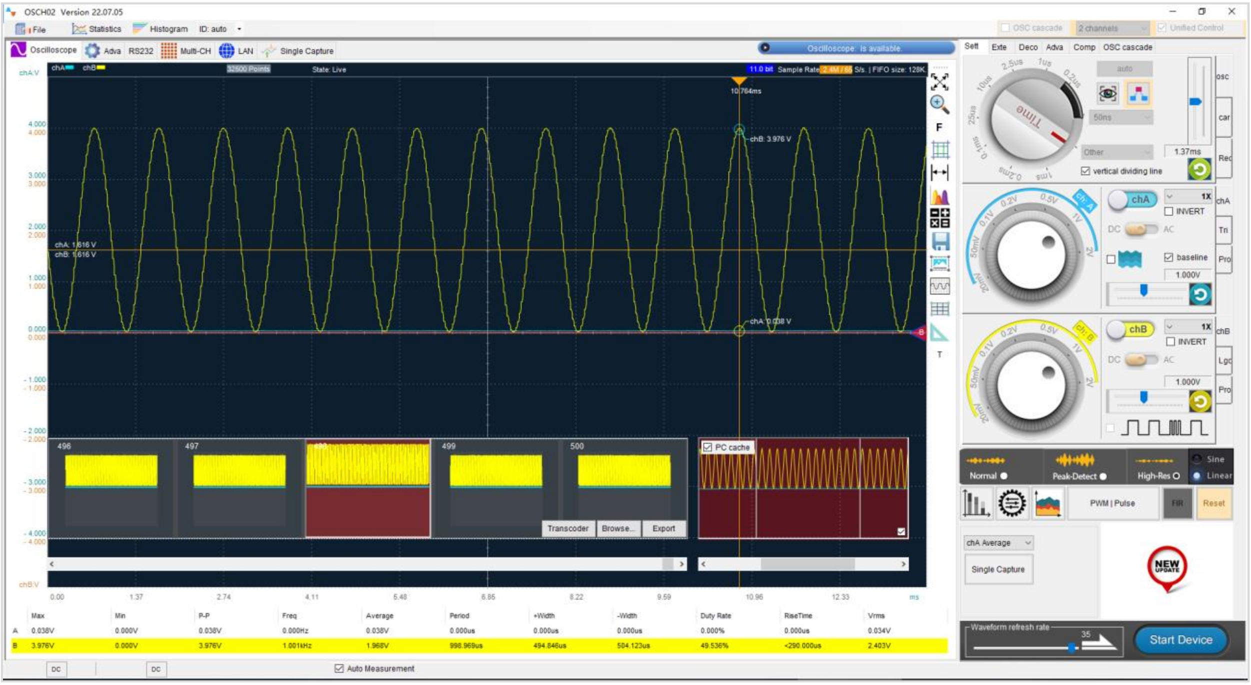
Such 1-minute data is placed on the screen, and in most cases, it will appear very dense. For example, there are more than 60,000 cycles of waveforms on the graph. We can zoom in and observe and analyze one by one. Generally, with the naked eye and the patience of engineers, people cannot see more than 1000 cycles will collapse, that is to say, the maximum waveform cycle you can view is 1000, which is a probability of 1 in 1,000. For an anomaly of 1ppm, you will definitely not be able to see it. You can now search by criteria, but what about criteria? Is it a pulse, an edge defect, or an overshoot? You can't predict, and you can't search. Searching according to various conditions takes a lot of time and mistakes. In some cases, use the oscilloscope mode of Mode 1 to quickly refresh the waveform. Once you see that the waveform flashes abnormally, you can immediately stop the acquisition and find it in the past 500 frame buffers.

Long storage will increase the computational processing requirements. With the same processor, the processing time will be longer if there is more data, and the waveform capture rate will become very slow. In order to facilitate the post-analysis of such long-term and large data collection, we can export the collected data into a data file, as shown in the following figure:



Then import it in 1 oscilloscope mode, and it becomes a segmented waveform of 500 frames. The total data volume is 250M bytes, which can be easily viewed and analyzed one by one, as shown in the following figure:

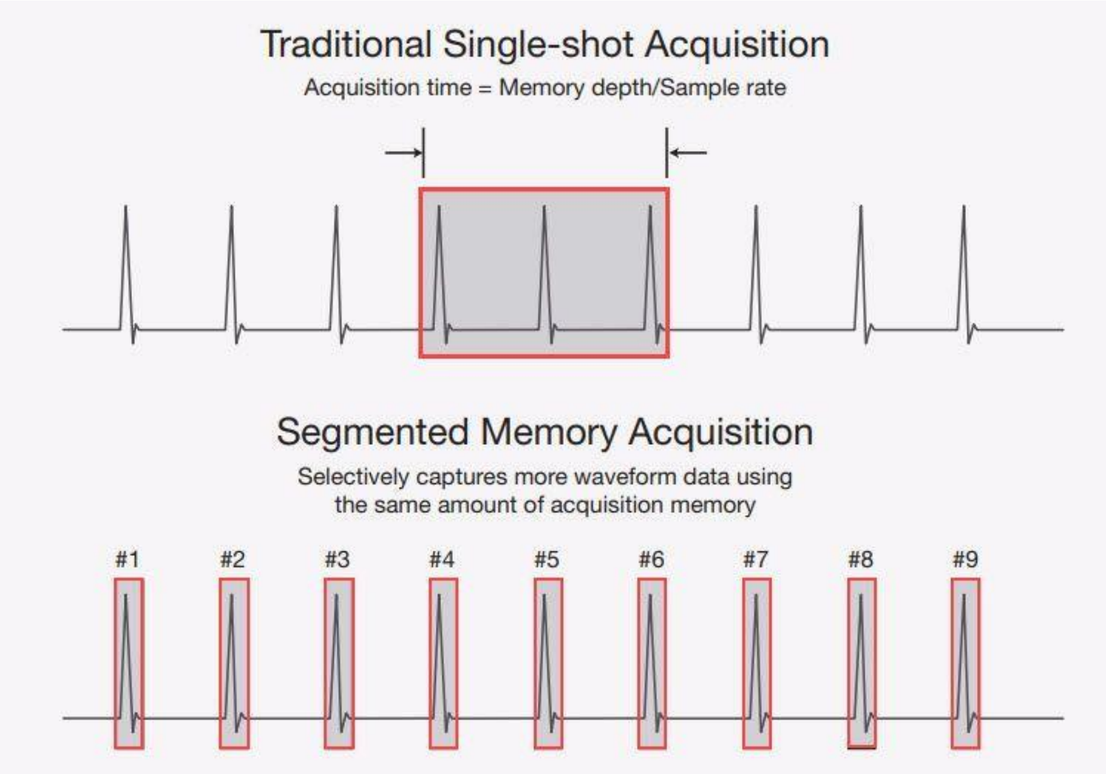




As shown in the figure above, the design of the LOTO oscilloscope takes into account the advantages of long-term acquisition, and tries to make up for the inconvenience of subsequent analysis and viewing. The LOTO oscilloscope can not only use the acquisition card mode in the case of 2-channel oscilloscope analog signal input, but also in the case of dual-channel combined acquisition, or in the case of logic analyzer digital channel input, and can export files. And import it into the PC cache for segmentation analysis. Once imported into the PC cache, the waveform can be reproduced for analysis, or exported frame by frame as a text file or spreadsheet.

