UPO7000L Series Digital Phosphor Oscilloscopes

10GSa/s | 2GHz | 1Gpts | 2,000,000wfms/s

User Manual REV.2.1

November 2024

Table of Contents

Foreword

Thank you for choosing this UNI-T instrument. For safe and proper use this instrument, please read this manual carefully, especially the safety instructions section.

After reading this manual, it is recommended to keep the manual in a convenient location, preferably near the device, for future reference.

Safety Instructions

This chapter contains information and warnings that must be observed. Ensure that the instrument is operated under the safe conditions. In addition to the safety precautions indicated in this chapter, you must also follow accepted safety procedures.

<table>
<tr><th colspan="3">Safety Precautions</th></tr>
<tr><td rowspan="2">Warning</td><td colspan="2">Please follow these guidelines to avoid possible electric shock and risk to personal safety.</td></tr>
<tr><td colspan="2">Users must adhere to standard safety precautions during the operation, servicing, and maintenance of this device. UNI-T will not be liable for any personal safety and property loss caused by the user's failure following the safety precautions. This device is designed for professional users and responsible organizations for measurement purposes.
Do not use this device in any manner not specified by the manufacturer. This device is intended for indoor use only, unless otherwise stated in the product manual.</td></tr>
<tr><th colspan="3">Safety Statements</th></tr>
<tr><td>Warning</td><td colspan="2">“Warning" indicates the presence of a hazard. It warns users to pay attention to a certain operation process, operation method or similar. Personal injury or death may occur if the rules in the “Warning" statement are not properly executed or observed. Do not proceed to the next step until you fully understand and meet the conditions stated in the “Warning" statement.</td></tr>
<tr><td>Caution</td><td colspan="2">“Caution" indicates the presence of a hazard. It warns users to pay attention to a certain operation process, operation method or similar. Product damage or loss of important data may occur if the rules in the “Caution" statement are not properly executed or observed. Do not proceed to the next step until you fully understand and meet the conditions stated in the “Caution" statement.</td></tr>
<tr><td>Note</td><td colspan="2">“Note" indicates important information. It reminds users to pay attention to procedures, methods, and conditions, etc. The contents of “Note" should be highlighted if necessary.</td></tr>
<tr><th colspan="3">Safety Signs</th></tr>
<tr><td></td><td>Danger</td><td>It indicates danger of electric shock, which may cause personal injury or death.</td></tr>
<tr><td></td><td>Warning</td><td>It indicates that there are factors you should be cautious of to prevent personal injury or product damage.</td></tr>
</table>

⚠	**Caution**	It indicates danger, which may cause damage to this device or other equipment if you fail to follow a certain procedure or condition. If the "Caution" sign is present, all conditions must be met before you proceed to operation.
⚠	**Note**	It indicates potential problems, which may cause failure of this device if you fail to follow a certain procedure or condition. If the "Note" sign is present, all conditions must be met before this device will function properly.
∿	**AC**	Alternating current of device. Please check the region's voltage range.
⎓	**DC**	Direct current device. Please check the region's voltage range.
⊥	**Grounding**	Frame and chassis grounding terminal
⏚	**Grounding**	Protective grounding terminal
⏚	**Grounding**	Measurement grounding terminal
○	**OFF**	Main power off
|	**ON**	Main power on
⏻	**Power**	Standby power supply: When the power switch is turned off, this device is not completely disconnected from the AC power supply.
CAT I		Secondary electrical circuit connected to wall sockets through transformers or similar equipment, such as electronic instruments and electronic equipment; electronic equipment with protective measures, and any high-voltage and low-voltage circuits, such as the copier in the office.
CAT II		Primary electrical circuit of the electrical equipment connected to the indoor socket via the power cord, such as mobile tools, home appliances, etc. Household appliances, portable tools (e.g., electric drill), household sockets, sockets more than 10 meters away from CAT III circuit or sockets more than 20 meters away from CAT IV circuit.
CAT III		Primary circuit of large equipment directly connected to the distribution board and circuit between the distribution board and the socket (three-phase distributor circuit includes a single commercial lighting circuit). Fixed equipment, such as multi-phase motor and multi-phase fuse box; lighting equipment and lines inside large buildings; machine tools and power distribution boards at industrial sites (workshops).
CAT IV		Three-phase public power unit and outdoor power supply line equipment. Equipment designed to "initial connection," such as power distribution system of power station, power instrument, front-end overload protection, and any outdoor transmission line.
CE	**Certification**	CE indicates a registered trademark of EU.
UK CA	**Certification**	UKCA indicates a registered trademark of United Kingdom.

	Certification	Conforms to UL STD 61010-1 and 61010-2-030. Certified to CSA STD C22.2 No.61010-1 and 61010-2-030.
	Waste	Do not place equipment and accessories in the trash. Items must be properly disposed of in accordance with local regulations.
	environmental protection	This environment-friendly use period (EFUP) mark indicates that dangerous or toxic substances will not leak or cause damage within this indicated time period. The environmentally friendly use period of this product is 40 years, during which it can be used safely. Upon expiration of this period, it should enter the recycling system.

Safety Requirements

Warning

Preparation before use	Please connect this device to AC power supply with the power cable provided. The AC input voltage of the line reaches the rated value of this device. See the product manual for specific rated value. The line voltage switch of this device matches the line voltage. The line voltage of the line fuse of this device is correct.
Check all terminal rated values	Please check all rated values and marking instructions on the product to avoid fire and the impact of excessive current. Please consult the product manual for detailed rated values before connection.
Use the power cord properly	You can only use the special power cord for the instrument approved by the local and state standards. Please check whether the insulation layer of the cord is damaged, or the cord is exposed, and test whether the cord is conductive. If the cord is damaged, please replace it before using the instrument.
Instrument grounding	To avoid electric shock, the grounding conductor must be connected to the ground. This product is grounded through the grounding conductor of the power supply. Please be sure to ground this product before it is powered on.
AC power supply	Please use the AC power supply specified for this device. Please use the power cord approved by your country and confirm that the insulation layer is not damaged.
Electrostatic prevention	This device may be damaged by static electricity, so it should be tested in the anti-static area if possible. Before the power cable is connected to this device, the internal and external conductors should be grounded briefly to release static electricity. The protection grade of this device is 4kV for contact discharge and 8kV for air discharge.
Measurement accessories	Measurement accessories designated as lower-grade, which are not applicable to main power supply measurement, CAT II, CAT III, or CAT IV circuit measurement.
Use the input / output port of this device properly	Please use the input / output ports provided by this device in a proper manner. Do not load any input signal at the output port of this device. Do not load any signal that does not reach the rated value at the input port of this device. The probe or other connection accessories should be effectively grounded to avoid product damage or abnormal function. Please refer to the product manual for the rated value of the input / output port of this device.

Power fuse	Please use a power fuse of exact specification. If the fuse needs to be replaced, it must be replaced with another one that meets the specified specifications by the maintenance personnel authorized by UNI-T.
Disassembly and cleaning	There are no components available for operators inside. Do not remove the protective cover. Qualified personnel must conduct maintenance.
Service environment	This device should be used indoors in a clean and dry environment with ambient temperature from 0 °C to +40 °C. Do not use this device in explosive, dusty, or high humidity conditions.
Do not operate in humid environment	Do not use this device in a humid environment to avoid the risk of internal short circuit or electric shock.
Do not operate in flammable and explosive environment	Do not use this device in a flammable and explosive environment to avoid product damage or personal injury.
Caution	
Abnormality	If this device may be faulty, please contact the authorized maintenance personnel of UNI-T for testing. Any maintenance, adjustment or parts replacement must be done by the relevant personnel of UNI-T.
Cooling	Do not block the ventilation holes at the side and back of this device. Do not allow any external objects to enter this device via ventilation holes. Please ensure adequate ventilation and leave a gap of at least 15cm on both sides, front and back of this device.
Safe transportation	Please transport this device safely to prevent it from sliding, which may damage the buttons, knobs, or interfaces on the instrument panel.
Proper ventilation	Insufficient ventilation will cause the device temperature to rise, thus causing damage to this device. Please keep proper ventilation during use, and regularly check the vents and fans.
Keep clean and dry	Please take actions to avoid dust or moisture in the air affecting the performance of this device. Please keep the product surface clean and dry.
Note	
Calibration	The recommended calibration period is one year. Calibration should only be conducted by qualified personnel.

Environmental Requirements

This instrument is suitable for the following environment:

- Indoor use
- Pollution degree: Class 2
- Operating: Altitude lower than 3000 meters; non-operating: Altitude lower than 15000 meters
- Unless otherwise specified, operating temperature is 0 to +40°C; storage temperature is -

20 to +70°C

- Operating: humidity temperature below to +35°C, ≤90% RH. (Relative humidity)

  Non-operating: humidity temperature +35°C to +40°C, ≤60% RH.

There are ventilation outlets on the rear and side panels of the instrument, please keep the air ventilation in the outlet of housing. To prevent excessive dust from blocking the vents, clean the instrument housing regularly. The housing is not waterproof; please cut off the power supply first and then wipe the housing with a dry cloth or a slightly moistened soft cloth.

Connecting Power Supply

The specification of the AC power supply is as shown in the following table.

Voltage Range	**Frequency**
100V-240VACrms (Fluctuations ±10%)	50/60Hz

Please use the attached power cord to connect to the power port.

Connecting to the service cable:

This instrument is a Class I safety product. The supplied power cables have reliable performance in terms of case grounding. This spectrum analyzer is equipped with a three-prong power cable that meets international safety standards. It provides good case grounding performance for the specifications of your country or region.

Please install the AC power cable as follows:

- Ensure the power cable is in good condition.
- Leave enough space to connect the power cord.
- Plug the attached three-prong power cable into a well-grounded power socket.

Electrostatic Protection

Electrostatic discharge may cause damage to components. Components can be invisibly damaged by electrostatic discharge during transportation, storage, and use.

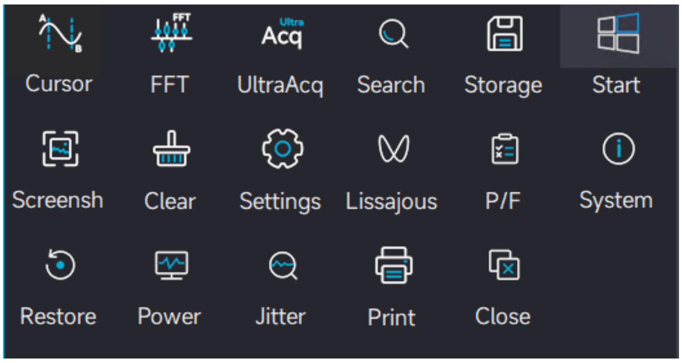
The following measures can reduce the damage caused by electrostatic discharge:

- Test in an antistatic area as far as possible.
- Before connecting the power cable to the instrument, briefly ground the inner and outer conductors of the instrument to discharge static electricity.
- Ensure all instruments are properly grounded to prevent the accumulation of static.

Document Formatting Conventions

Menu

"Menu Text (bold) + Character Shading" is used to represent a menu option, such as **Settings** represents the setting menu on the screen.



Operation Steps

Use "**>**" to show the next step.

Connector

In this manual, “Square Brackets + Text (bold)" is used to indicate a connector on the front or rear panel, such as **[TRIG OUT]**.

UPO7000L Overview

Compact Rack-Mounted Design for High-Density Integration: 214mm (half-width) × 43mm (1U) ×478mm

UPO7000L series digital phosphor oscilloscopes feature a compact, rack-mounted structural design with a slim and lightweight body. 1U height is designed for multi-machine system integration, high-density rack setups, and remote system operations, making it ideal for various application scenarios. The system supports multi-unit synchronous triggering and can be expanded to accommodate up to 128 oscilloscopes. Each unit integrates 4 analog channels, 1 external trigger channel, and 1 function/arbitrary waveform generator channel. With a flat body design and machine feet pads, the oscilloscopes are easy to stack and organize. leveraging the 7000 series platform, it ensures a smooth transition for users familiar with 7000X operation. Additionally, an external touch display can be connected, enabling a responsive touch experience similar to that of the 7000X series.

For multi-machine integration, the series includes a rack-mounting kit for quick and straightforward installation right out of the box. Whether in system development, testing, or other demanding environments, UPO7000L excels in reliability and performance.

UPO7000L series digital phosphor oscilloscopes include the following models.

Model	Analog Channel	Analog Bandwidth	AWG	Power Analysis	Jitter Analysis	Eye Diagram
UPO7204L	4	2GHz	○	○	○	○
UPO7104L	4	1GHz	○	○	○	○

○: Indicates option

Main Features

- Analog channel bandwidth: 1G/2GHz
- Maximum sampling rate: 10GSa/s
- Maximum storage depth: 1Gpts (Option)
- Wave capture rate: ≥600,000wfms/s (UltraAcq® mode) ; 2,000,000wfms/s (Sequence mode)
- Multiple triggering types: Edge, pulse width, slope, video, code pattern, timeout, runt, setup & hold, delay, duration, and Nth-edge
- 11 kinds of serial protocol analysis: RS232/422/485/UART, I^2C, SPI, CAN, CAN FD, LIN, FlexRay, SENT, MIL-STD-1553, ARINC 429, and AudioBus (I^2S/LJ/RJ/TDM)
- Integrates 6 kinds of instrument functions: Digital oscilloscope, spectrum analyzer, function/arbitrary waveform generator, digital voltmeter, frequency meter, and protocol analyzer
- 48 kinds of parameter measurement, it supports histogram, trace, and tendency chart
- Multiple advanced measurement analysis functions: Power analysis (Option), jitter & eye diagram (Option), template test, histogram, and search navigation
- Built-in WebServer can access the instrument and observe the measurement through a browser, supporting two styles of layout and operation of PC/smartphone, easy to realize cross-platform access
- Supports SCPI (Standard Command for Programmable Instruments)
- Various interfaces: USB Host & Device, LAN, HDMI, AUX Out, 10MHz Ref In/Out, and Audio
- 8-channel waveform operation, built-in frequency spectrum analysis and peak search function, supporting Matlab embedded programming and data presentation, and support enhanced resolution up to 3bits
- Built-in function/arbitrary waveform generator with a bandwidth of 60MHz
- Features a 1U height, a compact, rack-mounted structural design with a slim and lightweight body
- Provides a software development package to support secondary development needs
- Supports multi-machine synchronization and can be expanded to accommodate up to 128 oscilloscopes for simultaneous data acquisition.

1. Quick Guide

This chapter is to introduce basics of using the UPO7000L series oscilloscope for the first time, front panel, rear panels, and user interface.

1.1 General Inspection

It is recommended to inspect the instrument by following the steps below before using the UPO7000L series oscilloscope.

(1) Check for transport damage

If the packaging carton and the plastic foam cushions damage. If significant damage is found, please contact the UNI-T distributor.

(2) Check accessories

Refer to the appendix for the list of included accessories. If any accessories are missing or damaged, please contact the UNI-T distributor.

(3) Machine inspection

Examine the instrument for any visible damage, operational issues, or failures during the functionality test. If problems are detected, contact the UNI-T distributor.

If the instrument is damaged during shipping, retain the packaging materials and notify both the transportation department and the UNI-T distributor. UNI-T will arrange for maintenance or replacement as necessary.

1.2 Before Use

To perform a quick verification of the instrument's normal operations, follow the steps below.

1. Connecting Power Supply

The power supply voltage range ranges from 100VAC to 240VAC, with a frequency range of 50Hz to 60Hz. Use the assembled power cable or another power cable that meets the local country standards to connect the oscilloscope. When the power switch on the rear panel is disabled, the power soft indicator in the left bottom on the rear panel lights up orange, press the soft power key to turn on the oscilloscope; when the power switch on the rear panel is enabled, the oscilloscope will automatically power on.

2. Boot-up Check

Press the soft power key to turn on the oscilloscope, the indicator will change from orange to blue. The oscilloscope will show a boot animation before entering the normal interface.

3. Connecting Probe

Use the assembled probe, connect BNC of probe to CH1 BNC on the oscilloscope, connect the probe tip connects to "Probe Compensation Signal Connection Sheet", and connect the ground alligator clip to the "Ground Terminal" of probe compensation signal connection sheet, as shown in the following figure. The probe compensation signal connection sheet outputs an amplitude of approximately 3Vpp and a default frequency of 1kHz.

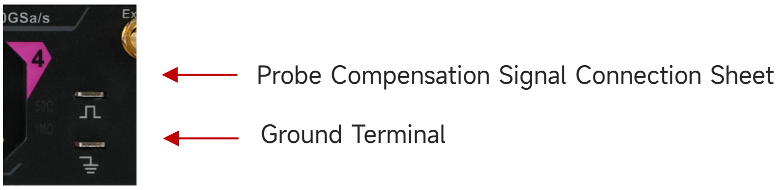


Figure Probe Compensation Signal Connection Sheet and Ground Terminal

4. Function Inspection

Press the Autoset (Automatic Setting) icon, a square wave with an amplitude of approximately 3Vpp and a frequency of 1kHz will appear on the screen. Repeat step 3 to check all channels. If the displayed square wave shape does not match the one shown in the figure above, proceed to the next step "Probe Compensation."

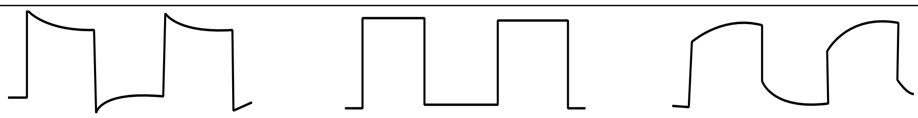
5. Probe Compensation

When the probe is connected to any input channel for the first time, this step may need to be adjusted to match the probe and the input channel. Uncompensated probes can lead to measurement errors or inaccuracies. Please follow the steps below to calibrate probe compensation.

(1) Set the attenuation coefficient in the probe menu to 10x and ensure the probe switch is set to 10x. Connect the probe to CH1 on the oscilloscope. If using the probe's hook head, ensure it makes stable contact with the probe.

(2) Connect the probe tip to the "Probe Compensation Signal Connection Sheet" and the ground alligator clip to the "Ground Terminal" of the "Probe Compensation Signal Connection Sheet." Open CH1 and press the Autoset icon.

View the displayed waveform, as shown in the following figure.



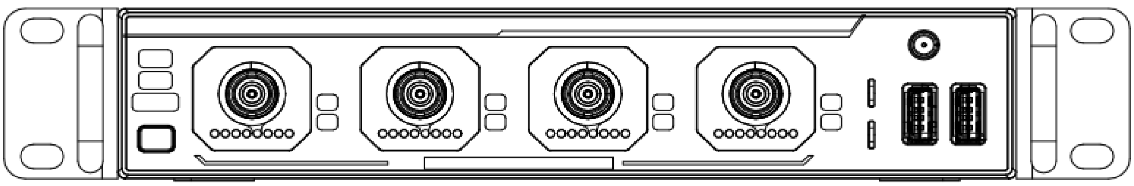
Excessive Compensation Correct Compensation Insufficient Compensation

Figure Probe Compensation Calibration

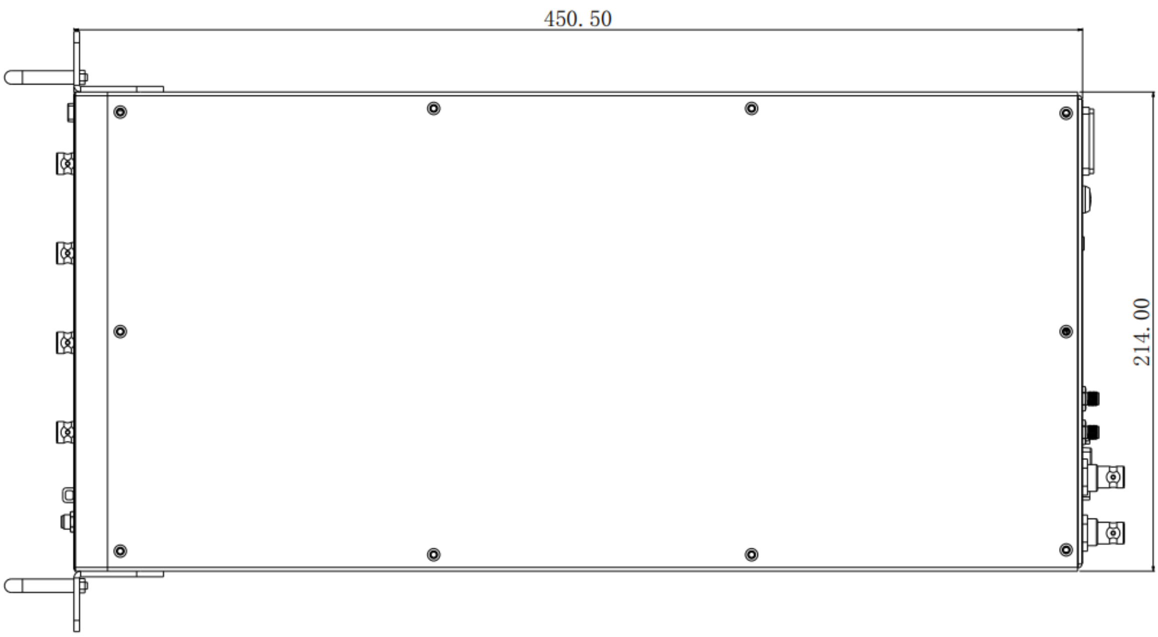
If the displayed waveform appears as "Insufficient Compensation" or "Excessive Compensation," use a non-metallic screwdriver to adjust the probe's variable capacitance until the display matches the "Correct Compensation" waveform.

Warning: To avoid the risk of electric shock when using the probe to measure high voltage, ensure that the probe insulation is intact and avoid physical contact with any metallic parts of the probe.

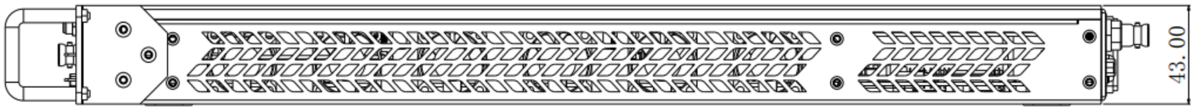
1.3 Appearance and Dimensions



Front View



Top View



Side View

1.4 Front Panel

Front Panel

Table 1 Front Panel Connectors

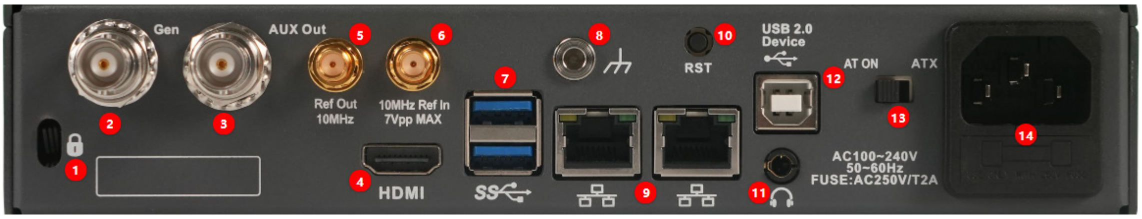
No.	Description	No.	Description
1	Nameplate/Model series	4	Probe compensation signal connection sheet and ground terminal
2	External trigger SMA connector	5	Analog channel input terminal
3	USB HOST 2.0	6	Soft power switch

Table 2 Front Panel Key Indicator

Key Indicator	Red	Green	Blue	Yellow	None
Power			Powered on	Powered on but not enabled	
RunStop	Stop	Run	The channel's microcontroller has been powered on, but the software has not been started yet	Abnormal	
Lan	Network connection failed	Network connection normal			
Acq	Stop acquisition	Triggered		The oscilloscope is currently	

				capturing pre-trigger data.	
Impedance			1MΩ	50Ω	The channel is not opened

1.5 Rear Panel



Rear Panel

Rear Panel Connectors

1. Safety Keyhole: A safety lock (purchased separately) can be used to lock the oscilloscope in a fixed position through the keyhole;
2. Output port of function/arbitrary waveform generator;
3. Aux Out: Trigger synchronous input; Pass/Fail test results; AWG trigger output;
4. HDMI: High-definition multimedia interface;
5. 10MHz Ref Out: A BNC on the rear panel that outputs the oscilloscope's 10MHz reference clock for synchronization with other external instruments;
6. 10MHz Ref In: Provides the reference clock for oscilloscope's acquisition system;
7. USB Host: Through this interface, USB-compatible storage devices can be connected to the oscilloscope. When connected, waveform files, setting files, data, and screen images can be saved or retrieved. Additionally, if updates are available, the oscilloscope's system software can be locally upgraded via the USB Host port.;
8. Ground Hole: The device can be grounded to generate static electricity;
9. LAN: Use this port to connect the oscilloscope to the LAN (local area network) for remote control;
10. RST: Restart device;
11. Audio port;
12. USB DEVICE 2.0: Use this port to connect the oscilloscope to PC for communication;
13. Auto Power On: Automatic power-on setting switch, toggle the switch to AT ON, oscilloscope power on automatically after startup;
14. AC power supply: 100-240VAC, 50-60Hz;

1.6 User Interface

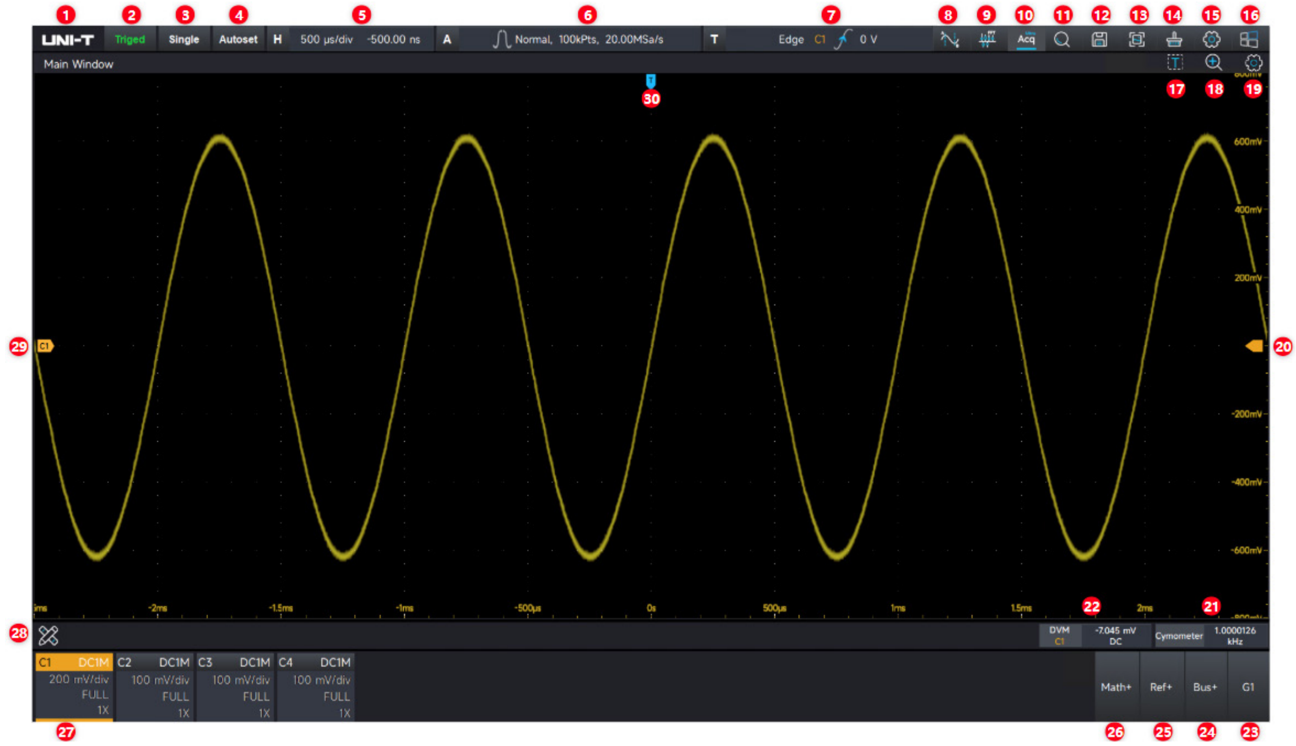


Table 3 Icon in User Interface

No.	Description	No.	Description
1	UNI-T Logo	17	Zone triggering
2	Trigger state icon	18	Window extension
3	Single trigger	19	Main window setting menu
4	Autoset	20	Trigger level cursor
5	Horizontal scale and delay	21	Frequency meter
6	Acquisition mode, storage depth and sampling rate	22	Digital voltmeter
7	Trigger information	23	Function/arbitrary waveform generator
8	Cursor measurement	24	Protocol analyzer
9	FFT	25	Reference waveform
10	UltraAcq® mode	26	Mathematical operation
11	Search Navigation	27	Channel state label
12	Save	28	Measurement menu
13	Screenshot	29	Analog channel cursor and waveform

14	Delete	30	Trigger position cursor
15	System setting		
16	Start menu		

1. Manufacture's LOGO: UNI-T
2. Trigger state icon: Triged (Triggered), Auto, Ready, Stop, and Roll.
3. Single trigger: Sets the oscilloscope's trigger mode to "Single." When activated, this icon turns green, and the oscilloscope executes one trigger upon receiving a signal that meets the trigger condition. If the "Run/Stop" icon displays red, indicating the "Stop" state, the oscilloscope will not refresh even when a signal meeting the trigger condition is received.
4. Autoset: Click this icon with the mouse to automatically configure the oscilloscope's vertical scale, time base, and trigger mode. The oscilloscope adjusts these settings to display the most appropriate waveform based on the input signal.

   Note: When the waveform automatic function is activated, if the measured signal is a sine wave, its frequency cannot be less than 20Hz, and its amplitude must range from 10mVpp to 80Vpp. Otherwise, the waveform automatic function may malfunction.
5. Horizontal scale and delay: Click this icon with the mouse to access "Time Base" setting menu.

   Time base: This represents the time value per division on the horizontal axis of the waveform display area. The current horizontal scale can be adjusted by clicking  and  icons, changing the scale in 1-2-5 steps.

   Delay: This indicates the distance of the current waveform's trigger point from the horizontal center of the display. Positive values indicate the trigger point is shifted to the left, while negative values indicate it is shifted to the right. The delay can be adjusted by clicking the  and  icons, or clicking the delay text field to pop up the numeric keypad for inputting the desired delay time. Clicking the delay zero key centers the trigger point on the display.
6. Acquisition mode, memory depth, and sampling rate: Displays the acquisition mode (normal, peak detection, high definition, average, and envelope), memory depth, and real-time sampling rate.
7. Trigger information: Displays the trigger source, trigger type, and trigger level.
   - Trigger source: C1-C4, EXT, EXT/5, and AC. The state color of trigger source matches the corresponding channel's color.
   - Trigger type: Edge, pulse width, video, slope, runt amplitude, delay, overtime, duration, setup & hold, N-edge, code pattern and serial triggering.

- Trigger level: Displays the trigger level, it corresponds to the icon on the right side of the screen (The trigger level's state color corresponds to the channel's color.) Clicking the − and + icons in the trigger setting popup to adjust the trigger level, or clicking the trigger level text field to pop up the numeric keypad for inputting the desired trigger level.

8. Cursor measurement: Click this icon with the mouse to access the cursor setting menu. Turn on/off the cursor, select the cursor type, set the sync movement, adjust the cursor position, and specify the cursor measurement unit.
9. FFT: Open an independent Math window with operation type of FFT.
10. UltraAcq® mode: Click the "Acq" icon to activate UltraAcq® (Fast sampling mode), enhances the waveform capture rate to 600,000wfms/s.
11. Search navigation: Search and navigate through the waveforms.
12. Save: Click the "Save" icon to access the save setting menu.
    - Save waveform: Waveform data can be saved with these formats: .bin/.txt/ .mat/ .xlsx/.csv/ .tsv/ .dat, or .bsv.
    - Save screenshot: Save the screen area or grid area; save color: standard, black and white, and reverse color; save picture format: .bmp/.tiff/.gif/.png/.jpeg.
    - Save system settings: The current system setting can be saved as .set file. The user can read the saved setup file to restore the last saved setting state.
13. Screenshot: Take a quick screenshot based on the configured screenshot save settings, and display a prompt confirming the successful save.
14. Delete: Delete the old waveform displayed on the screen, including its associated measurement parameters.
15. Setting: Click the "Setting" icon to access the setting menu, configure brightness, automatic settings and calibration, communication, auxiliary output, and other options.
16. Start menu: Click the "Start" icon to access the start menu, configure cursor measurement, FFT, Lissajous, P/F test, power analysis, jitter analysis, and eye diagram.
17. Histogram zone: Draw the waveform in the histogram zone, with vertical and horizontal statistical histograms.
18. Window extension: Used to enlarge a portion of the waveform to observe the image details. Clicking the icon to open the view extension and clicking the icon to close the view extension.
19. Main window settings: Click the main window setting icon to adjust the persistence, horizontal

or vertical cursor marker position, grid type, grid brightness, waveform type, and waveform brightness.

20. Trigger level cursor: Displays the trigger level position on the current channel. The state color of the trigger level cursor matches the corresponding channel's color.
21. Frequency meter: 8-digit high precision hardware frequency meter
22. Digital voltmeter: 4-digit DC/AC RMS/DC+AC RMS voltage measurement
23. Function/arbitrary waveform generator: Click the G1 icon to activate the function/arbitrary waveform generator.
24. Bus: Click the "Bus+" icon to access the analyzer function, supports 11 types of protocol analysis.
25. Reference: Click the "Ref+" icon to add the reference waveform to the oscilloscope for analysis and measurement, supporting two formats .bin and .csv.
26. Math: Click the "Math+" icon to access the mathematical operations, supporting enhanced FFT, basic operations, filter, advanced formula editing, embedded Matlab programming operation and rendering, and enhanced resolution. Additionally, it supports 8 math waves.
27. Channel state label: Displays the channel activate state, channel coupling, bandwidth limit, vertical scale, probe attenuation coefficient, and inverse phase.
    - Channel activate state: If the channel menu is lit it means the channel is activated. If it is gray, it means the channel is disabled.
    - Channel coupling: DC1MΩ, AC1MΩ, DC50Ω, Ground
    - Bandwidth limit: "FULL" indicates full bandwidth. When the bandwidth limit is enabled, the current bandwidth limit will be displayed.
    - Vertical scale: Displays the vertical scale of the analog channel, clicking the channel state label to pop up the analog channel window, clicking the  and  icons to adjust this parameter.
    - Probe attenuation coefficient: Displays the probe attenuation coefficient of the analog channel, including 1X, 10X, 100X, and a user-defined value.
    - Inverse phase: The channel state label displays "↓" when the inverse phase is enabled. The channel state label does not display "↓" when the inverse phase is disabled.
    - Click the analog channel color label with mouse to open the corresponding channel; double-click the mouse or slide down to close the corresponding analog channel. Slide down to close other channels.
28. Measurement menu: Click the measurement menu icon to access the menu, enable the digital

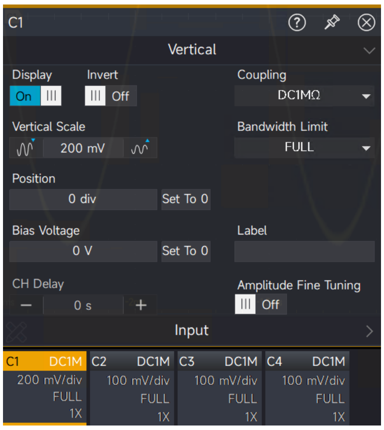
voltmeter measurement, frequence meter measurement, open the parameter snapshot, view measurement statistics, and add parameter measurements.