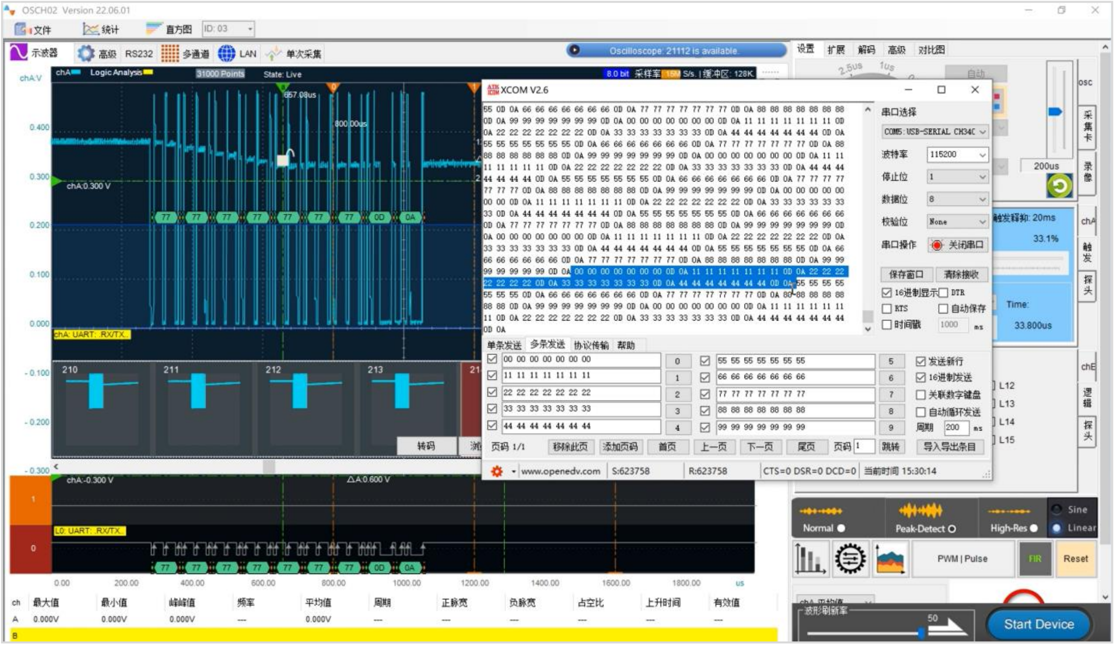
3-segment storage mode:

As mentioned earlier, in the oscilloscope mode, turning on the PC cache function is basically equivalent to having a segmented storage capacity of 500 segments. This segmented memory mode has a particularly typical application scenario: low duty cycle pulse or burst signals. There is a long idle time between signals. In many cases, it is difficult to achieve the desired acquisition time even with a large storage or by reducing the sampling rate. Imagine that it may occur 100 times in a day. , every time the signal that appears is very sharp and short, it needs a high sampling rate to capture, no matter how large the oscilloscope has a large storage, it is impossible to store a day's data, and no matter how low the sampling rate is, it is impossible to capture all 100 signals. The segmented storage can be done very well.

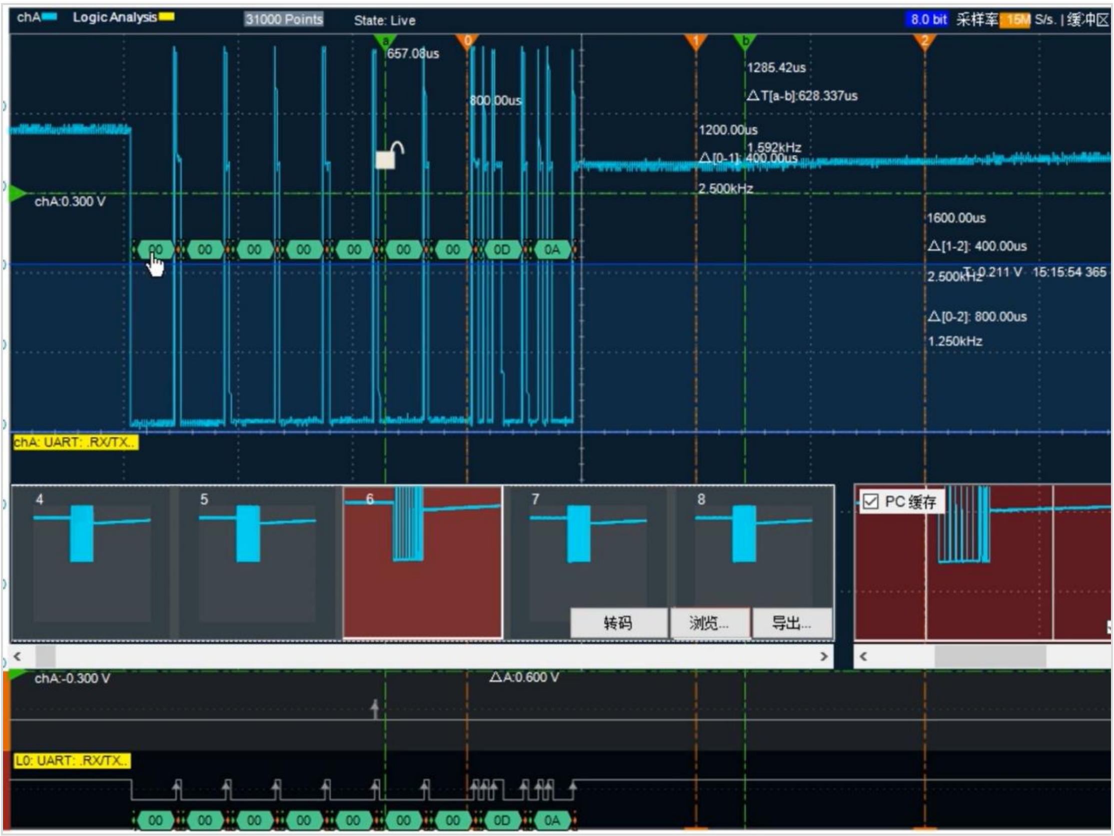


As shown in the figure above, the segmented storage is triggered multiple times during the high sampling rate acquisition process, and the data sampled for each trigger is stored in a small storage that divides the storage space into segments. The oscilloscope trigger fills one segment at a time, and the idle signals or parts of the signal that are not of interest between segments are not acquired and stored.

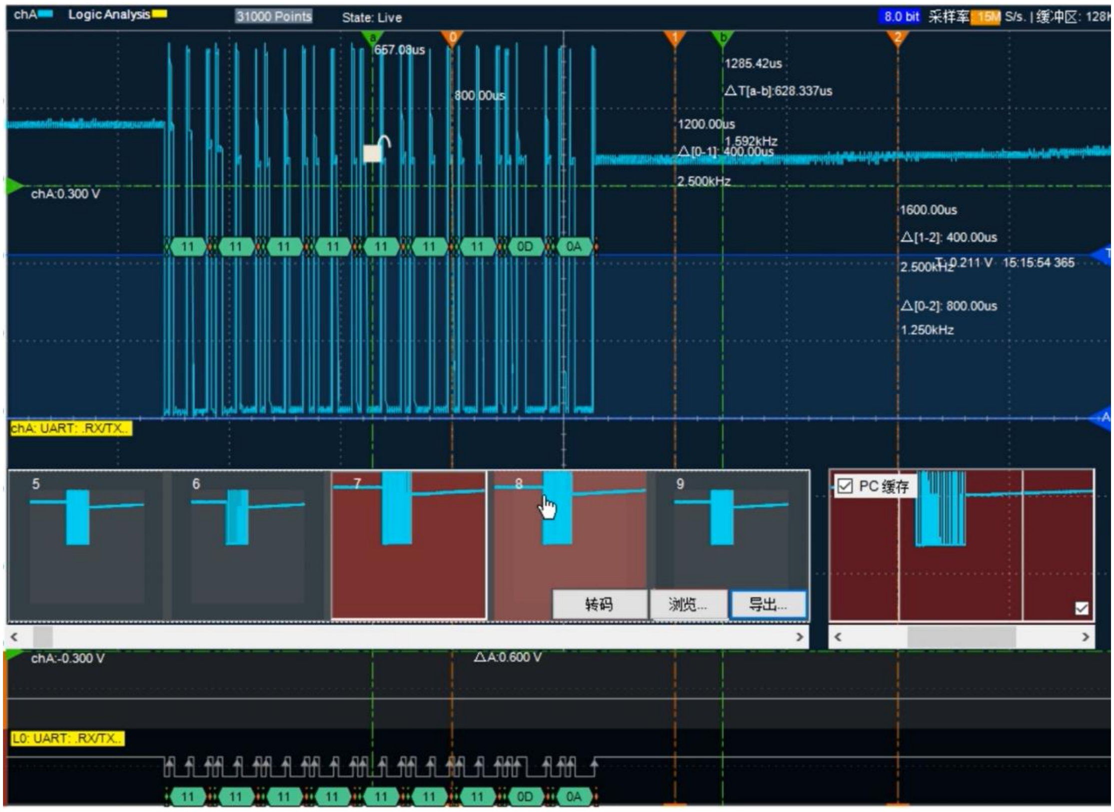
Another common scenario that is particularly suitable for segmented storage is serial bus analysis – the serial bus is transmitted in packets, and the idle time between packets will occupy the precious storage resources of the oscilloscope. With segmented storage, the oscilloscope can only collect data packets without sampling during idle time. While maintaining a high sampling rate, more data packets can be collected to facilitate decoding and analysis. Next, let's look at an example of a LOTO oscilloscope using segmented memory to acquire and decode such a serial bus.