29. Analog channel waveform position
30. Trigger point position

2. Vertical System

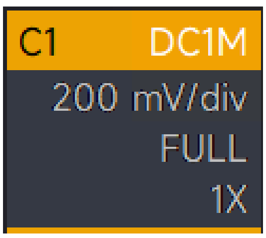
- Open/Activate/Close Analog Channel
- Channel Coupling
- Bandwidth Limit
- Vertical Scale
- Vertical Position
- Offset Voltage
- Probe Unit
- Inverse Phase
- Label
- Probe Attenuation Factor
- Multi-window Display
- Channel Delay

Note: UPO7000L provides 4 analog channels of C1-C4. The setup method for the vertical system of each channel is identical. This chapter explains the vertical channel settings using C1 as an example.

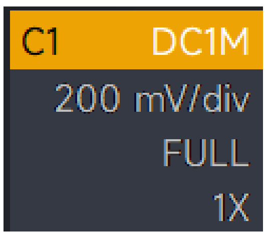


2.1 Open/Activate/Close Analog Channel

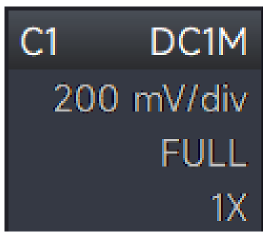
C1-C4 analog channels have three states: open, closed, and activated.



Activated State



Open but not Activated



Off State

Open: The analog channels are identified by different colors, and the waveform colors displayed on the screen correspond to the colors of the channel input connectors.

- Peripheral operation: When an analog channel appears gray, click the gray block with the mouse

to open the corresponding channel.

Close: The channel's waveform. is not displayed.

- Peripheral operation: Any channel can be turned off by clicking and sliding down with the mouse,

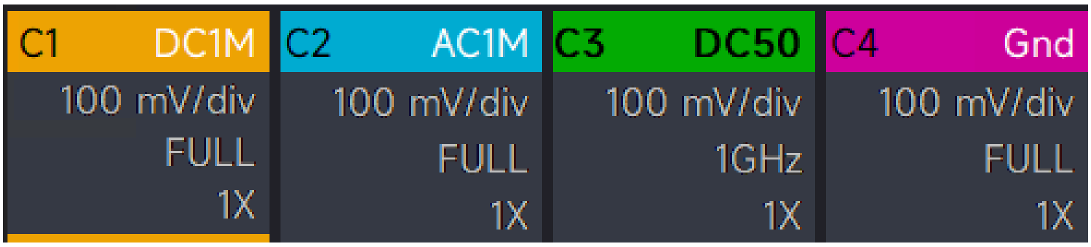
regardless of whether the channel is activated.

Activated: When multiple channels are turned on simultaneously, only one channel can be activated (the channel must be open). In the activated state, the channel's vertical scale, vertical shift, and settings can be adjusted.

- Peripheral operation: Click the channel menu with the mouse to select the corresponding channel.

2.2 Channel Coupling

The channel coupling can be set in the channel menu, where four coupling types are available: DC 1MΩ, DC 50Ω, AC 1MΩ, and Ground. In UltraAcq® mode, Ground is not available.



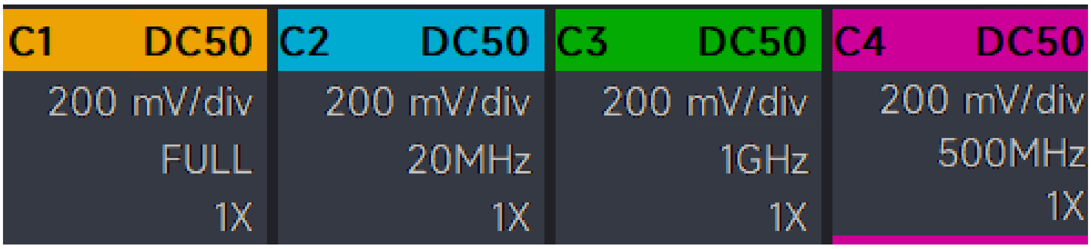
DC 1MΩ AC 1MΩ DC 50Ω Ground

2.3 Bandwidth Limit

1MΩ impedance state: The bandwidth limitation can be set to either full bandwidth or 20MHz.

50MΩ impedance state: The bandwidth limitation can be set to full bandwidth, 1GHz, 500MHz, or 20MHz.

If the soft key menu is set to 20MHz, the oscilloscope's bandwidth limit will be restricted to 20MHz. High-frequency signals above 20MHz will be attenuated, which is often used to reduce high-frequency noise when observing low-frequency signals.



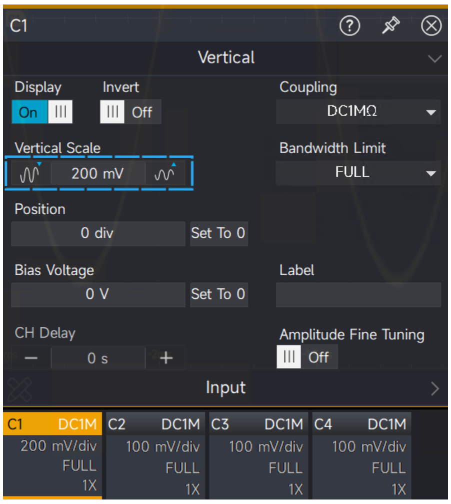
Full Bandwidth 20MHz 1GHz 500MHz

2.4 Vertical Scale

The vertical scale range of oscilloscope is 1MΩ: 1mV/div to 10V/div; 50Ω: 1mV/div to 1V/div, with steps of 1-2-5.

Note: "div" indicates the grid in the oscilloscope's waveform display area, with each "div" representing one grid.

- Mouse operation: Click the channel menu with the mouse to open the setting menu, clicking the  and  icons in the vertical scale menu to adjust the vertical scale.
- Numeric keypad: Click the text field in the channel menu to open the numeric keypad and enter the desired vertical scale value.



2.5 Vertical Position

The vertical position indicates the position of the current waveform on the screen. Changing the vertical position will not affect the waveform's voltage value. The vertical position of the current channel can be adjusted using mouse operation.

- Peripheral operation: Select the waveform with the mouse and drag it up or down to adjust its position.

Note: Dragging the waveform up or down cannot exceed the current vertical limit of 5.5div.

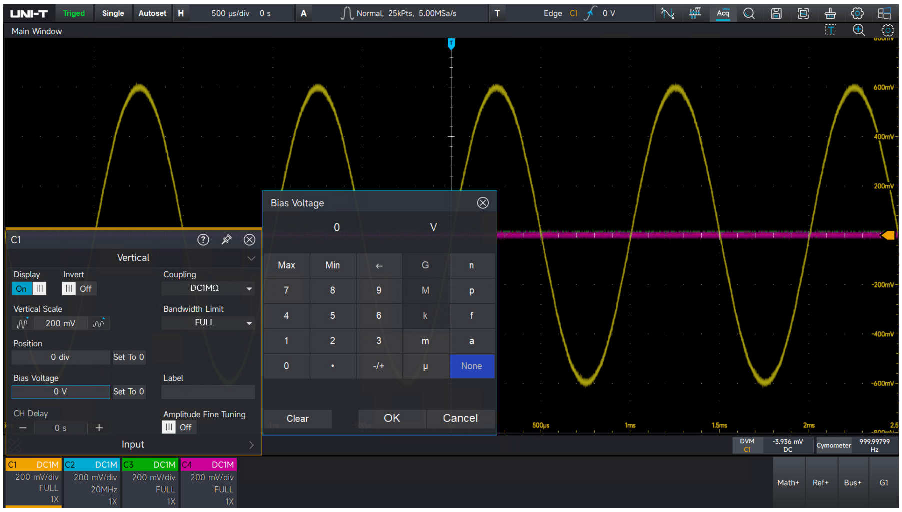
- Numeric keypad: Click the position text field to open the numeric keypad and enter the desired vertical position value. A positive value indicates an upward shift, while a negative value indicates a downward shift.

Note: The input value cannot exceed the current vertical limit of 1.5div.

Set to 0: Click the channel menu and set the value to 0, which will adjust the channel waveform position to the center of the screen.

2.6 Offset Voltage

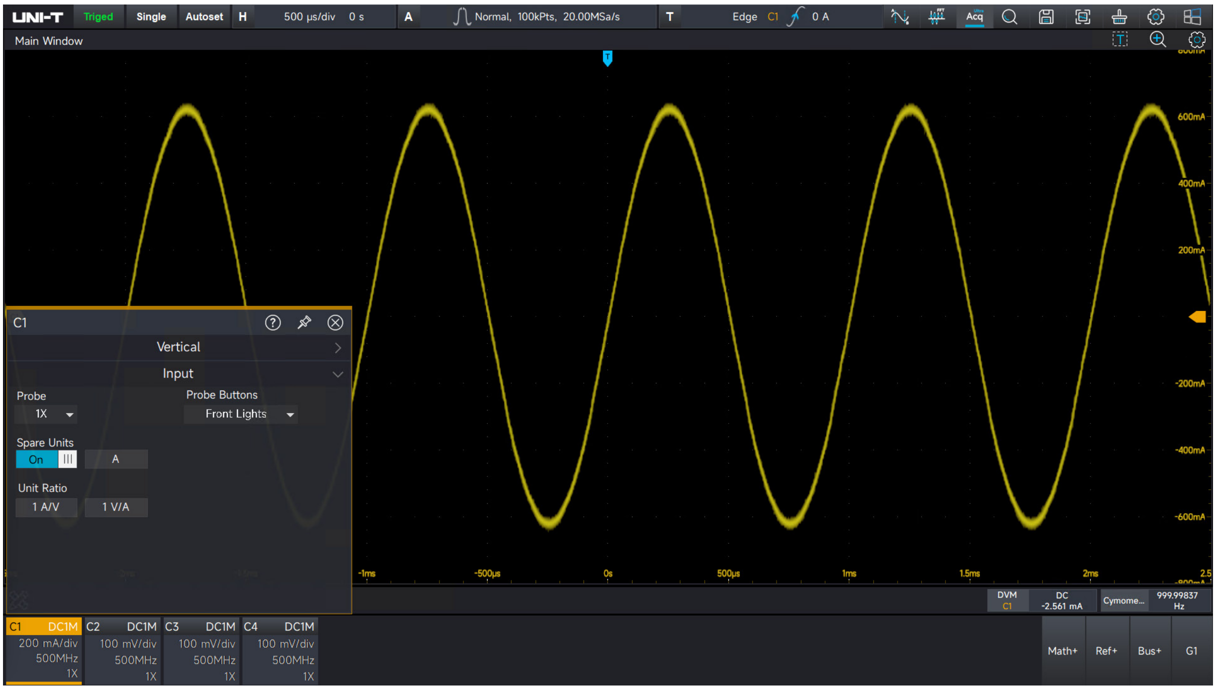
The offset voltage indicates the voltage offset value of the current channel, it can be set by the numeric keyboard. The channel voltage will change with the setup and the waveform will also move with the vertical movement.



Set to 0: Click the 0 label next to the offset voltage, and adjust the channel's offset voltage to 0mV.

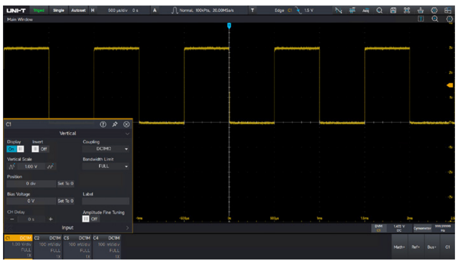
2.7 Probe Unit

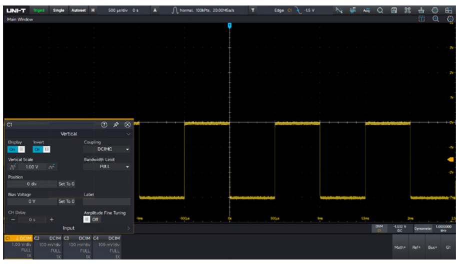
The probe unit can be configured in the probe input setting menu. Set different units to suit different measurement scenarios. For example, when using a current probe to measure current, the unit should be set to A/mA for easier observation. UPO7000L supports unit conversion between V/A/W, and U.



2.8 Inverse Phase

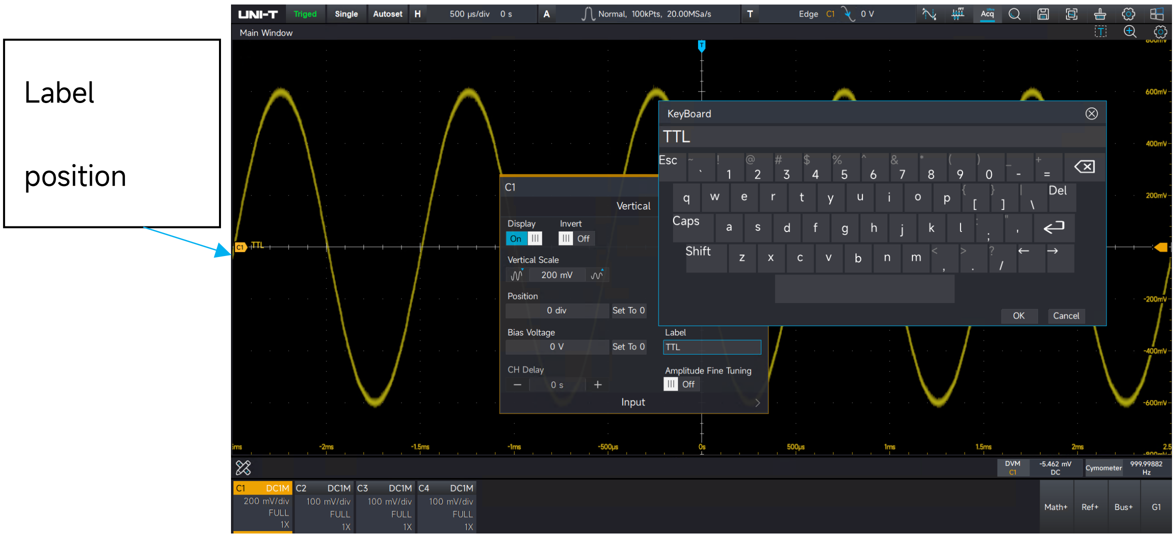
The inverse phase can be configured in the channel menu. When the inverse phase is enabled, the waveform voltage is inverted, and the reverse icon  will be displayed in the vertical state.


Inverse Phase OFF


Inverse Phase ON

2.9 Label

Click the label in the channel menu to open the soft keyboard. The input channel label can be customized through the soft keyboard or by connecting an external physical keyboard to distinguish the markings of the channel's waveform during use. The input label will appear at the channel cursor on the left side of the screen.

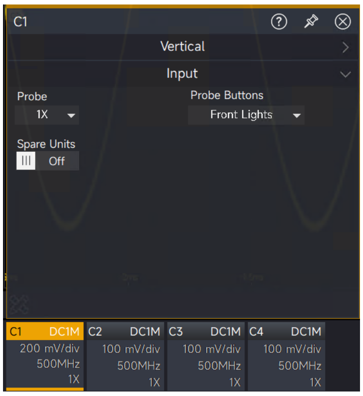


2.10 Probe Attenuation Factor

To match the probe attenuation factor setting, the probe attenuation factor must be configured accordingly in the channel operation menu. For example, if the probe attenuation factor is 10:1, indicating that the probe attenuates the measured signal to one-tenth before inputting it into the oscilloscope, then the probe factor in the oscilloscope's channel menu should be set to ×10. This setting ensures that the oscilloscope enlarges the input signal by a factor of 10 to display the correct measured voltage.

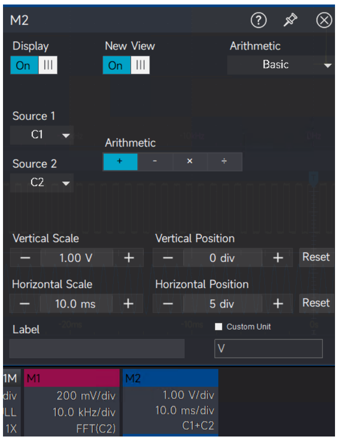
The probe can be set to ×1, ×10, ×100, or a user-defined value.

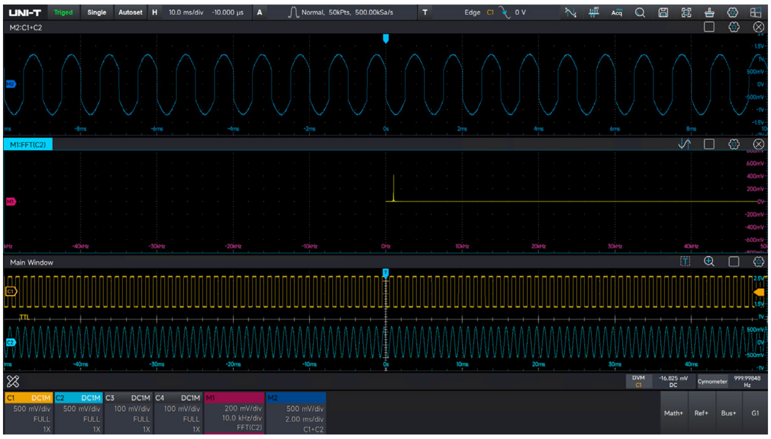
Note: When the oscilloscope is connected to a probe equipped with a probe attenuation factor detection pin (which uses different resistors to represent various attenuation factors), the oscilloscope automatically detects the probe's attenuation factor and adjusts its setting to match it.



2.11 Multi-window Display

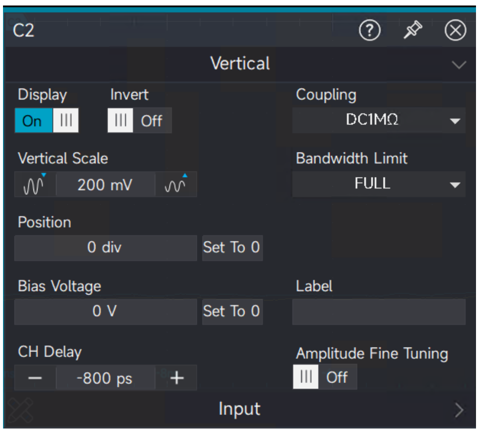
The independent window can be enabled by clicking the channel menu. It supports functions such as mathematical operations, reference waveforms, protocol analysis, and digital channels. When the independent window is enabled, a corresponding horizontal dividing line will appear at the bottom, and the horizontal and vertical scales will be displayed independently. The window layout can be adjusted using the mouse by dragging and dropping the window to the desired position.



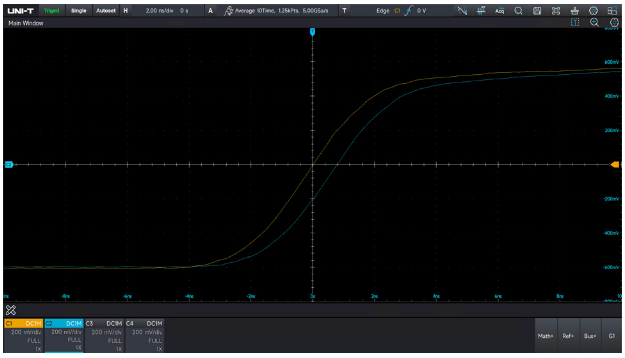


2.12 Channel Delay

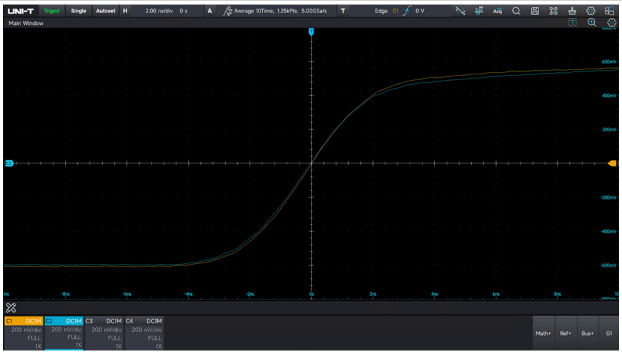
When performing actual measurements with an oscilloscope, the transmission delay of the probe cable may introduce significant errors, such as a zero offset. UPO7000L ensures that the phases among channels remain synchronized by setting the channel delay step. This feature is useful in application scenarios that require high-precision measurements, such as power analysis. To adjust the delay step, click the channel menu and select the desired value on the settings page. The delay step can be set in increments as small as 100 picoseconds within a range of ±800 picoseconds.



The channel delay calibration is valid only when the time base is set to 50ns/div or lower.



Before Delay Calibration



After Delay Calibration

3. Horizontal System

- Horizontal Scale
- Horizontal Delay
- Roll Mode

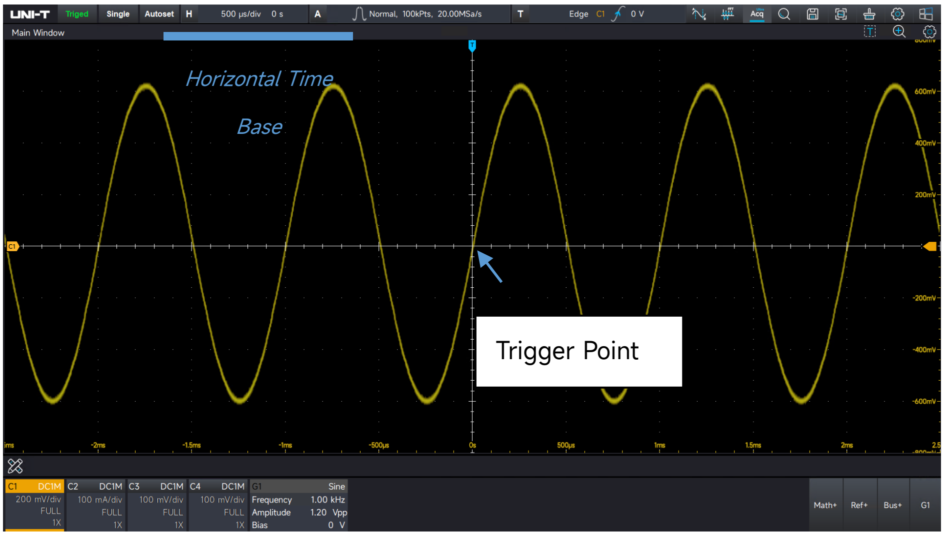
3.1 Horizontal Scale

The horizontal scale is also known as the horizontal time base, i.e., the time value represented by each scale in the horizontal direction of the screen, usually expressed as s/div.

Mouse operation: Click the horizontal time base label at the top left of the screen with the mouse to open the setting menu. Use the and icons to adjust the horizontal time base, or click the horizontal scale text field to open the numeric keypad and input the desired time base value.

icon indicates increase the time base, while icon indicates decrease the time base.

Note: The scale changes in 1-2-5 steps, and the input value does not affect the stepping pattern.



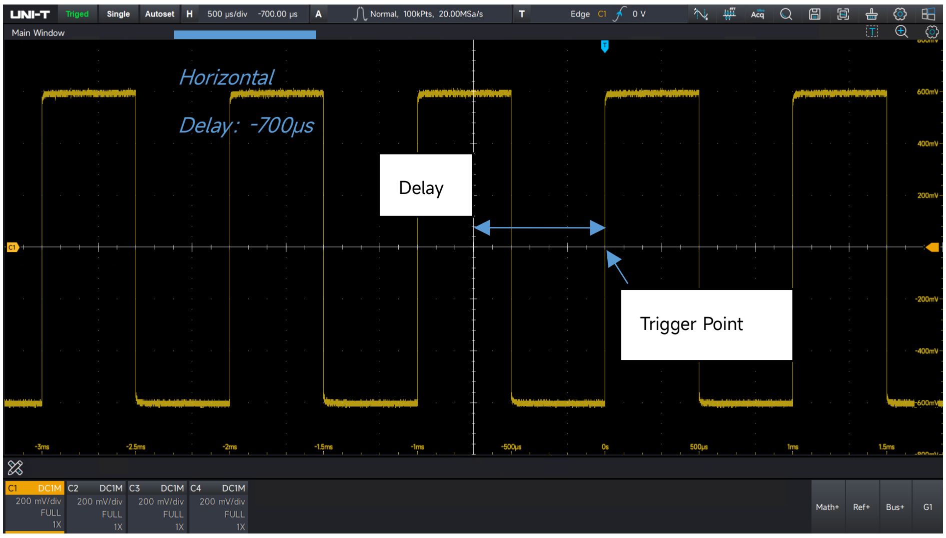
3.2 Horizontal Delay

The horizontal delay is also known as the horizontal shift, i.e., the movement of the trigger point left or right relative to the center of the screen.

Mouse operation: Click the horizontal time base label at the top left of the screen with the mouse to open the setting menu. Use the and icons to adjust the horizontal delay, or click the horizontal delay text field to open the numeric keypad and input the desired horizontal delay value.

Click the delay zero to set the horizontal delay of the current channel to zero.

Note: A positive value moves the trigger point to the left, while a negative value moves it to the right.

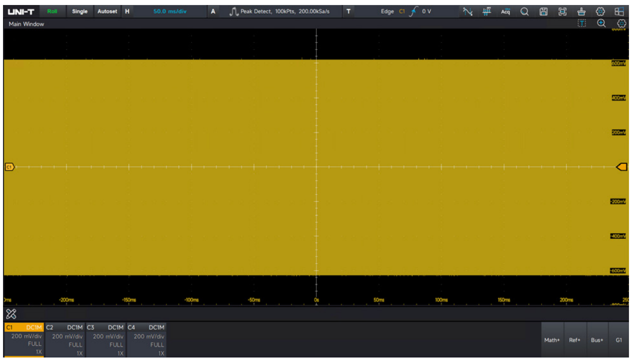


When changing the horizontal delay, the waveform will move left or right along with the trigger point.

Peripheral operation: When the channel is selected, click on the waveform with the mouse and drag it left or right to adjust the waveform delay.

3.3 Roll Mode

When the trigger mode is set to automatic and the oscilloscope's horizontal time base is slower than 50ms/div, the oscilloscope will enter Roll mode. The oscilloscope will continuously draw a voltage-time trend of the waveform on the screen. In ROLL mode, the waveform scrolls from right to left, with the latest waveform drawn at the far-right end of the screen.



When using Roll mode to observe low-frequency signals, it is recommended to set the "Channel Coupling" to "DC."

4. Sampling System

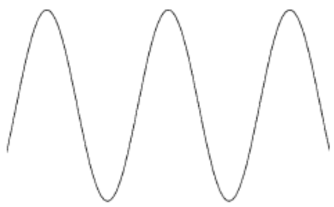
- Sampling Rate
- Acquisition Mode
- Memory Depth
- Sequence Mode
- ERes

Sampling is the conversion of the signal from an analog input channel, through an analog-to-digital converter (ADC), into a discrete point.

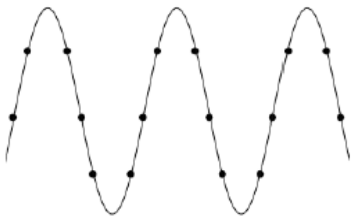
4.1 Sampling Rate

(1) Sampling and Sampling Rate

Sampling refers to the process where the oscilloscope takes samples from the input analog signal, converts them into digital data, and then gathers the digital data into waveform records. These waveform records are stored in the oscilloscope's memory.



Analog Input Signal



Sampling Point

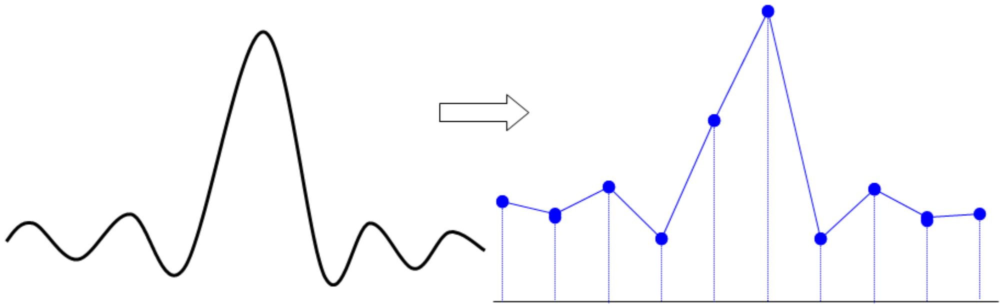
The sampling rate refers to the number of data points the oscilloscope samples per second. UPO7000L series digital phosphor oscilloscopes support a maximum sampling rate of 10GSa/s.

The sampling rate depends on the number of active channels: Single channel (10GSa/s), dual-channel (5GSa/s), and four channel (2.5GSa/s).

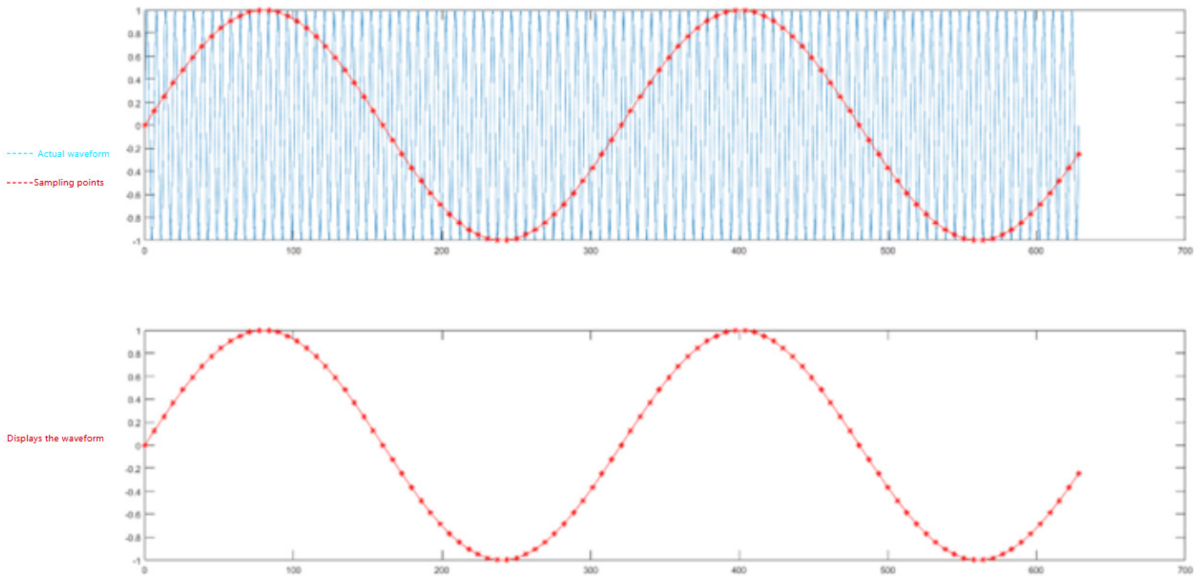
The sampling rate is influenced by the time base scale and storage depth settings.

(2) Effect of Low Sampling Rate

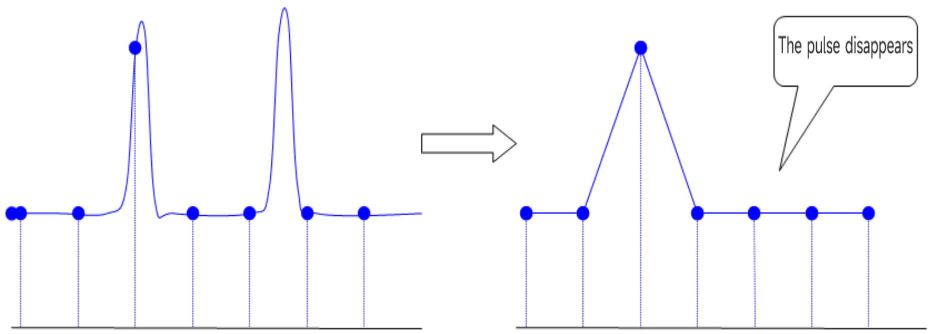
Waveform distortion: Due to low sampling rate, the details of the waveform might be missing, the sampled waveform might have large different than the actual signal, as shown in the following figure.



Waveform aliasing: Since the sampling rate is 2 times lower than the actual signal frequency (Nyquist frequency), the waveform frequency is less than the frequency of actual signal when sampling data is reconstructing, as shown in the following figure.



Waveform missing: Due to the low sampling rate, the waveform does not reflect all the actual signals, as shown in the following figure. The pulse disappears



(3) Interpolation Method

Interpolation involves inserting calculated values between data points collected by the Analog-to-Digital Converter (ADC) based on a specific algorithm. This process increases the sampling rate, providing clearer signal detail analysis.

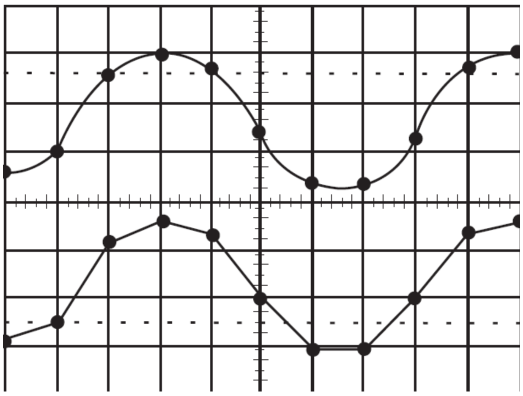
Linear Interpolation:

Linear interpolation inserts a calculated value between adjacent ADC sample points using a linear polynomial method. The interpolated value lies on the line connecting the two sampled points.

Sine Interpolation

Sine interpolation is based on the principle that any waveform can be decomposed into an infinite combination of sine waves. Using this method, the real waveform signal can be restored more accurately and smoothly by connecting sample points with curves rather than straight lines.

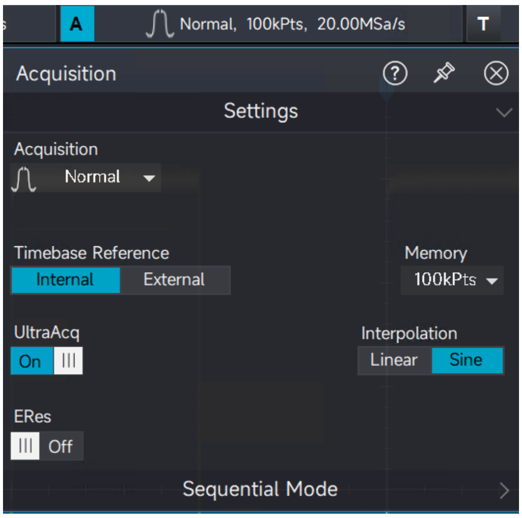
This approach employs mathematical processing to calculate values within the intervals between actual sample points, resulting in a more precise representation of the original signal.