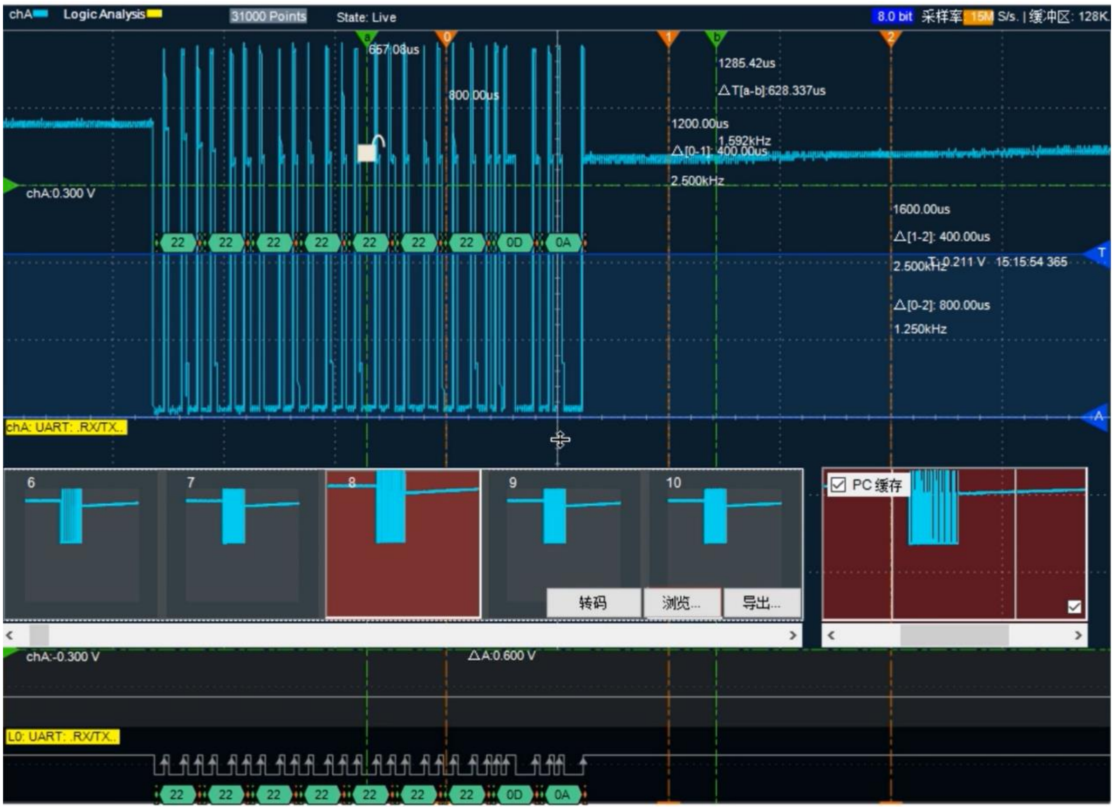
As shown in the figure above, the serial port RS232 will send a series of numbers every 200ms (if you want the effect to be more obvious, you can set a longer interval, such as 1 minute interval), 0000000 all the way to 999999999 and sent cyclically. We started triggering and 500 frames of PC cache function as segment storage.



We see that the 6th frame of the segmented storage as shown in the figure above, the data is 0000000, and it is decoded. In the 7th frame of the picture below, the data is 1111111, and it is decoded.



The 8th frame of the segmented storage in the following figure, the data is 2222222, and it is decoded...



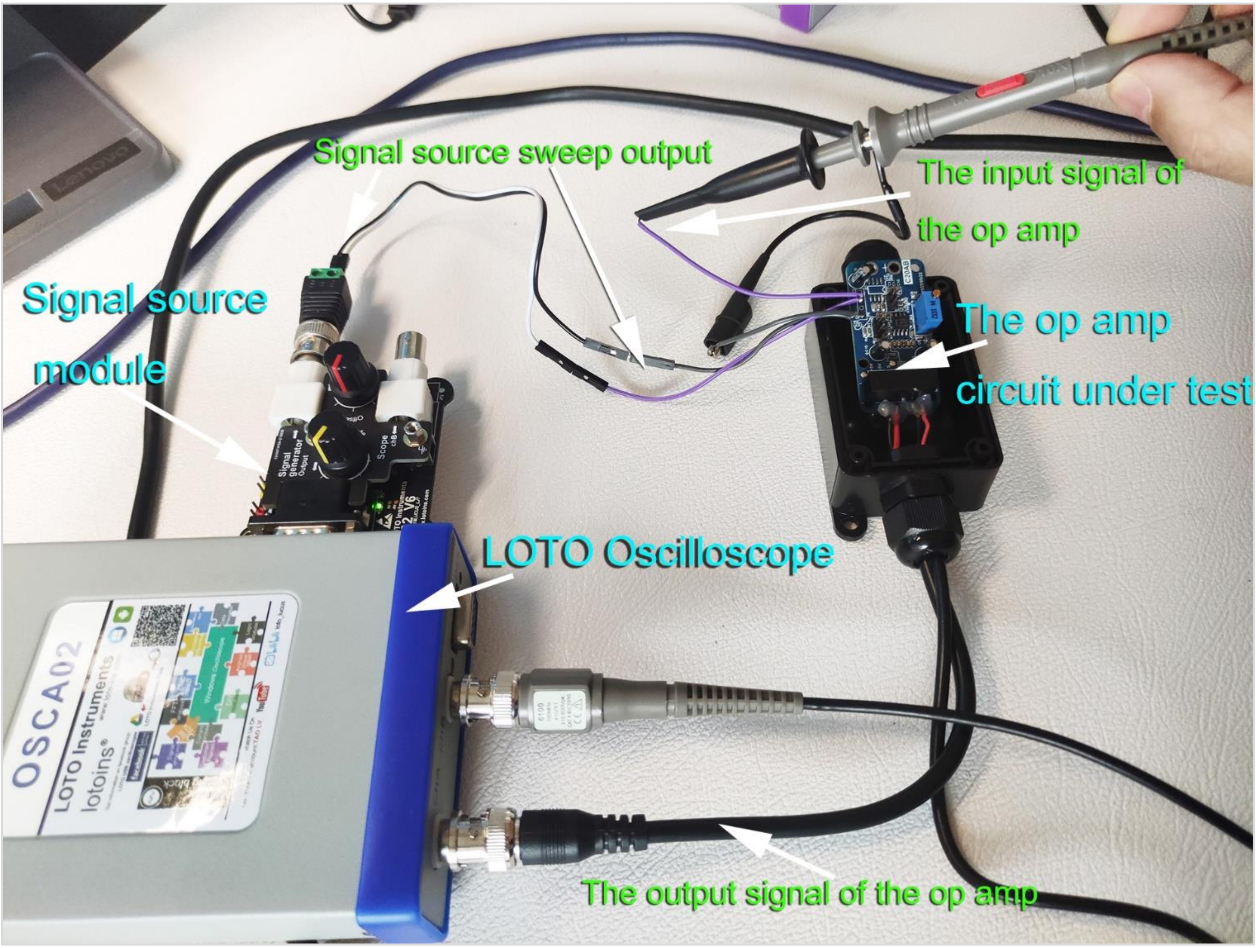
In this way, until the end, each frame is captured by a data packet and decoded, and 500 frames can be captured without omission, no matter how long the entire sending process is.