Sinusoidal Interpolation (Up line) and Linear Interpolation (Down line)

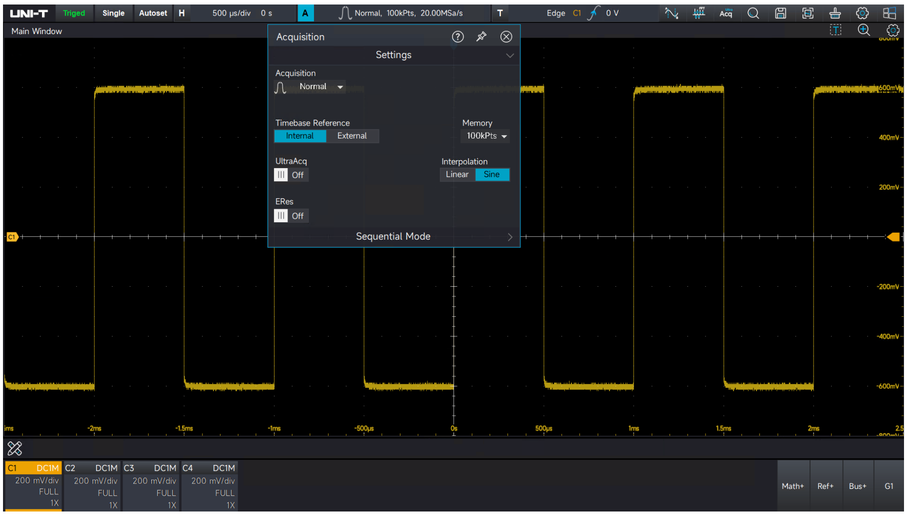
4.2 Acquisition Mode

The acquisition mode can be configured in the sampling menu. The acquisition mode has normal sampling, peak detection, high resolution, average sampling, and envelope.



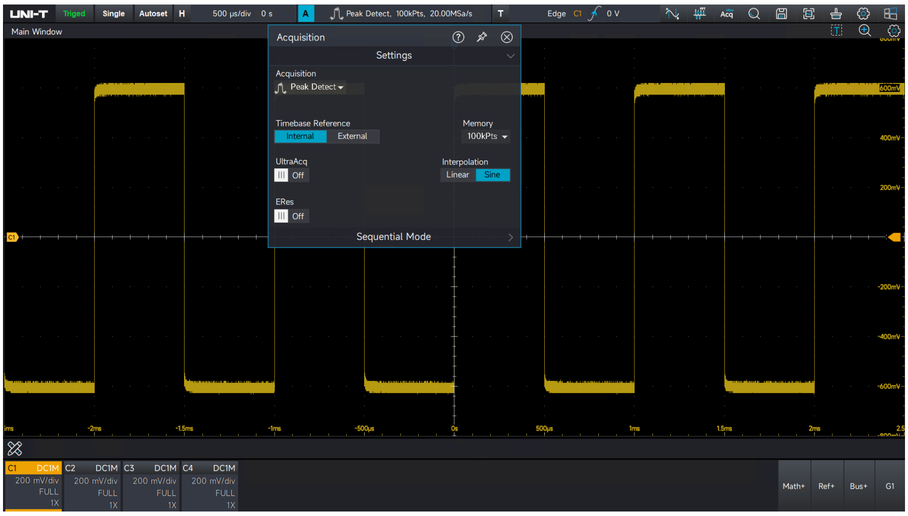
(1) Normal Sampling

In this mode, the oscilloscope samples the signal and reconstructs the waveform at uniform time intervals. For most waveforms, this mode provides an optimal display effect.



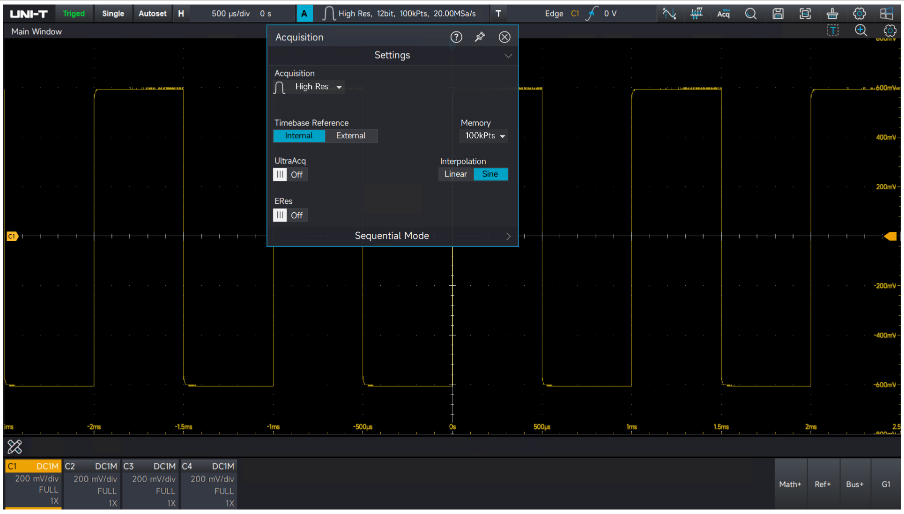
(2) Peak Detection

In peak detection mode, the oscilloscope identifies the maximum and minimum values of the input signal within each sampling interval and uses these values to reconstruct the waveform. This method ensures that narrow pulses, which may be missed in normal sampling, are captured and displayed. However, in this mode, noise may also appear more pronounced.



(3) High Resolution

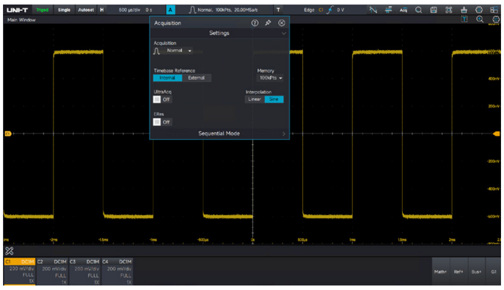
The oscilloscope averages adjacent sample points to reduce random noise in the input signal, producing a smoother waveform on the screen.



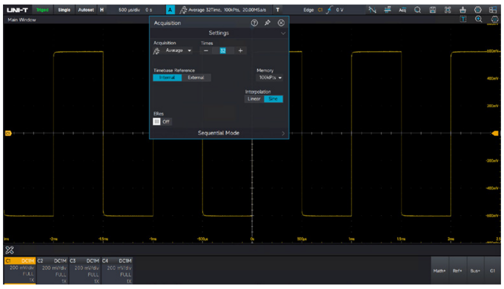
(4) Average Sampling

In this mode, the oscilloscope acquires multiple waveforms, calculates their average values, and displays the final waveform. Averaging helps reduce random noise and provides a clearer waveform representation. The maximum average time is 65, 536.

By adjusting the acquisition mode settings to observe the resulting changes in the waveform display. When significant noise is present in the signal, the figure below shows the sampled waveform without averaging and the waveform with 32-time averaging applied.



Not Averaged Waveform

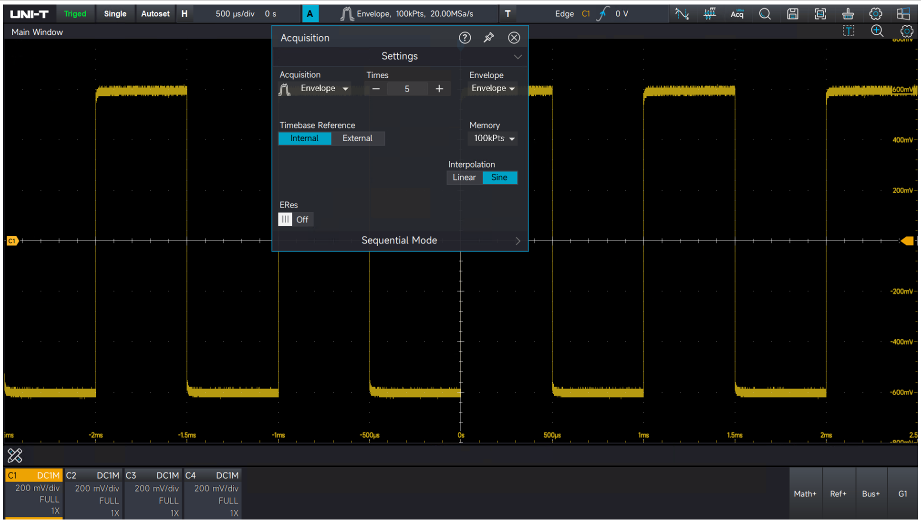


Waveform of 32 Times Averaged

Note: Average and high-resolution use different averaging methods. The average mode uses "multiple sampling averaging," while the high-resolution mode uses "single sampling averaging."

(5) Envelope

The envelope mode collects multiple waveforms and calculates the maximum and minimum values for all sampling points at the same moment relative to the trigger point. Typically, the envelope mode utilizes peak detection for each individual acquisition. The maximum number of envelope operations is 65,536.



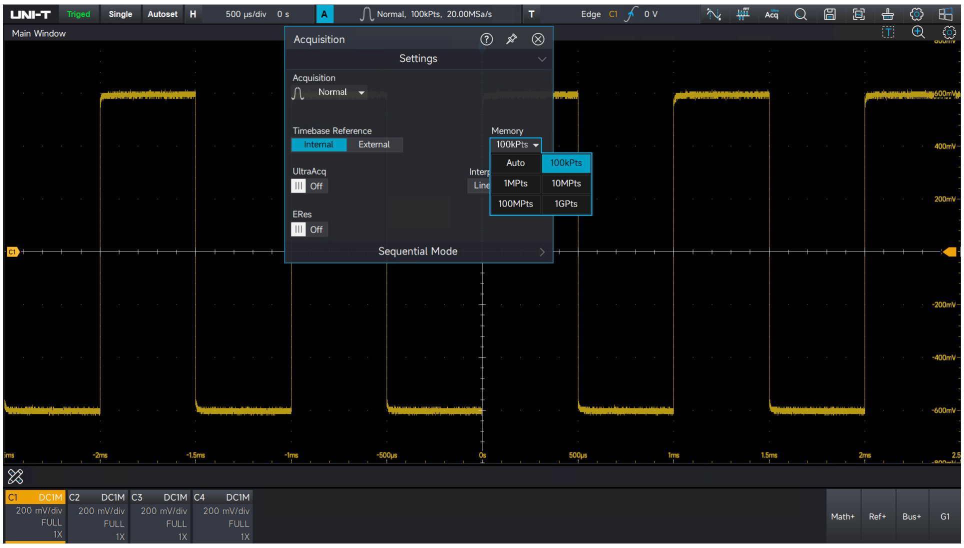
4.3 Memory Depth

The memory depth is the number of waveforms that can be stored in the oscilloscope during a trigger acquisition. It reflects the memory storage capacity of the memorizer.

The relation of the memory depth, sampling rate, and waveform length:

Memory depth=Sampling rate×Horizontal time base× Number of grid in the horizontal direction

The maximum memory depth of UPO7000L is 1Gpts (Single channel: 1Gpts; dual-channel: 500Mpts; four channel: 250Mpts.) In the "Sampling Setting → Memory Depth" menu, when the single channel is activated, the memory depth can be set to auto, 100Kpts, 1Mpts, 10Mpts, 100Mpts, and 1Gpts.



4.4 ERes

Unlike ERes in mathematics, ERes in acquisition is implemented by FPGA, which directly processes waveforms without converting them into mathematical models.

The filtering in Enhanced Resolution (ERes) mode for signal improvement has the following two characteristics:

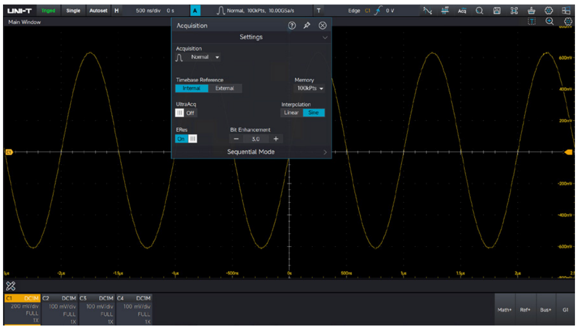
(1) In all cases, each filter uses a fixed value to improve the resolution (i.e., the ability to distinguish between closely spaced voltage levels). This effectively enhances resolution regardless of whether the signal contains noise, is a single-shot signal, or is a repetitive signal.

(2) The Signal-to-Noise Ratio (SNR) can also be improved, depending on the noise characteristics in the original signal. The Enhanced Resolution (ERes) mode reduces the signal bandwidth, allowing certain noise components to be filtered out.

Set ERes Mode

Select "ERes" from the acquisition menu, and set the enhance bit (0.5 to 3bits).

ERES OFF



ERES (Enhance 3bits)

5. Triggering System

Trigger determines when the oscilloscope starts to collect data and displays waveform. Once the trigger is correctly set, it can convert unstable signals into stable waveform. In the beginning of data acquisition, it collects enough data to compose the waveform starting at the left of the trigger point and continues until the trigger condition is met. When a trigger is detected, the oscilloscope will continuously collect enough data to draw the waveform to the right of the trigger point.

5.1 Noun Explanation of Triggering System

Trigger

The trigger sets specific trigger conditions based on user requirements. When a segment of the waveform meets the specified condition, the oscilloscope instantly captures that segment along with its neighboring parts and displays them on the screen.

The trigger serves two main purposes: isolating the event of interest and synchronizing the waveform to ensure a stable display. Only a steady trigger can produce a stable display.

The trigger circuit ensures that each time-base scan or acquisition starts when the input signal satisfies the user-defined trigger condition. This ensures that each scan or acquisition aligns with the captured waveform, thereby stabilizing the waveform display.

Trigger Source

A signal is used to generate a trigger. Triggers can be obtained from a variety of sources, such as analog channel (C1, C2, C3, and CH4) C4), Ext, Ext5, and AC.

1. Analog channel: Select one of the analog signal input terminals (C1–C4) on the front panel of the oscilloscope as the trigger signal.
2. EXT: Select the EXT TRIG input terminal on the front panel of the oscilloscope as the trigger signal. For example, an external clock can be inputted into the EXT TRIG terminal to serve as a trigger source. The trigger level range for the EXT signal is -1V to +1V.
3. AC: The trigger signal is derived from the AC power input of the oscilloscope. The mains supply trigger is typically used to measure signals related to the frequency of the AC power supply. It is primarily used in the power industry.

Trigger Mode

Trigger Mode determines the behavior of the oscilloscope when a trigger condition is met. This oscilloscope provides three trigger modes: Auto, Normal, and Single.

(1) Auto trigger: When there is no trigger signal, the system will automatically collect data and display it. Once a trigger signal is generated, the system will automatically switch to trigger scanning and synchronize with the signal.

Auto trigger mode is suitable for:

- Checking DC signals or signals with unknown level characteristics.

(2) Normal trigger: The oscilloscope will only collect data when the trigger condition is met. When there is no trigger signal, the oscilloscope stops collecting data and enters a waiting state for a trigger. Once the trigger condition is met, the oscilloscope will refresh the waveform data. Otherwise, the oscilloscope maintains the last triggered waveform.

Normal trigger mode is suitable for:

- Collecting only the particular event specified by the trigger settings.
- Rare trigger events. Normal mode can prevent the oscilloscope from automatically triggering, allowing the waveform to display stably.

(3) Single Trigger: In single trigger mode, pressing the Single key once clears the waveform on the screen, and the oscilloscope enters a waiting state for the trigger. Once a trigger is detected, the waveform is sampled and displayed, and the oscilloscope enters the STOP state.

Single trigger mode is suitable for:

- Capturing random events or aperiodic signals, such as rising or falling electrical waveforms.
- Rare trigger events.

Trigger Coupling

Trigger coupling determines which part of the signal is transmitted to the trigger circuit. The coupling types include DC, AC, LF rejection, HF rejection, and noise suppression.

- DC: Allows all components of the signal to pass through.
- AC: Blocks the DC component of the signal.
- HF rejection: Attenuates high-frequency components above 40kHz.
- LF rejection: Attenuates low-frequency components below 40kHz.
- Noise suppression: Suppresses high-frequency noise in the signal to reduce the probability of triggering errors.

Pre-trigger/Delay Trigger

This feature allows the oscilloscope to collect data before or after a trigger event.

The trigger position is typically set at the horizontal center of the screen. The user can observe 5div grid sections of pre-trigger and delay trigger information. The user can move the waveform horizontally to view additional pre-trigger information. By examining the pre-trigger data, the waveform before the event can be observed. For example, capturing a glitch at the start of a circuit and analyzing the pre-trigger data can help identify the cause of the glitch.

Force Trigger

Press the Force key to manually generate a trigger signal.

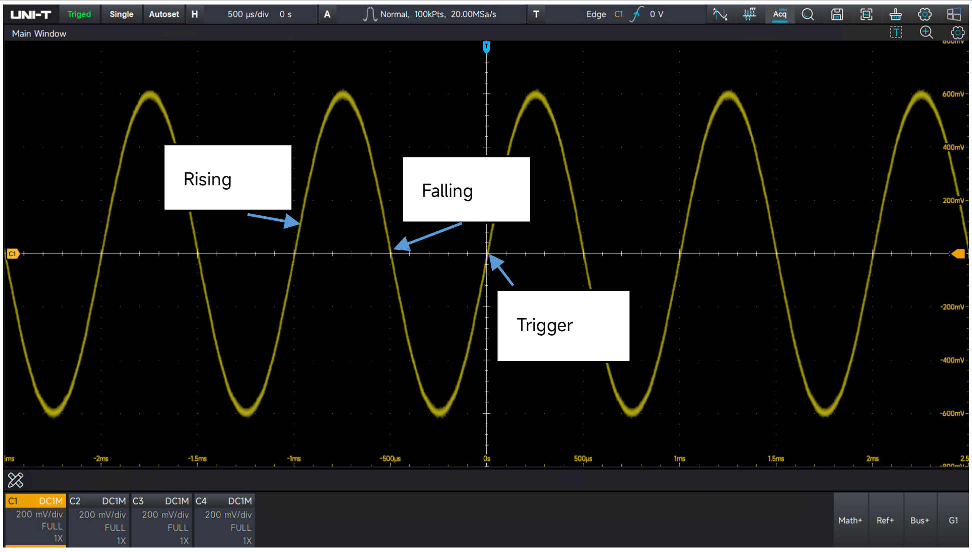
If the waveform is not displayed on the screen in normal or single trigger mode, press the Force key to sample the signal baseline and confirm whether the sampling is performed correctly.