Appendix Ⅳ: oscilloscope detection frequency response characteristic curve

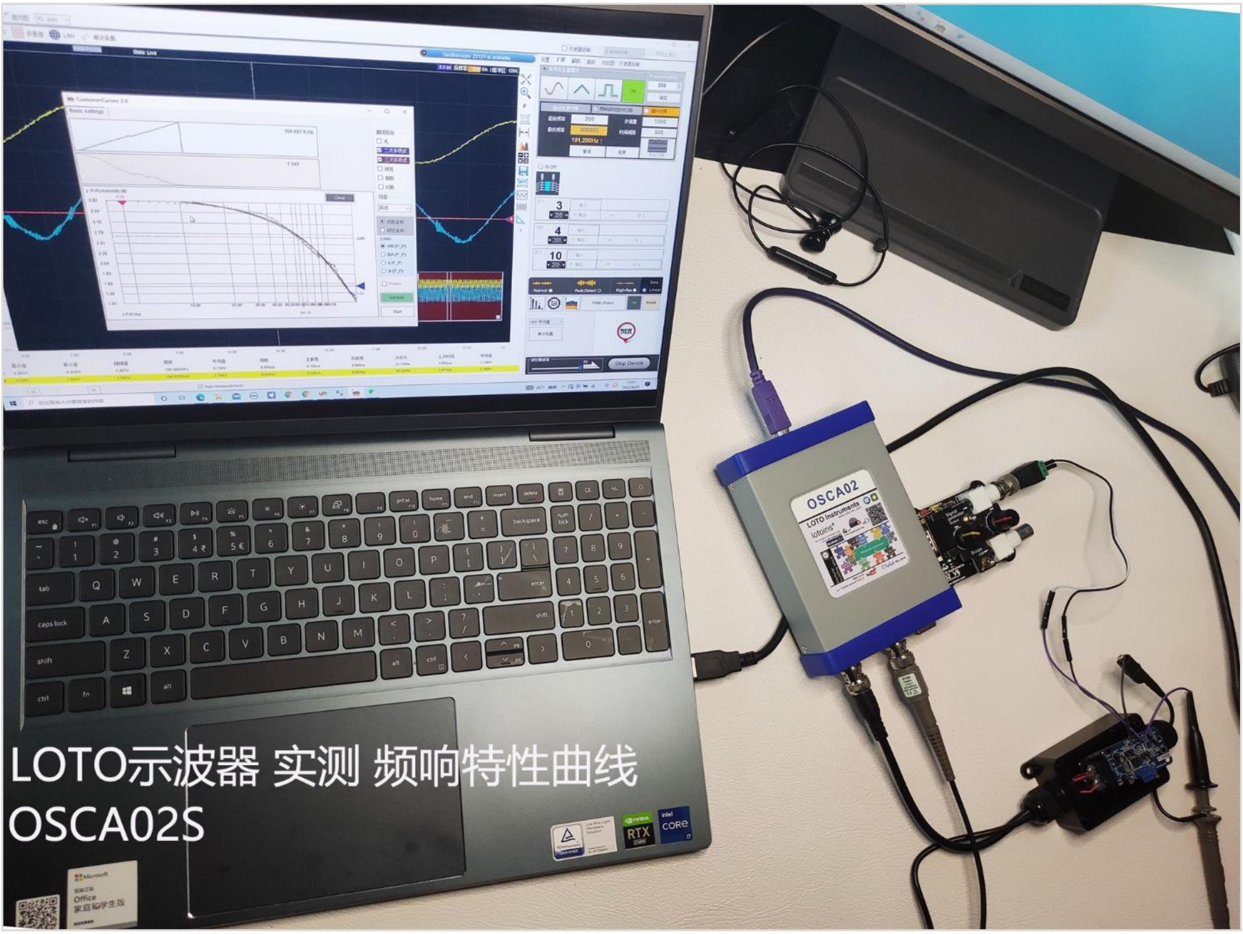
In work and projects, it is often encountered that a functional circuit module performs signal conditioning, or filtering, or amplification, or attenuation, or impedance transformation. These functional circuit modules may be passive resistance-capacitance, active op-amp circuits, or more complex systems. But the most important characteristic of their signal conditioning is the frequency response characteristic. Most of the time, we need to know its frequency response curve for system design and verification.

The host computer software of the LOTO oscilloscope has the functions of sweep frequency and frequency response characteristic curve mapping. If you purchase a model that combines oscilloscope + signal source module, such as OSCA02S, you can map the frequency response characteristic curve of the circuit under test without additional products.

The following figure shows the wiring for a typical frequency response characteristic curve test:

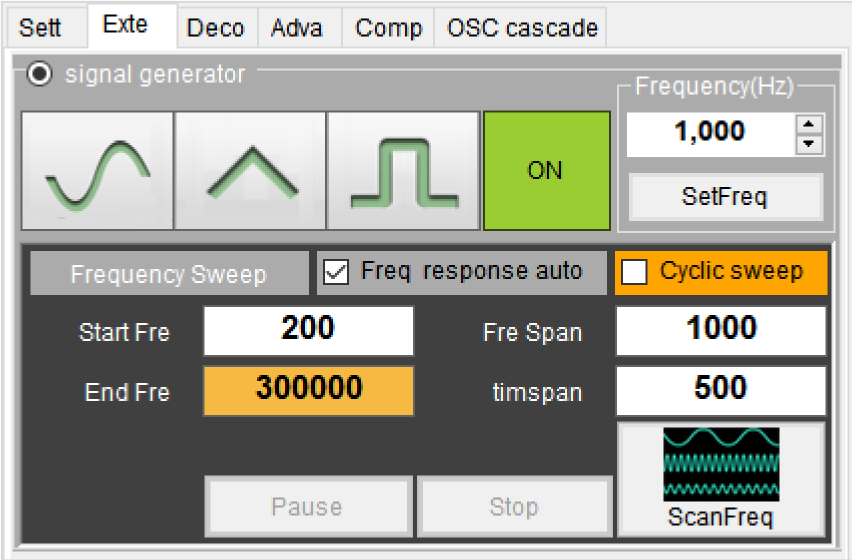


In the figure above, we have prepared an op amp module as the circuit under test. We connect the output of the signal source module to the input of the op amp, and the software will control the sweep frequency to output a sine wave signal to excite the op amp circuit. This input is connected in parallel with an oscilloscope probe, and the signal is input to channel B of the oscilloscope, so that we can see the sweep output of the signal source on the oscilloscope, that is, the input signal waveform of the circuit under test. The operational amplifier circuit under test amplifies the input signal and outputs it, and we directly connect the output signal to channel A of the oscilloscope.

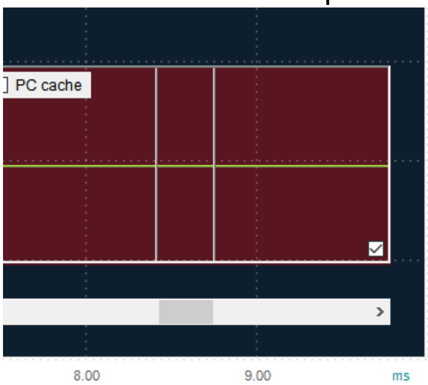


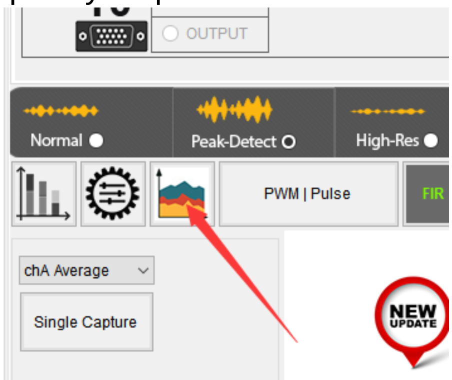
This way the hardware and wiring are ready. If you don't have a suitable circuit under test in hand to get familiar with the function, you can also manually adjust the amplitude attenuation of the signal source during the frequency sweep to simulate. We can measure the curve of the amplitude of the output signal of the op amp as a function of frequency, and we can also measure the curve of the magnification of the op amp as the frequency of the signal changes.

On the software, we first need to set the parameters of the frequency sweep, as shown in the figure below, output sine wave, the start frequency of the frequency sweep, the final frequency, the step size and the time interval of the step, etc. It is especially important to note that, We need to select "Automatic sweep of frequency response curve", this option will help us automatically set the time scale of the oscilloscope during the frequency sweep process, so that we do not need to manually adjust the time scale continuously, so that the waveform is not too dense or too sparse .

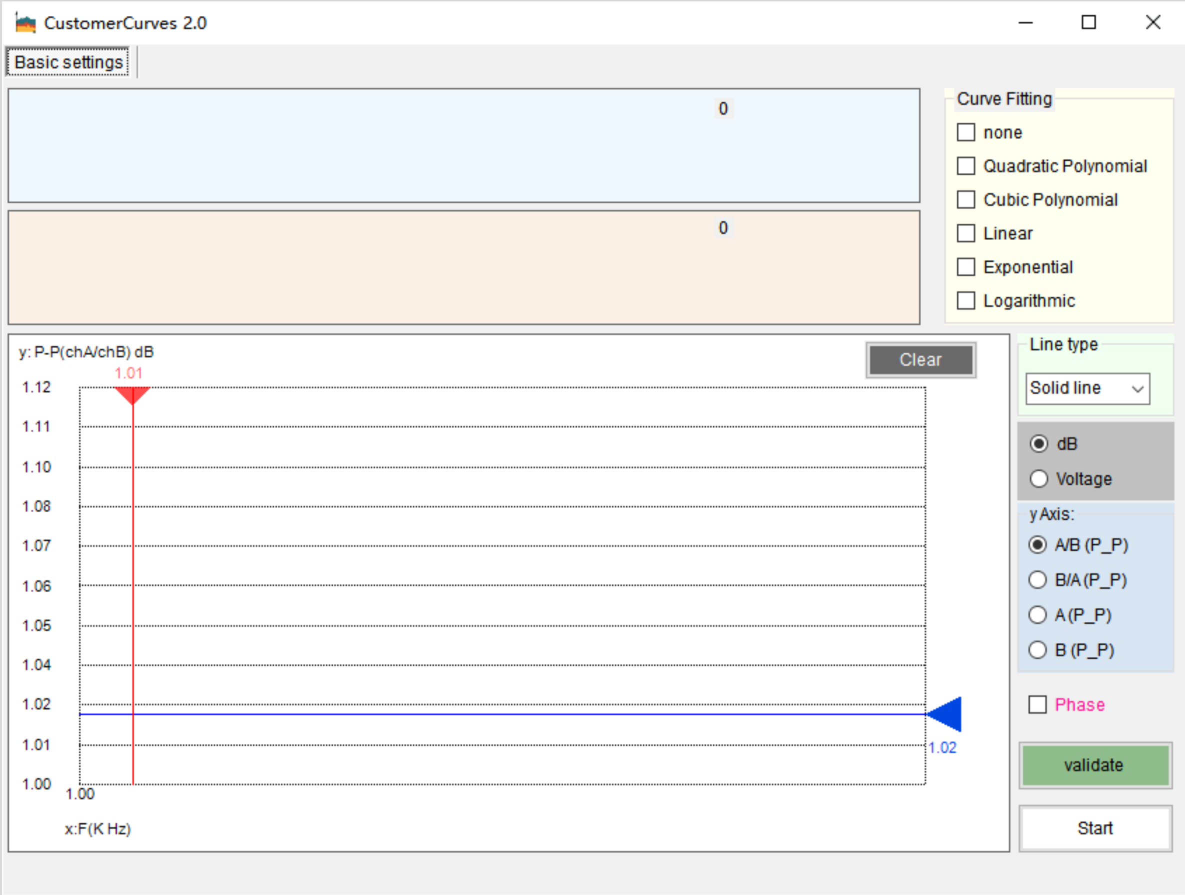


Then, we click this button to open the frequency response curve function interface:

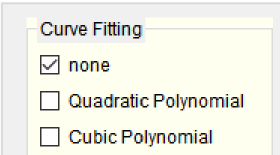




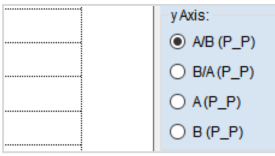
We will open the control panel and graphical interface of the frequency response characteristic curve:



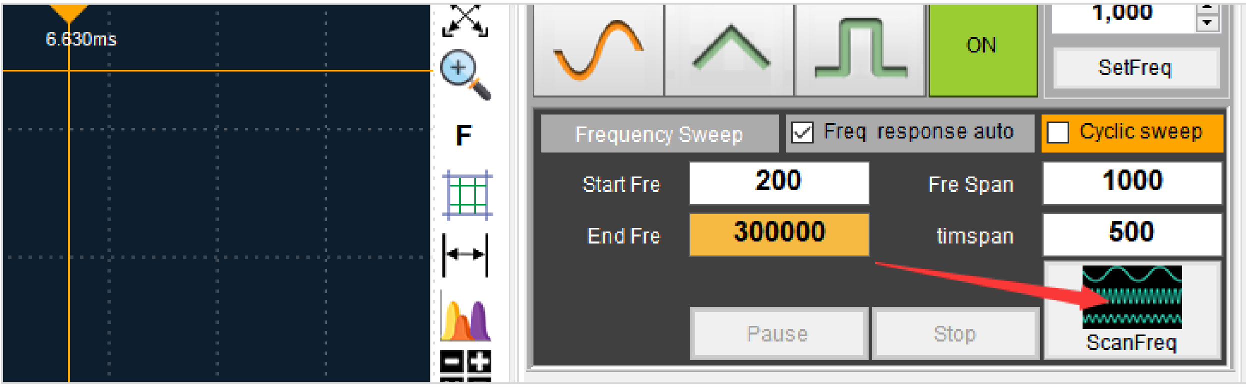
In the curve fitting setting area, we need to select "None", that is, no curve fitting. We use the real measurement points to represent the entire frequency response curve. After completion, we can do curve fitting.



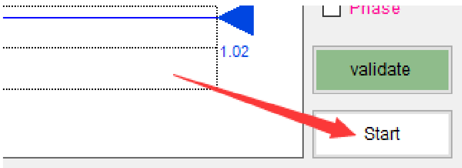
For example, what we want to measure is the response curve of the output amplitude of the op amp with the frequency change, then we can choose the amplitude of the channel corresponding to the oscilloscope to which the output signal of the op amp is connected as the ordinate of the frequency response curve:



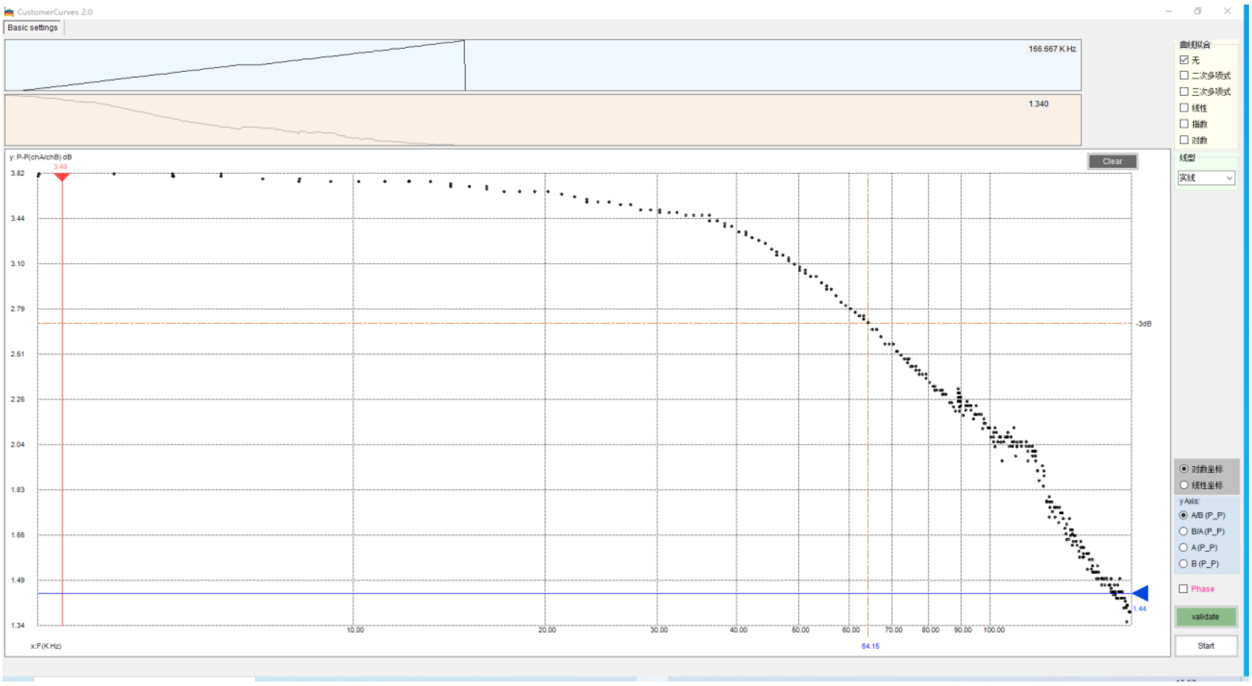
After setting, we switch back to the interface of oscilloscope and signal source, and click the sweep button to start sweeping:



The frequency sweep will be performed according to the set parameters, but there will be a switching time of one or two seconds at the beginning. The signal output during this time may not be stable, so we click the start of the frequency response curve interface after the waveform is stable. button to start the drawing of the frequency response curve, as shown in the following figure:

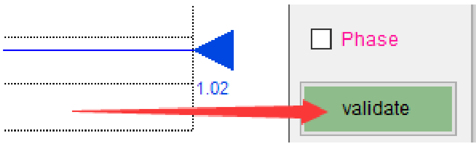


Next, we don't have to do anything, just observe the drawing of the frequency response curve and wait for the end.