Trigger Holdoff

Trigger holdoff helps to stably capture complex and repeating waveforms (such as waveforms with multiple edges or other events between them, like pulse strings). The trigger holdoff time specifies how long the oscilloscope waits before restarting the trigger circuit. During the trigger holdoff, even if the trigger condition is met, the oscilloscope will not trigger until the holdoff time has expired.

5.2 Edge Triggering

The edge can be triggered by detecting a specific edge (rising edge, falling edge, or arbitrary edge) of the waveform and its electrical level. Press the edge trigger menu to set the source, trigger coupling, trigger mode, and edge type. The waveform will be stably generated when the condition is satisfied.



Select Source

The edge trigger can select from C1-C4, EXT, EXT/5, or mains supply.

For more details, please refer to the "Noun Explanation of Triggering System"section.

Select Edge Type

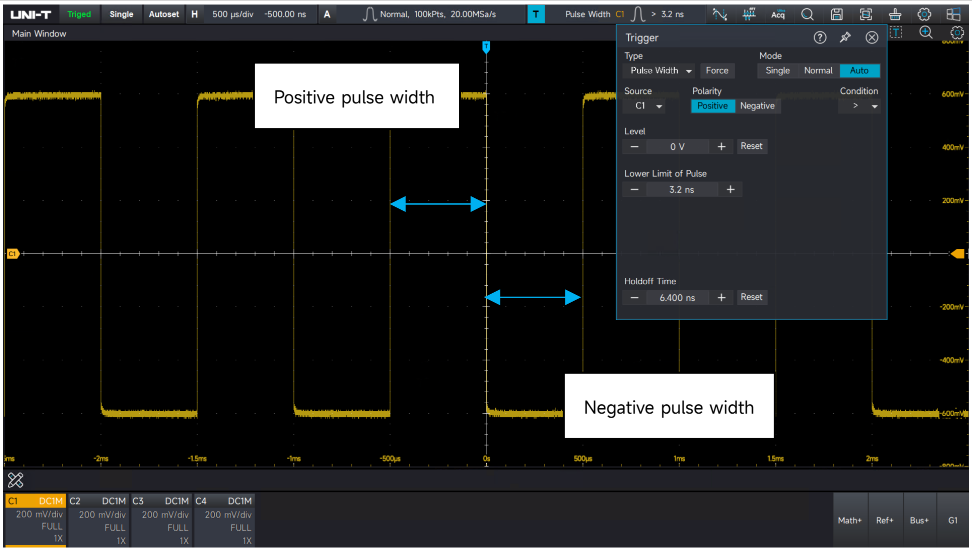
1. Rising edge: Set to trigger on the rising edge of the signal.
2. Falling edge: Set to trigger on the falling edge of the signal.
3. Arbitrary edge: Set to trigger on both the rising and falling edges of the signal.

Set Threshold Level

Set the position of the edge trigger level: Click on the trigger level cursor with the mouse and drag it up or down to adjust the trigger level position, or use the numeric keyboard or external physical keyboard to enter specific values to move the trigger level to a precise location.

5.3 Pulse Width Triggering

The pulse width trigger sets the oscilloscope to trigger on a positive or negative pulse with a specified pulse width that meets the condition criteria. The pulse width trigger menu allows the user to set the source, pulse width condition, upper and lower limits of pulse width, pulse width polarity (positive/negative), trigger coupling, and trigger mode.



Pulse Width Condition: Select the trigger condition to “>”, “<”, or “[…]”.

>： When the pulse width of the trigger signal is greater than the set pulse width, the lower limit of the pulse width can be set.

<： When the pulse width of the trigger signal is less than the set pulse width, the upper limit of the pulse width can be set.

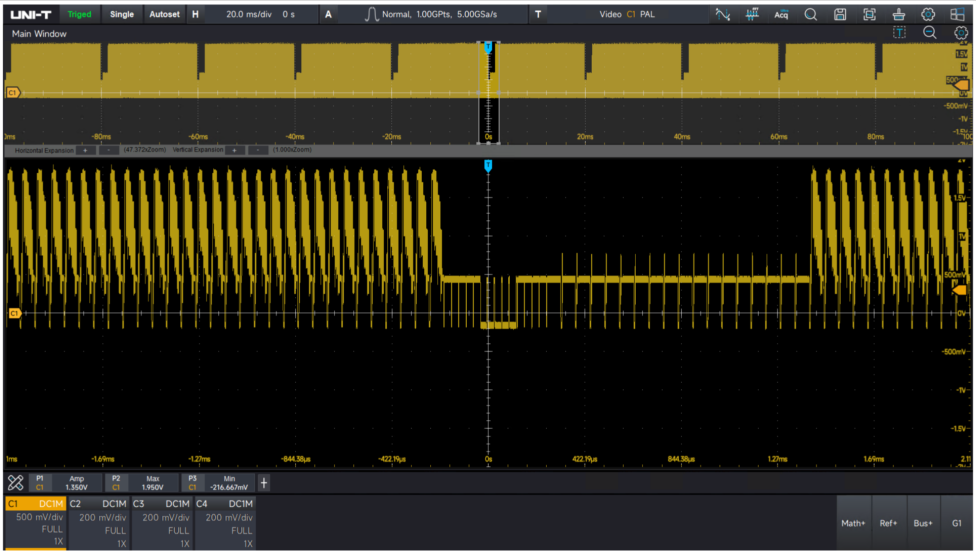
[…]： When the pulse width of the trigger signal is approximately equal to the set pulse width, or the signal's pulse width falls within the set range, both the lower and upper limits of time can be set.

Upper/Lower Limit of Pulse Width

The pulse width value of the set pulse is compared with the signal's pulse width. The trigger will be activated if the condition is met. The range can be set from 3.2ns to 10s.

5.4 Video Triggering

The video signal includes both image and time sequence information, and it has multiple standards and formats. UPO7000L can trigger on the field or line of NTSC (National Television Standards Committee) and PAL (Phase Alternating Line).



Video Format

PAL: The frame rate is 25 frames per second, with 625 TV scan lines. The odd field comes first, followed by the even field.

NTSC: The field rate is 60 fields per second, with 30 frames per second. The TV scan lines total 525, with the even field coming first, followed by the odd field.

Video Synchronization

Even field: Set to trigger and synchronize on the even field of the video signal.

Odd field: Set to trigger and synchronize on the odd field of the video signal.

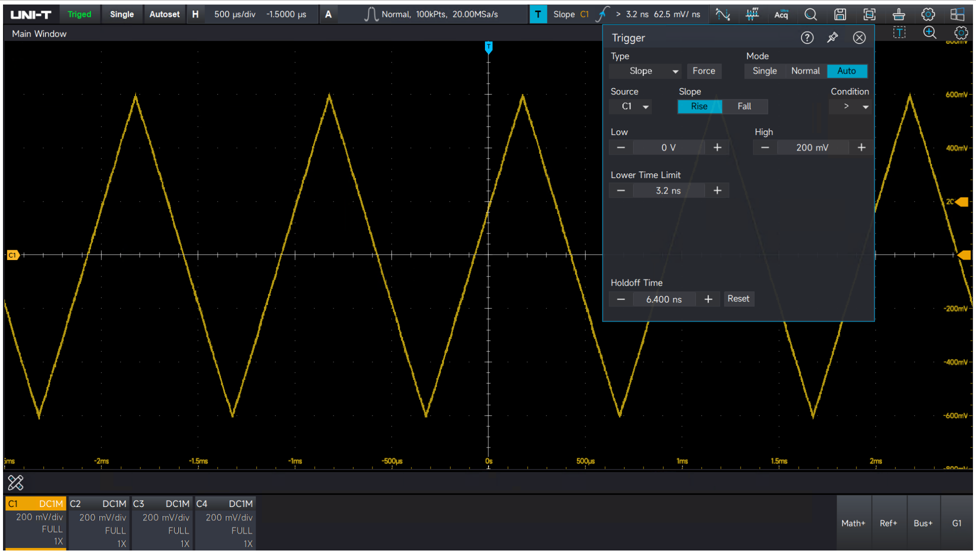
All lines: Set to trigger and synchronize on the entire video signal's line.

Specified line: Set to trigger and synchronize on a specific video line. When the specified line is selected, the line number can be assigned. The line number can be adjusted by − and + icons or using the numeric keypad. The range of line numbers is from 1 to 625.

Note: To observe the waveform details in the video signal, the user can increase the memory depth slightly. UPO7000L series utilizes UNI-T's original digital 3D technology, which features a multi-level greyscale display function. This allows different brightness levels to reflect the frequency of different parts of the signal. Experienced users can quickly assess the signal quality during the debugging process and identify any unusual conditions.

5.5 Slope Triggering

The slope trigger generates a trigger when the rising or falling slope of the signal conforms to the set value. The slope trigger menu allows you to set the source, trigger coupling, trigger mode, edge type (rising edge or falling edge), condition, high/low levels, and duration time.



Slope Type

Select the trigger edge for the slope trigger.

Rising edge: Perform slope trigger by using the rising edge of the trigger signal.

Falling edge: Perform slope trigger by using the falling edge of the trigger signal.

Condition

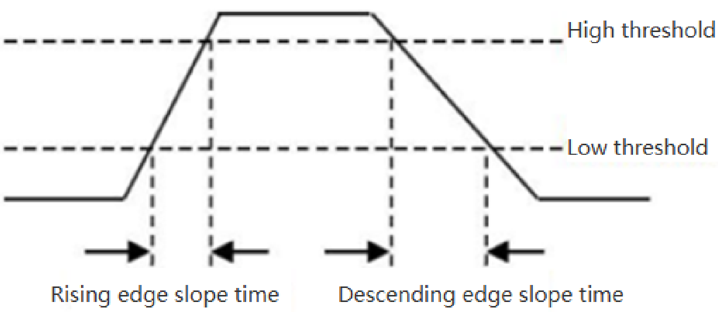
Select the trigger condition to “>”, “<”, or “[...]”.

>: The trigger will occur when the slope duration of the signal exceeds the set duration time. The lower limit of the duration time can be configured.

<: The trigger will occur when the slope duration of the signal is less than the set duration time. The upper limit of the duration time can be configured.

[...]: The trigger will occur when the slope duration of the signal approximately matches the set slope duration or falls within the specified range. Both the lower and upper limits of the duration time can be configured.

Note: The slope duration of the trigger signal refers to the "slope duration of the rising edge" and the "slope duration of the falling edge," as illustrated in the following figure.



Select Level

Level can be set to low level, high level, and high-low level. Click the “Level” with the mouse to quickly switch these options.

Lower Limit of Time

The time can be set from 3.2ns to 10s.

Note: In slope trigger mode, the set duration time and high-low threshold values are displayed in the top right corner of the screen.

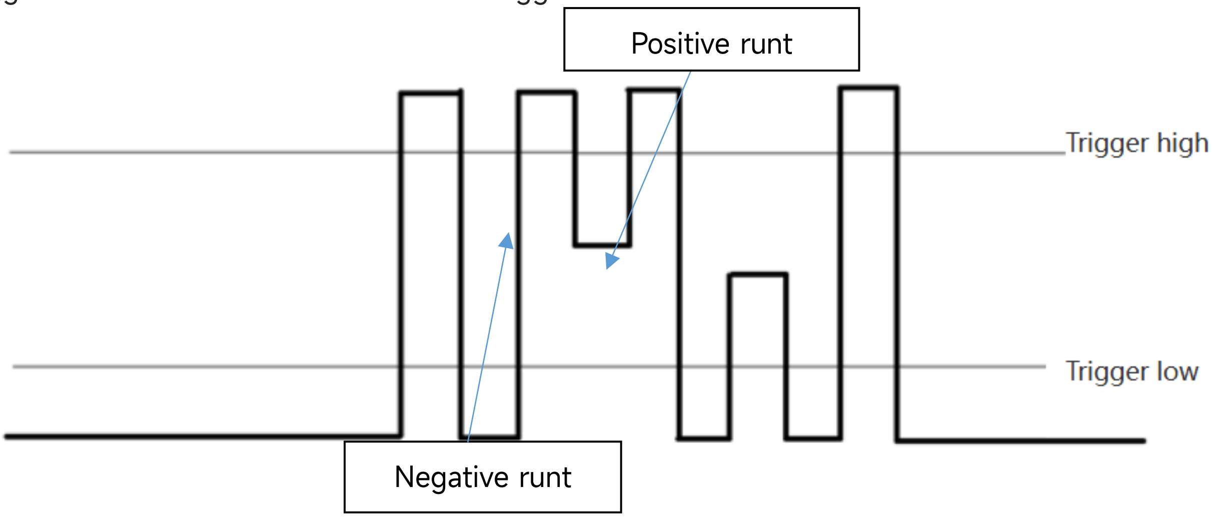
Slew Rate

The calculation formula of slew rate value: **(High level threshold - low level threshold) ÷ Time**

For the set slew rate, the time refers to the slope time value specified for the measurement.

5.6 Runt Pulse Triggering

The runt pulse trigger is used to trigger a pulse that has crossed one trigger level but fail to reach the other. In this oscilloscope, the positive runt pulse is the pulse that crosses the low trigger level but does not cross the high trigger level; the negative runt pulse is the pulse that crosses the high trigger level but does not cross the low trigger level.



The runt trigger menu can set source, trigger coupling, trigger mode, trigger polarity (positive, negative), runt condition (irrelevance, <, >, [...]), low level, high level, and the upper/lower limit of pulse width.

Trigger Polarity

Positive pulse: Set to trigger on the positive runt pulse.

Negative pulse: Set to trigger on the negative runt pulse.

Runt Condition

Irrelevance: The trigger limit condition of the runt pulse trigger is not set.

＞: The trigger will occur when the runt pulse width exceeds the set pulse width. The lower limit of time can be configured.

＜: The trigger will occur when the runt pulse width is less than the set pulse width. The upper limit of time can be configured.

[...]: The trigger will occur when the runt pulse width within the set pulse width or falls within the specified range. Both the lower and upper limits of the time can be configured.

Upper/Lower Limit of Pulse Width

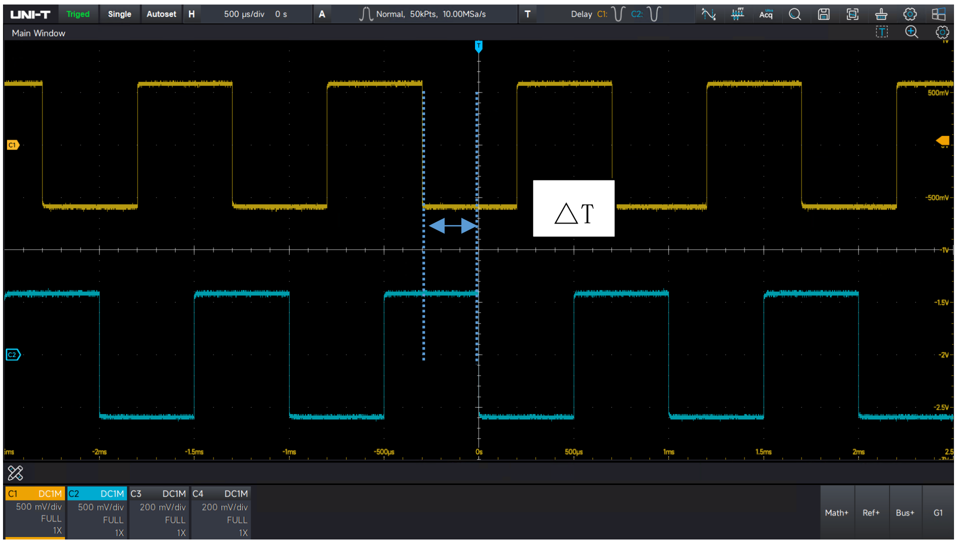
The pulse width value of the set pulse is compared with the channel pulse width. It will be triggered when the condition is met. The range can be set from 3.2ns to 10s.

Select Level

Level can be set to low level, high level, and high-low level. Click the "Level" with the mouse to quickly switch these options.

5.7 Delay Triggering

The delay trigger requires the configuration of Trigger Source 1 and Trigger Source 2. The oscilloscope generates a trigger when the time difference (ΔT) between the edge defined by Source 1 (Edge 1) and the edge defined by Source 2 (Edge 2) falls within the preset time limit, as shown in the following figure.



Edge 1 is set to the rising edge, and Edge 2 is also set to the rising edge. △T indicates the time range, **as marked in blue in the figure above**.

Note: Edge 1 and Edge 2 must be adjacent edges. A stable trigger can only be achieved if the channel is connected to an active signal.

Delay Condition

Select the delay trigger to "＞", "＜", "[...]", or "]...[".

>: When the time difference (ΔT) between the edge set by Source 1 and the edge set by Source 2 is greater than the lower time limit, the lower time limit can be configured.

<: When the time difference (ΔT) between the edge set by Source 1 and the edge set by Source 2 is less than the upper time limit, the upper time limit can be configured.

[…]: When the time difference (ΔT) between the edge set by Source 1 and the edge set by Source 2 is greater than or equal to the lower time limit and less than or equal to the upper time limit, both the upper and lower limits can be configured.