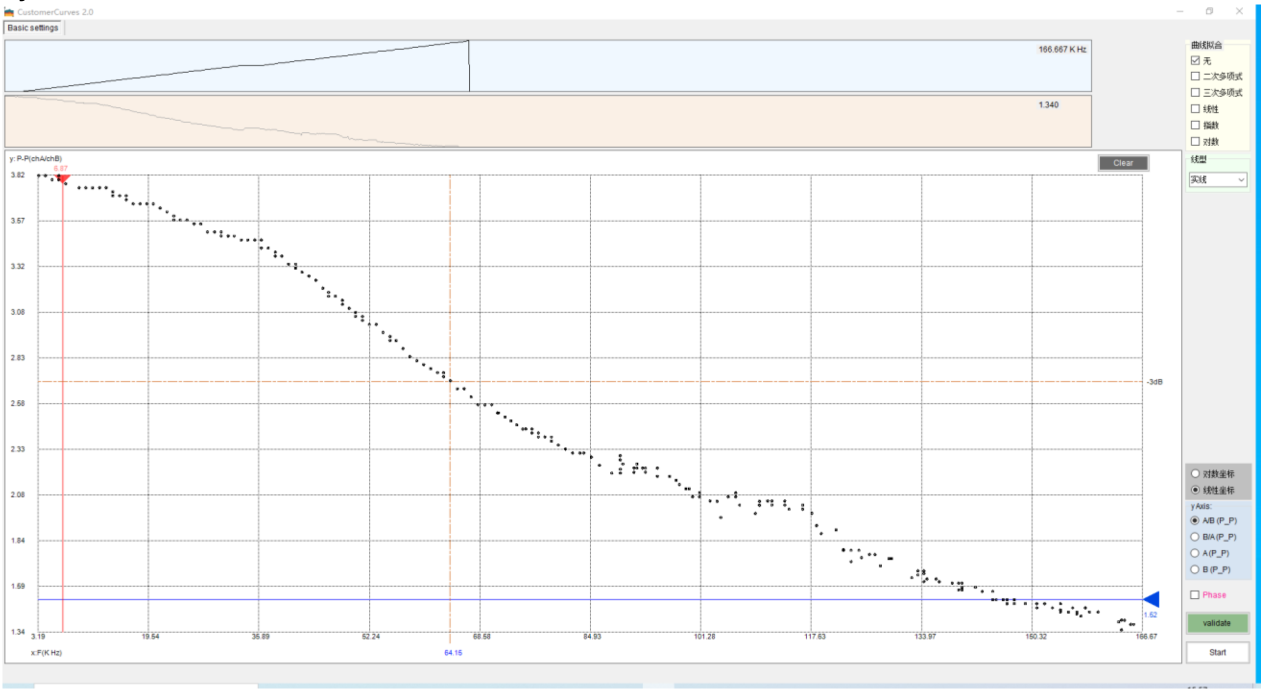


After we sweep to the end, or we observe that we have obtained enough curves, we can click the pause button in the lower right corner to end the drawing. At this time, even if the oscilloscope and the signal source software continue to sweep the frequency, the interface of the frequency response curve will not continue to be updated.

At this time, we can click the "validation" button. The function of this button is to normalize and detect these data points in the frequency sweep process, and remove some illegal data points caused by interference or misoperation, so as to better perform curve fitting. ,As shown below:

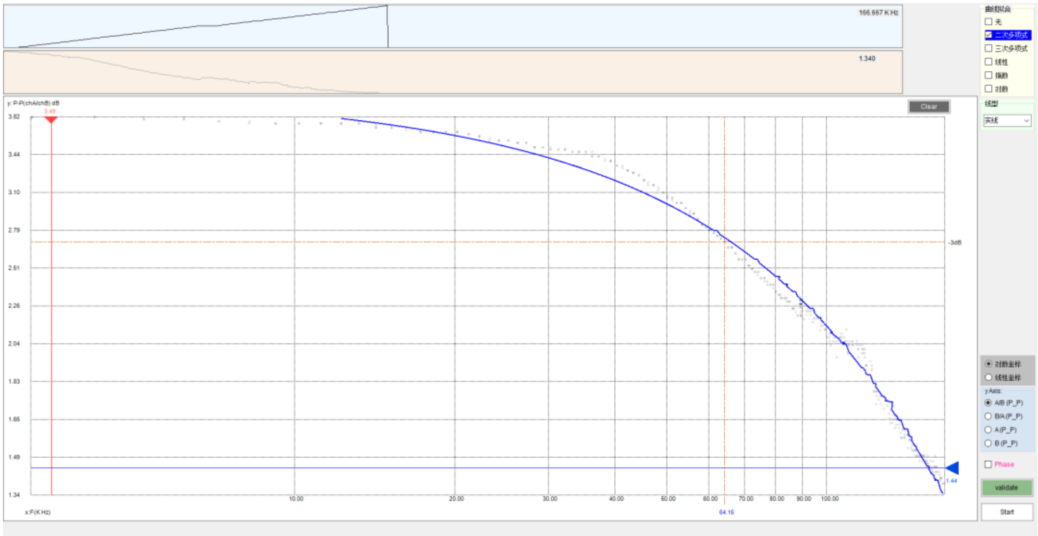


Above we saw the frequency response characteristic curve of the logarithmic coordinate obtained by sweeping the frequency, because when we sweep the frequency, the logarithmic coordinate option is used by default. We can also select the linear coordinate option to display the frequency response curve of the linear coordinate system:

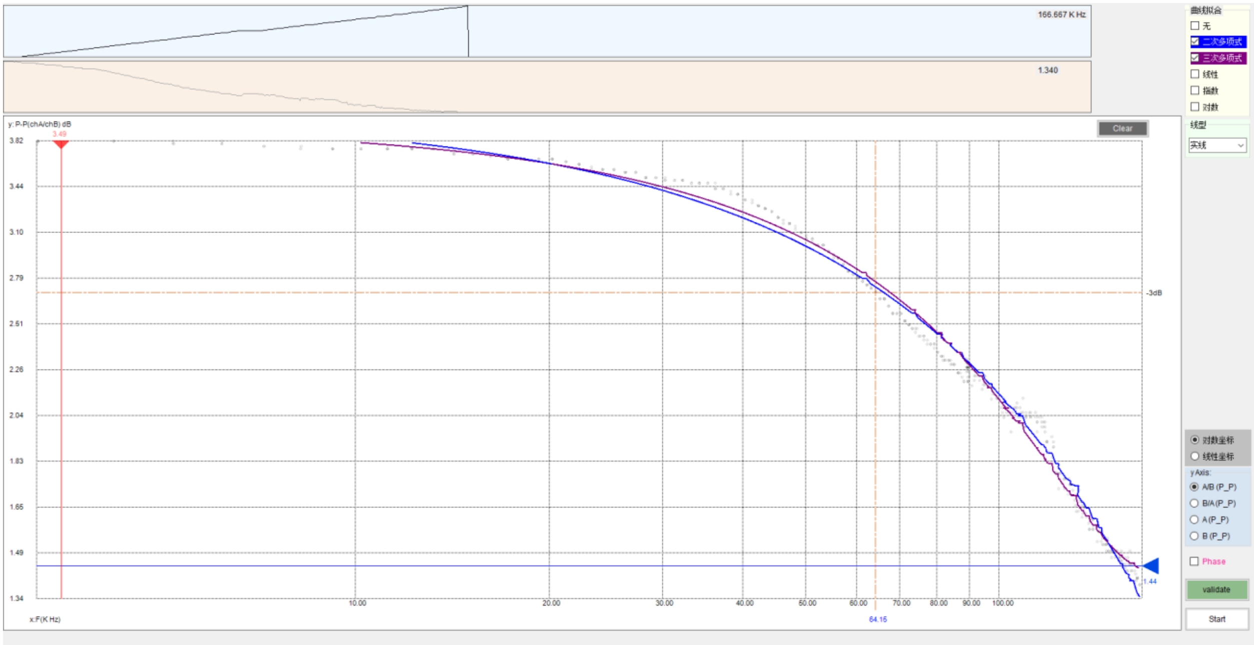


We can see that the frequency response curve has automatically identified the position of -3DB and the corresponding cutoff frequency. The cutoff frequency is around 64K Hz. We can choose from a variety of curve fitting methods: linear, quadratic polynomial, cubic polynomial, exponential fit, logarithmic fit.

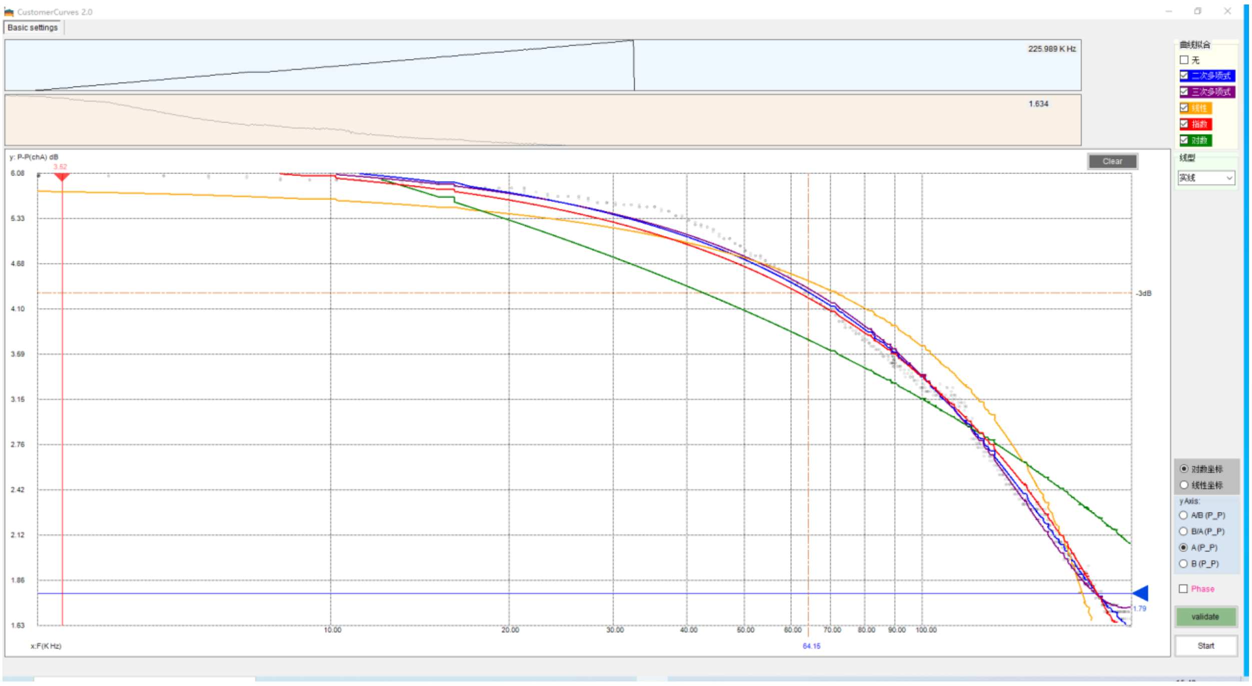
Taking the logarithmic coordinate system as an example, the quadratic polynomial fitting:

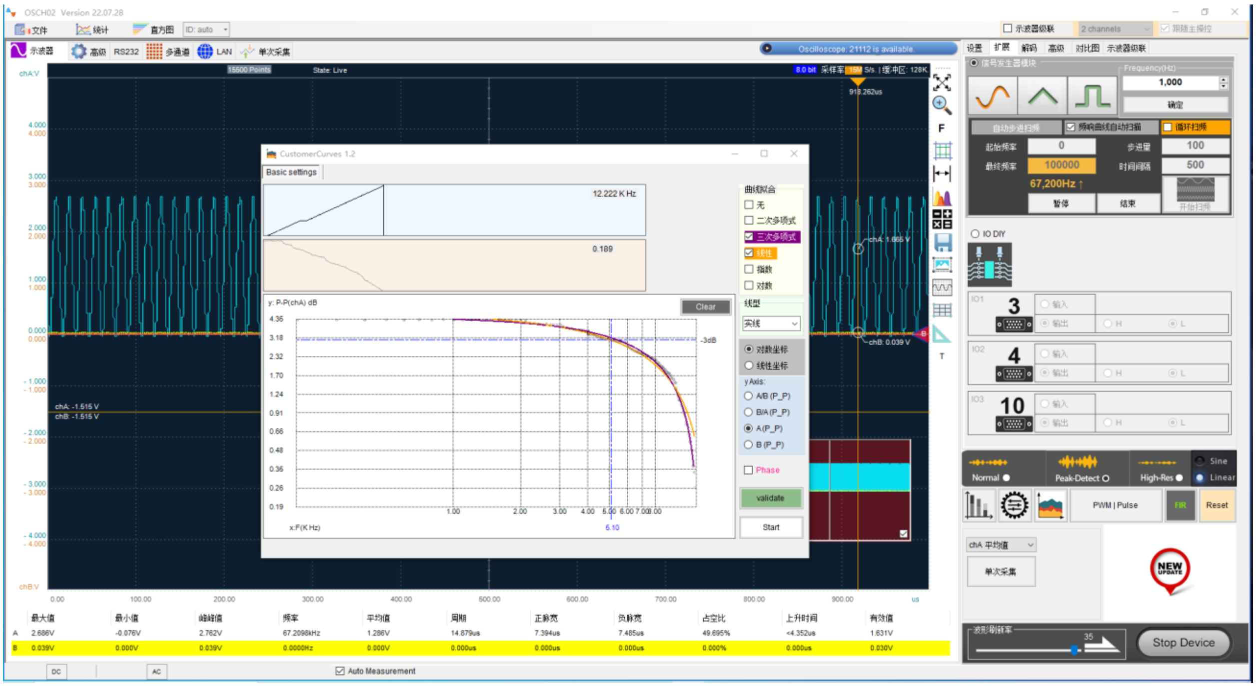


Taking the logarithmic coordinate system as an example, quadratic and cubic polynomial fitting:



Taking the logarithmic coordinate system as an example, all fitting options are enabled:





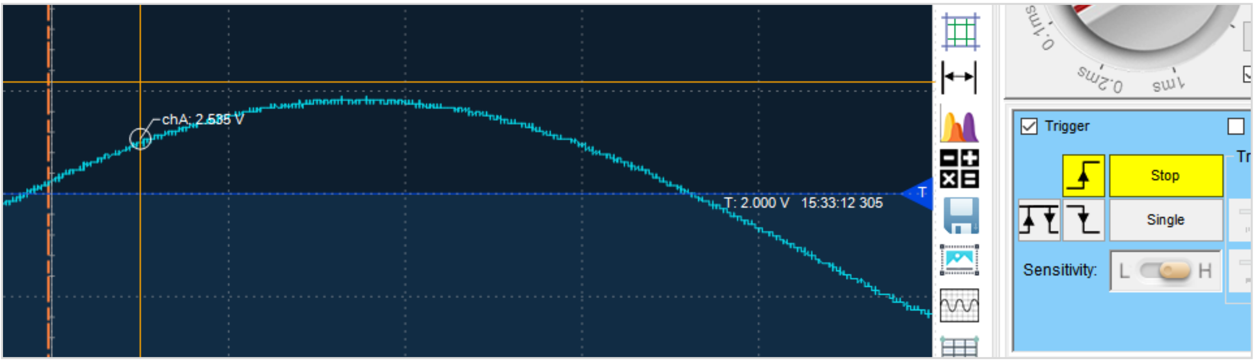
Taking the linear coordinate system as an example, all fitting options are enabled:

We have made a video record and demonstrated the whole process of the above process, you can refer to the following video link:

https://www.youtube.com/watch?v=frLPk0uoD_c&t=10s

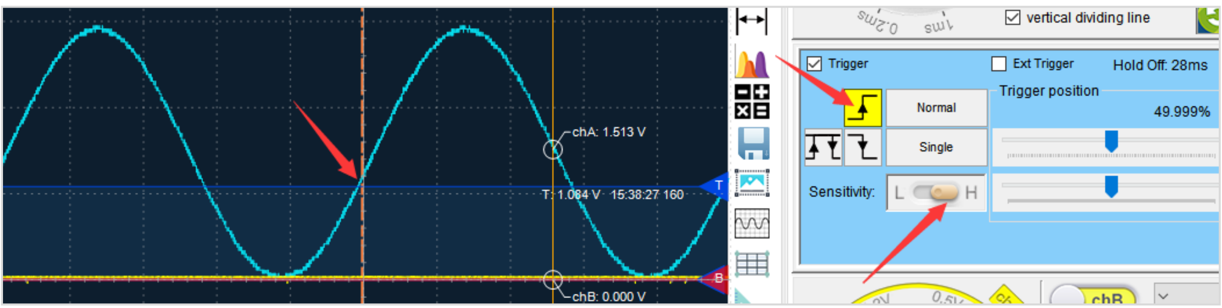
Appendix Ⅴ**: oscilloscope Trigger Sensitivity**

The trigger level is just a reference voltage, and the actual waveform has jitter at the edge. As shown in the figure below, it is a measured screenshot of a normal rising edge trigger:

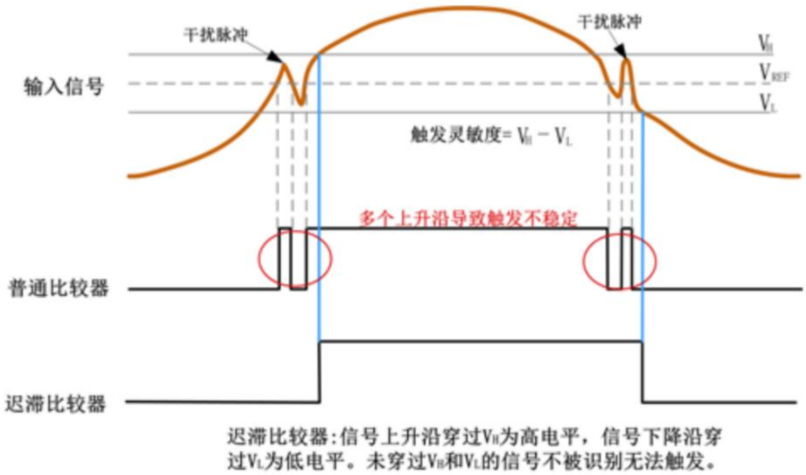


The blue T line is the reference voltage of the trigger level. The interference of the waveform in the figure is very small, but we zoom in and then zoom in on the edge of the signal. As can be seen in the figure below, there is still a sawtooth shape. When the noise is large, the jitter will be more severe. .