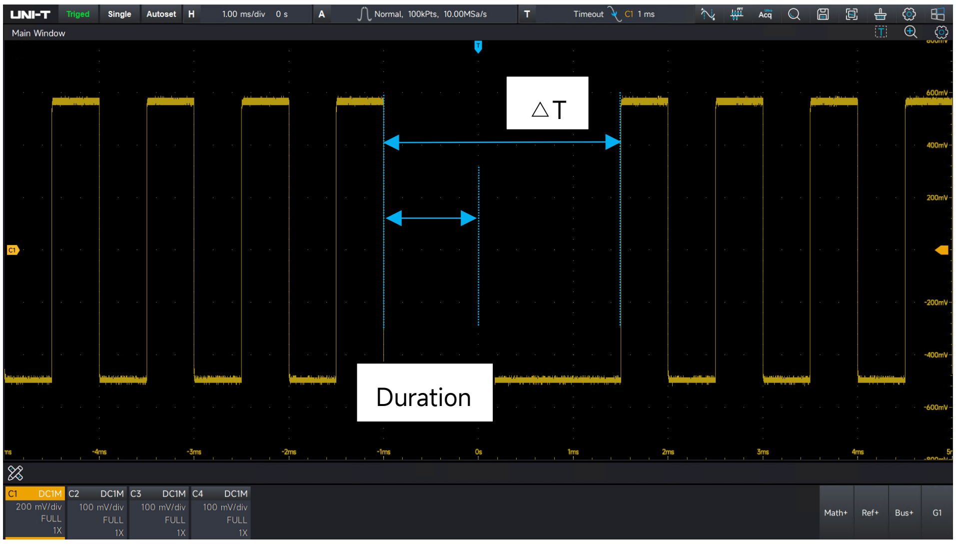
]…[: When the time difference (ΔT) between the edge set by Source 1 and the edge set by Source 2 is less than the lower time limit or greater than the upper time limit, both the upper and lower limits can be configured.

Upper/Lower Limit of Time

The set time is compared with △T, it will be triggered when the condition is met. The range can be set from 3.2ns to 10s.

5.8 Timeout Triggering

The timeout triggering generates a signal when the time interval (ΔT) between the rising edge (or falling edge) of the input signal crossing the trigger level and the adjacent falling edge (or rising edge) crossing the trigger level exceeds the set duration, as shown in the following figure.



△T> Duration Triggering

Edge Type

Select the edge of the input signal to trigger the oscilloscope. The edge type can be set to rising edge, falling edge, or arbitrary edge. The current edge type is displayed in the top right corner of the screen.

Rising edge: The timer starts when the rising edge of the input signal crosses the trigger level.

Falling edge: The timer starts when the falling edge of the input signal crosses the trigger level.

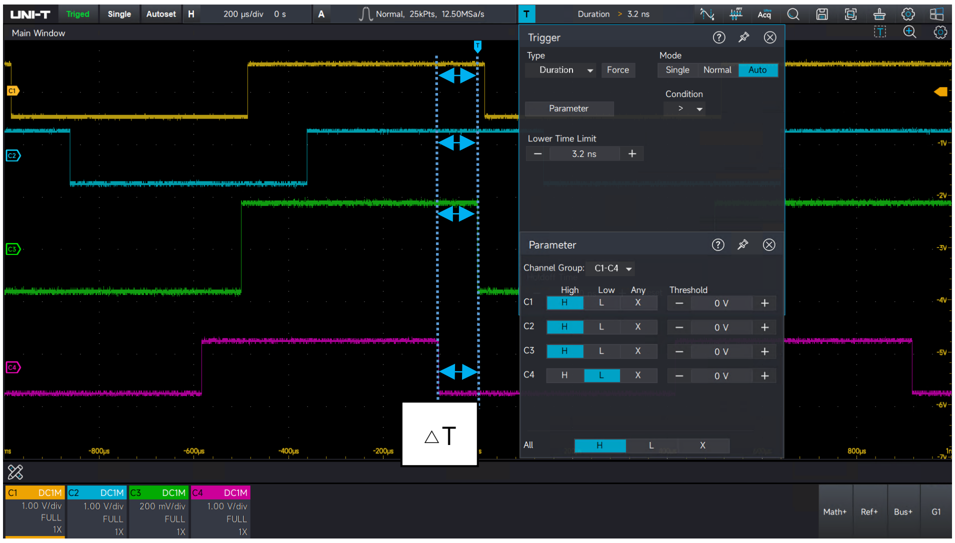
Arbitrary edge: The timer starts when either the rising edge or the falling edge of the input signal crosses the trigger level.

Duration

The set duration is compared with △T, it will be triggered when duration < △T. The time can be set from 3.2ns to 10s.

5.9 Duration Triggering

When the duration trigger is selected, the oscilloscope identifies the trigger condition by analyzing the duration of the specified code pattern. The code pattern is a combination of channel logic using the "AND" operation, with each channel having a value of H (high), L (low), or X (Arbitrary). The trigger is activated when the duration (△T) of the code pattern matches the preset time, as shown in the following figure.



Code Pattern

Click the parameter setup to open the configuration window, where the code pattern can be set to H (high), L (low), or X (Arbitrary). The threshold is used to determine the voltage levels for H, L, and X.

H: Set the code pattern value of the selected channel to "High," meaning the voltage level is higher than the trigger level of the channel.

L: Set the code pattern value of the selected channel to "Low," meaning the voltage level is lower than the trigger level of the channel.

X: Set the code pattern value of the selected channel to "X," meaning the channel is not part of the pattern. The oscilloscope will not trigger if all channels in the code pattern are set to "X."

Trigger Condition

The trigger condition can be set to" > ", " < ", or "[…]".

> : When the duration of code pattern is greater than the lower limit of time, the lower limit of time can be configured.

< : When the duration of code pattern is less than the upper limit of time, the upper limit of time can be configured.

[…]: When the duration of code pattern is less than or equal to the upper limit of time and greater than or equal to the lower limit of time, both the upper/lower limit of time can be configured.

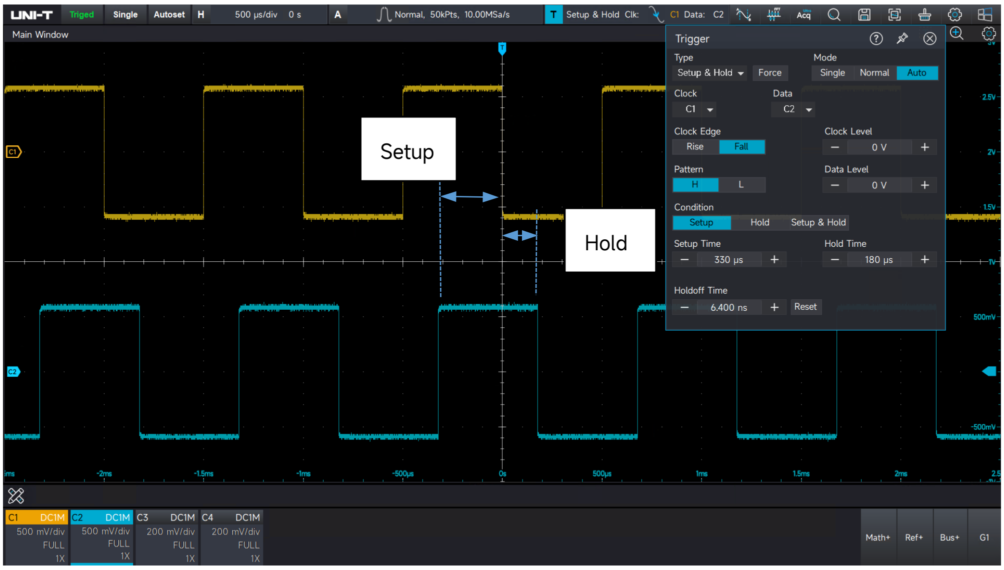
Upper/Lower Limit of Time

The duration of code pattern is compared with △T, it will be triggered when the condition is met. The range can be set to from 3.2ns to 10s.

5.10 Setup & Hold Triggering

In the setup & hold trigger, the oscilloscope requires the selection of both the data source and clock source. The setup time begins when the data signal crosses the trigger level and ends when the specified clock edge arrives. The hold time begins when the specified clock edge arrives and ends when the data signal crosses the trigger level again. The oscilloscope will trigger when either the setup time or the hold time is shorter than the preset time.

This feature is primarily used to locate errors and quickly identify signals that fail to meet the setup & hold time requirements.



Code Pattern

H (high): Set the valid code pattern of the data signal to high-level.

L (low): Set the valid code pattern of the data signal to low-level.

Edge Type

Rising edge: Set the clock edge type to rising edge.

Falling edge: Set the clock edge type to falling edge.

Hold Type

Setup: It will be generated when the setup time is less than the set value.

Hold: It will be generated when the hold time is less than the set value.

Setup & Hold: It will be generated when the setup time or hold time is less than the set value.

Level

Clock level: Set the clock source to generate the trigger level.

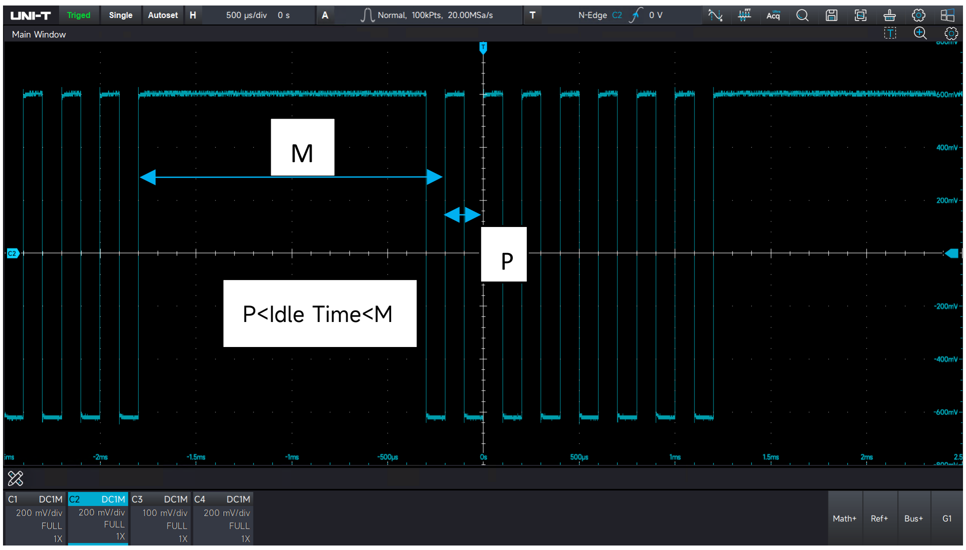
Data level: Set the data source to generate the trigger level.

Time

The setup time or hold time △T of code pattern is compared with the set time, it will be triggered when the condition is met. The range can be set from 3.2ns to 10s.

5.11 Nth Edge Triggering

The Nth edge trigger is triggered on the Nth edge after assigning the specified idle time. For example, the waveform as shown in the following figure, it is set to trigger on the 2nd rising edge after the specified idle time (the time between two adjacent rising edge), then set the idle time to P< idle time <M. M is the time between the 1st rising edge and the next rising edge, P is the maximum time between the counting rising edge, as shown in the following figure.



Edge Type

Select the edge of the input signal that will trigger the oscilloscope. The edge type can be set to rising edge or falling edge. The current edge type will be displayed in the top right corner of the screen.

Rising edge: Set to trigger on the rising edge of the signal.

Falling edge: Set to trigger on the falling edge of the signal.

Idle Time

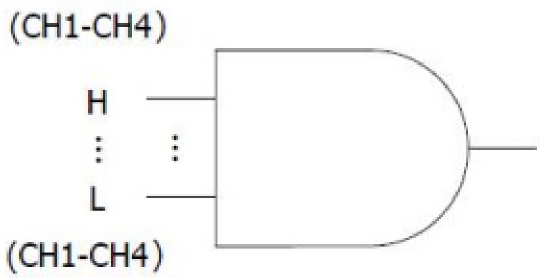
The idle time is compared with the pulse time, it will be triggered when the condition is met. The range can be set from 3.2ns to 10s.

Edge Count

The number of edges specifies at which edge of the pulse string the trigger will occur. Press the − and + icons or use the numeric keyboard to set the edge value. The range of edges is from 1 to 65535.

5.12 Code Pattern Triggering

The code pattern trigger identifies the trigger condition by matching the specified code patterns. The pattern trigger type is based on the logical combination of the channels using the "AND" operator. Each channel can be set to H (High), L (Low), or X (Arbitrary). Users can also specify a channel in the code pattern as either a rising edge or falling edge (only one edge can be selected). When an edge is assigned, the oscilloscope will trigger on the specified edge if the code patterns of the other channels are judged to be "true" (i.e., the actual pattern matches the preset pattern type). If no edge is assigned, the oscilloscope will trigger on the last edge of the "true" code pattern. If all channels are set to “Arbitrary”, the oscilloscope will not trigger.



Code Pattern

The code pattern can be set to H (High), L (Low), or X (Arbitrary). The code pattern for each channel will be displayed at the bottom of the screen.

H: Set the code pattern value of the selected channel to “High”, that is, the voltage level is higher than the trigger level of the channel.

L: Set the code pattern value of the selected channel to "Low", that is, the voltage level is lower than the trigger level of the channel.

X: Set the code pattern value of the selected channel to "X", that is, the channel is not part of the pattern. The oscilloscope will not be trigger if all channels in the code pattern are set to "X".

R: Set the code pattern to the rising edge of the selected channel.

F: Set the code pattern to the falling edge of the selected channel.

5.13 Serial Bus Triggering

UPO7000L supports 11 types of serial bus triggering, data triggering, envelope triggering, and includes an event list and search function.

Software Suite	Description	Option	Standard/Option
Computer serial bus triggering analysis	RS-232/422/485/UART	-	Standard
Embedded serial bus triggering analysis	I^2C, SPI	-	Standard
Automobile serial bus triggering analysis	CAN, LIN	-	Standard
Automobile serial bus triggering analysis	CAN-FD	UPO7000L-CANFD	Option
Automobile serial bus triggering analysis	FlexRay	UPO7000L-FLEX	Option
Automobile serial bus triggering analysis	SENT	UPO7000L-SENT	Option
Audio serial bus triggering analysis	I^2S, LJ, RJ, TDM	UPO7000L-AUDIO	Option
Aerospace serial bus triggering analysis	MIL-STD-1553, ARINC 429	UPO7000L-AREO	Option

Close

Close the serial bus triggering function.

RS232

RS23S decoding triggering can configure trigger mode, data, and decoding parameters.

(1) Trigger mode: Set the trigger mode to start bit, stop bit, parity check bit, or data bit for RS232.

Start bit: The waveform will be triggered at the start bit of the RS232 protocol. This trigger can be used when a single string or repeated strings are sent, allowing users to observe a stable signal waveform. If the sent data changes, the corresponding waveform will also change.

Stop bit: The waveform will be triggered at the stop bit of the RS232 protocol.

Parity check bit: When the RS232 protocol includes a parity check bit, it should be set to 0 or 1 based on the odd-even parity check.

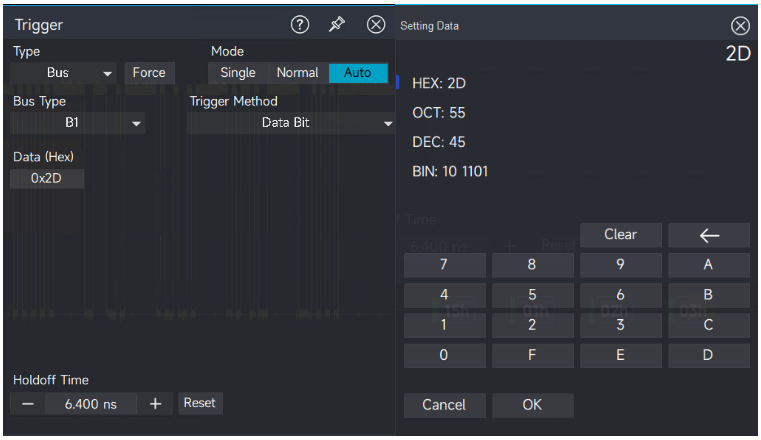
- Odd parity check: If the number of '1' bits in the data bits and the parity bit is odd, the transmission is considered correct.
- Even parity check: If the number of '1' bits in the data bits and the parity bit is even, the transmission is considered correct.

The odd-even check in RS232 communication helps quickly identify errors in the transmission procedure, making it easier to locate and analyze faults.

Data bit: A trigger will occur when the data acquired by the oscilloscope matches the 2-bit hexadecimal value set by the user. This option allows the user to quickly identify and capture transmission signals for specific data of interest.

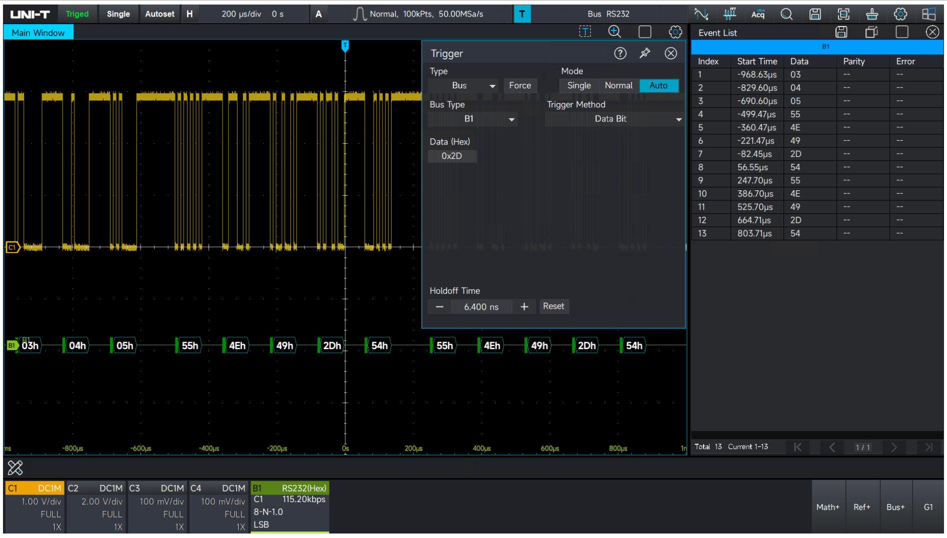
(2) Data

It will be valid when the trigger mode is set to “Data bit”. In this case, the trigger data can be configured.