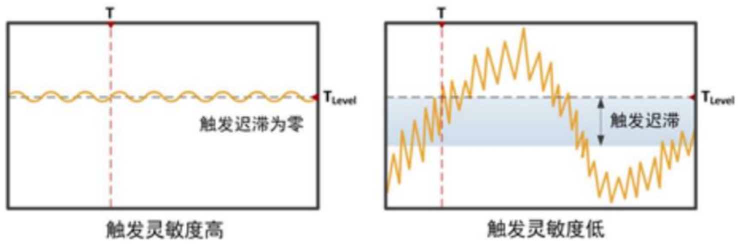
If a very sensitive trigger function is used, these glitches and noise disturbances near the trigger level can cause triggering. The phenomenon shown is that the waveform keeps shaking and shaking, and even the rising edge becomes the falling edge, and the falling edge becomes the rising edge trigger. As shown in the figure below, the rising edge trigger will occasionally trigger to the falling edge of the waveform, because there are glitches in the above figure near the trigger level, and the setting with high sensitivity is selected, then a certain value will be set at the trigger level. The rising glitch is considered to meet the trigger condition, thus triggering a falling edge.



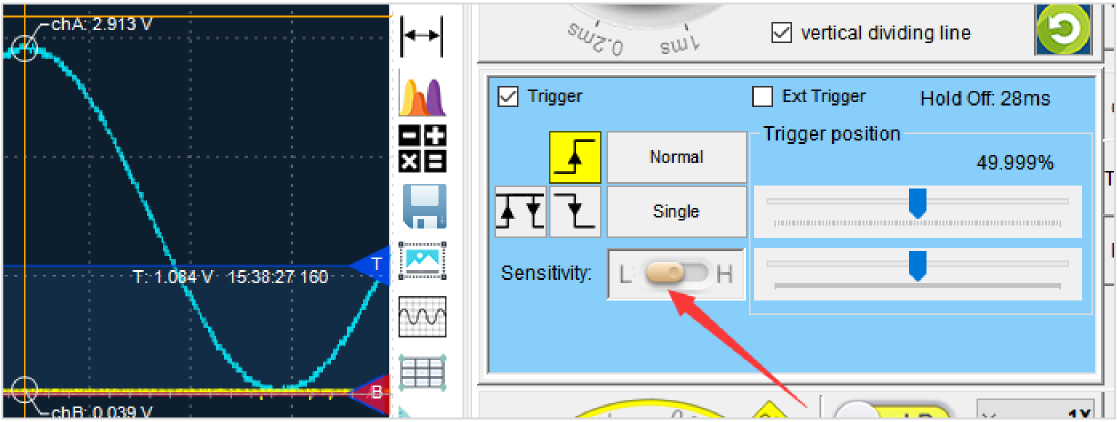
If you want to stabilize the rising edge of the trigger waveform, you need to use a hysteresis comparison in the upper and lower range of the trigger level to filter waveform jitter and glitches near the trigger level. This hysteresis range is the trigger sensitivity.



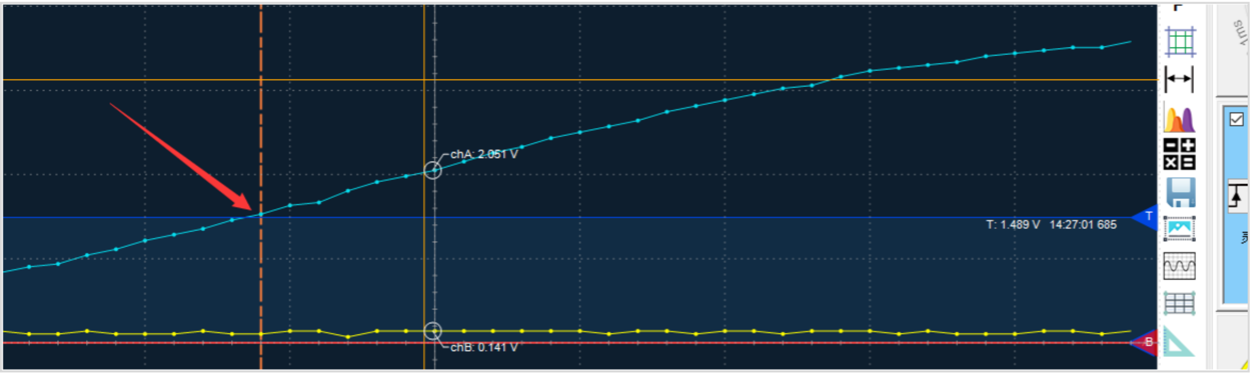
When measuring small signals, or waveforms without significant noise interference, a high trigger sensitivity is required to stabilize the signal and trigger at the correct location. When the waveform noise is large, it needs to be adjusted to a lower trigger sensitivity, which can effectively filter out the noise that may be superimposed on the trigger signal, thereby preventing false triggering, as shown in the figure below.



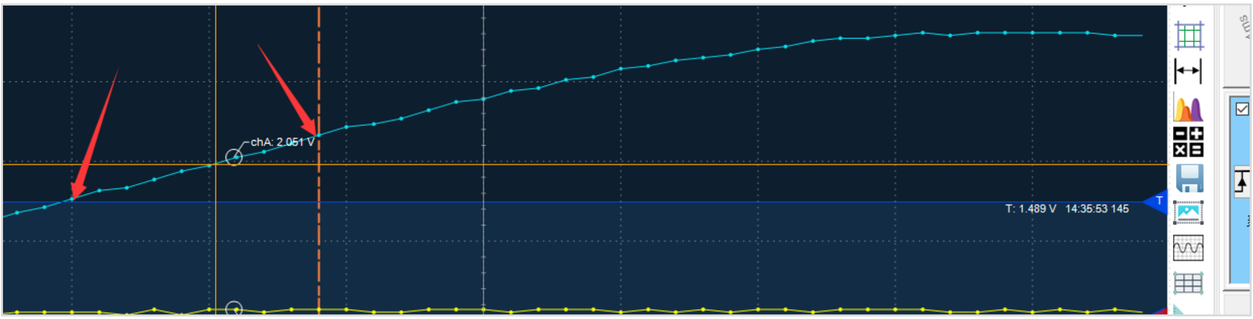
LOTO oscilloscope adds the function of adjusting trigger sensitivity to some mainstream models of USB virtual oscilloscopes. This function requires hardware support, so the oscilloscope equipment before the release of the new hardware does not have this function. Due to limited hardware resources, the newly added trigger sensitivity has only two options, high sensitivity and low sensitivity. We can find it in the trigger setting area of the host computer software:



When the number of waveform points is dense or the noise is obvious, we can choose low sensitivity (L). For the case where the noise is small or the number of sampling points is small, if the low sensitivity is used, the hysteresis algorithm is used to avoid the influence of noise, which will lead to The trigger position is imprecise, so select High Sensitivity (H). For example the following situation:



In the above picture, due to the high frequency of the waveform, the sampling points are relatively sparse. At this time, a high sensitivity trigger must be used to be stuck at the trigger position at the appropriate sampling point as in the above picture. Otherwise, due to the algorithm, a little A hysteretic comparison algorithm that removes the effects of noise glitches will misplace many sampling points. When the sampling points are so sparse, the displacement of more than 10 sampling points will obviously deviate from the trigger position visually, and in this case, there is no burr and noise to be removed. As shown in the figure below, it is a screenshot of the low sensitivity trigger:



It is offset by about 9 samples from the high-sensitivity trigger setting.

We have made a video record and demonstrated the whole process of the above process; you can refer to the following video link:

https://youtu.be/c6xDgHnn8Yc