(3) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "RS232" section in the protocol decode chapter.



I²C

I2C decoding triggering can configure trigger mode, data, trigger coupling, and decoding parameters.

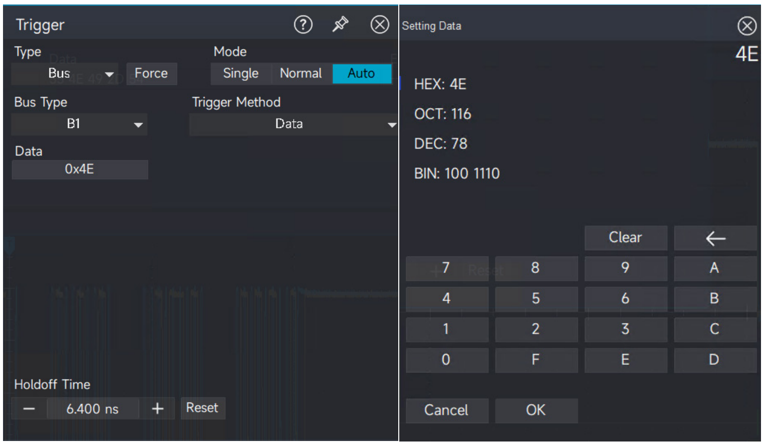
(1) Data direction

- Write: It will be triggered when the "read/write" bit in the I²C protocol is set to "Write".
- Read: It will be triggered when the "read/write" bit in the I²C protocol is set to "Read".

(2) Trigger mode: Set the trigger mode to start bit, restart, stop, response failed, address, data, or address and data for I²C.

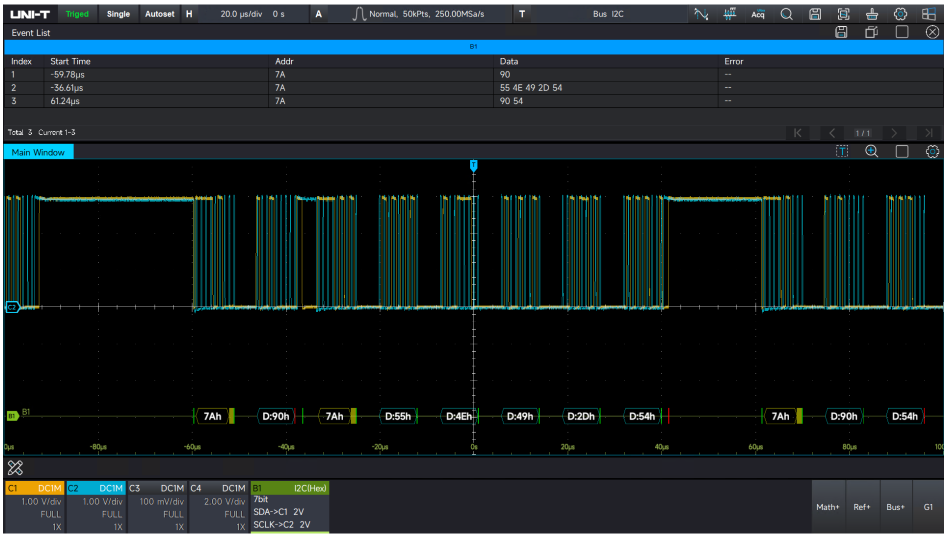
- Start bit: Triggered when I²C begins transmission, i.e., when SCL is high and a falling edge occurs on the SDA signal.
- Restart: Triggered when a start signal is immediately followed by another start signal before a stop signal occurs.
- Stop: Triggered when the stop bit occurs, i.e., when SCL is high and SDA transitions from low to high.
- Response failed: In the I²C protocol, after each 8-bit message, the receiver is expected to send an acknowledgment signal (ACK bit). This trigger is activated when SCL is high and SDA is also high, indicating a failed response.
- Address: Triggered when the communication address matches the user-defined value, helping to quickly locate address transmissions.
- Data: Triggered when the data acquired by I²C matches the user-defined value, allowing users to find the specific data transmission of interest.
- Address data: Triggered when both the address and data match the user-defined conditions, enabling easy implementation of specific address and data triggers in I²C for

detailed transmission analysis.



(3) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "I²C" section in the protocol decode chapter.



SPI

SPI decoding triggering can configure the trigger mode, data channel, data values, trigger coupling, frame size, and decoding parameters.

(1) Trigger mode

- Valid chip: It will be triggered at the edge where the chip level jumps from invalid to available.
- Data: The waveform will be triggered when the data acquired by SPI matches the user-defined value. This helps the user quickly locate the transmission signal of interest.

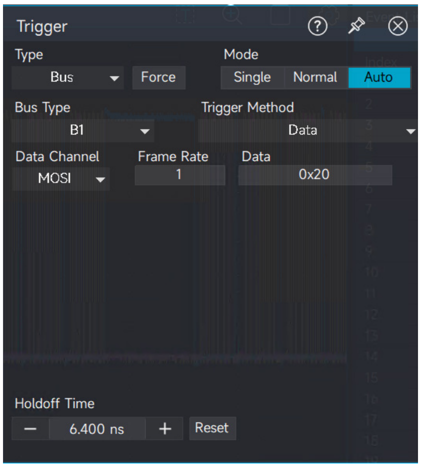
(2) Data channel

- MOSI: Data transfer between the master and slave devices, where data is output from the master device and input into the slave device.

- MISO: Data transfer between the master and slave devices, where data is input from the master device and output from the slave device.

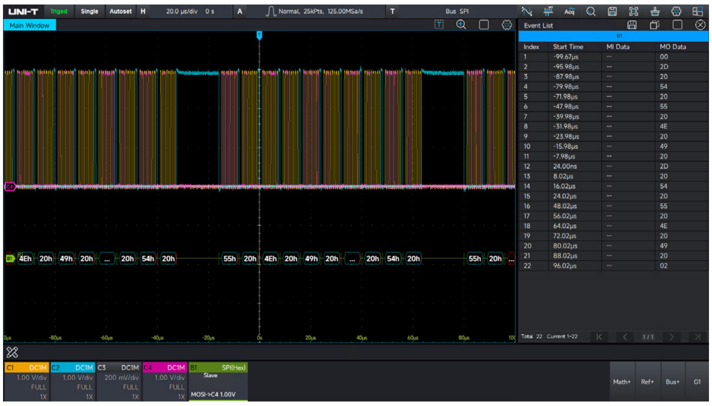
(3) Frame size

Set the data frame size, with a range of 1 to 8.



(4) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "SPI" section in the protocol decode chapter.



CAN

CAN decoding triggering can configure the trigger mode, ID standard trigger, ID, frame direction, data, and decoding parameters.

(1) Trigger mode

Set the start frame, frame type (data frame, remote frame, error frame, overload frame), ID, data, ID and data, end of frame, or error (response failed, bit filling error) for CAN.

- Start frame: Trigger the waveform at the start of the frame.
- Frame type: Data frames, remote frames, error frames, overload frames.

- Data frame: The oscilloscope will generate a waveform on the data frame that matches the CAN signal.
- Remote frame: The oscilloscope will generate a waveform on the remote frame of the CAN signal.
- Error frame: The oscilloscope will generate a waveform on the error frame of the CAN signal.
- Overload frame: The oscilloscope will generate a waveform on the overload frame of the CAN signal.
- ID: The oscilloscope will generate a waveform on the specified ID.
- Data: The oscilloscope will generate a waveform on a data frame that matches the set data condition.
- ID and data: The oscilloscope will generate a waveform on the specified ID and data frame that matches the set data condition.
- End of frame: The oscilloscope will generate a waveform at the end of the CAN signal frame.
- Error: Response failed, bit filling error
- Response failed: Occurs when the transmitting node sends data and then performs a readback, detecting an error if the data on the bus differs from the sent data.
- Bit filling error: The oscilloscope will trigger when a bit stuffing error is detected in a segment that requires bit filling, specifically when six consecutive bits of the same level are detected.

(2) ID standard

The trigger condition is valid when the trigger mode is set to “ID” or “ID & Data”. The format can be selected between the “Standard Format” and “Extended Format”, with different ID ranges for each format.

(3) ID

The trigger condition is valid when the trigger mode is set to “ID” or “ID & Data”. The ID range is from 0 to 2047.

(4) Frame direction

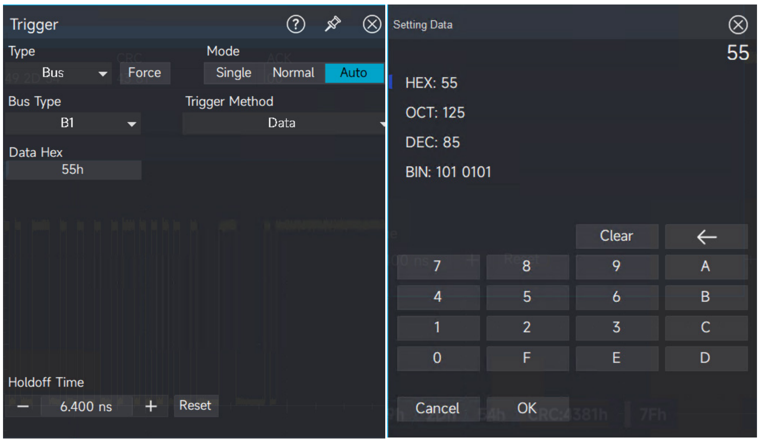
This parameter is valid if the trigger condition is set to ID or ID and Data, and you can set Write, Read, or Any.

- Write: The waveform will be triggered when the “read/write” bit of the CAN protocol is set to “Write”.

- Read: The waveform will be triggered when the "read/write" bit of the CAN protocol is set to "Read".
- Arbitrary: The waveform will be triggered when the "read/write" bit of the CAN protocol is set to either "Write" or "Read".

(5) Data

The user can set the data condition to trigger the waveform. This is valid when the trigger mode is set to "Data" or "ID & Data". The number of bytes of data can be set in relation to the value of the bytes, and it can be set in the range from 0 to 255.



(6) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "CAN" section in the protocol decoding chapter.

CAN_FD

CAN_FD decoding triggering can configure the trigger mode, trigger coupling, data, frame type, frame direction, error, and decoding parameters.

(1) Trigger mode

Set the start frame, frame type (data frame, remote frame, rate frame, error frame, overload frame), ID, data, ID and data, end of frame, or error (response failed, bit filling error) for CAN-FD.

- Start frame: Trigger the waveform at the start of the frame.
- Frame type: Data frames, remote frames, variable speed frames, error frames, overload frames.
- Data frame: The oscilloscope will generate a waveform on the data frame that matches the CAN_FD signal.
- Remote frame: The oscilloscope will generate a waveform on the remote frame of the CAN_FD signal.
- Rate Frame: The oscilloscope will generate a waveform when there is a change in the rate of the CAN_FD signal.
- Error frame: The oscilloscope will generate a waveform on the error frame of the CAN_FD signal.
- Overload frame: The oscilloscope will generate a waveform on the overload frame of the CAN_FD signal.
- ID: The oscilloscope will generate a waveform on the specified ID.
- Data: The oscilloscope will generate a waveform on a data frame that matches the set data condition.
- ID & Data: The oscilloscope will generate a waveform on the specified ID and data frame that matches the set data condition.
- End of frame: The oscilloscope will generate a waveform at the end of the CAN_FD signal frame.
- Error: Response failed, bit filling error
- Response failed: Occurs when the transmitting node sends data and then performs a readback, detecting an error if the data on the bus differs from the sent data.
- Bit filling error: The oscilloscope will trigger when a bit stuffing error is detected in a segment that requires bit filling, specifically when six consecutive bits of the same level are detected.

(2) ID standard

The trigger condition is valid when the trigger mode is set to “ID” or “ID & Data”. The format can be selected between the “Standard Format” and “Extended Format”, with different ID ranges for each format.

(3) ID

The trigger condition is valid when the trigger mode is set to "ID" or "ID & Data". The ID range is from 0 to 2047.

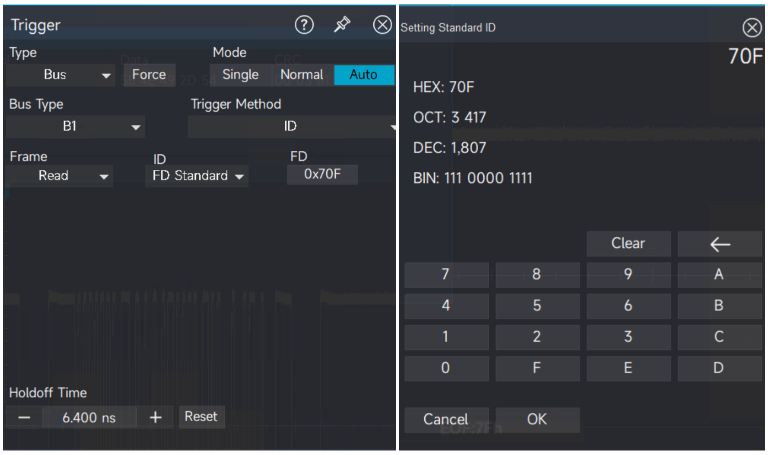
(4) Frame direction

This is valid when the trigger mode is set to "ID" or "ID & Data". You can set "Write, Read,Any".

- Write: The waveform will be triggered when the "read/write" bit of the CAN protocol is set to "Write".
- Read: The waveform will be triggered when the "read/write" bit of the CAN protocol is set to "Read".
- Arbitrary: The waveform will be triggered when the "read/write" bit of the CAN protocol is set to either "Write" or "Read".

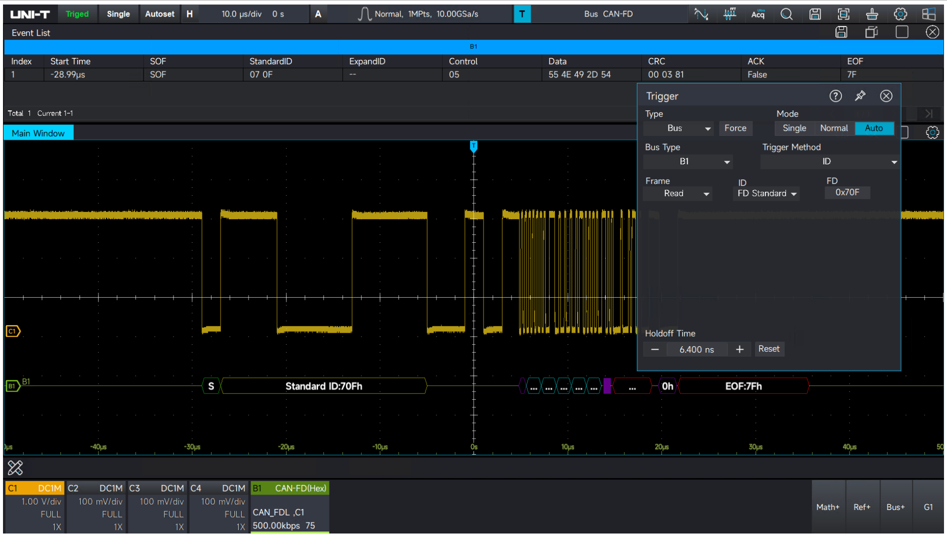
(5) Data

The user can set the data condition to trigger the waveform. This is valid when the trigger mode is set to "Data" or "ID & Data". The number of bytes of data can be set in relation to the value of the bytes, and it can be set in the range from 0 to 255.



(6) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "CAN-FD" section in the protocol decoding chapter.



LIN

LIN decoding triggering can configure the trigger mode, trigger coupling, and decoding parameters.

(1) Trigger mode:

Set the start frame, ID, data, ID and data, wake-up frame, sleep frame, or end of frame for LIN.

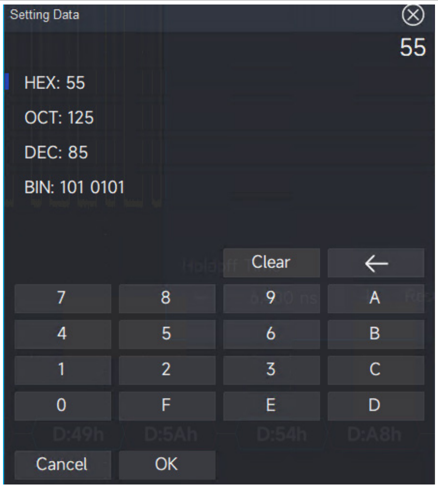
- Start frame: Trigger the waveform at the start of the frame.
- ID: The oscilloscope will generate a waveform on the specified ID.
- Data: The waveform will be triggered when the acquired data from the LIN signal matches the user-defined value, helping to quickly find the specific transmission signal of interest.
- ID & Data: The oscilloscope will generate a waveform on the specified ID and data frame that matches the set data condition.
- Wake-up frame: The oscilloscope will generate a waveform at the wake-up frame of the LIN signal.
- Sleep frame: The oscilloscope will generate a waveform at the sleep frame of the LIN signal.
- End of frame: The oscilloscope will generate a waveform at the end of the LIN frame.

(2) ID

The trigger condition is valid when the trigger mode is set to “ID” or “ID & Data”. The ID range is from 0 to 255.

(3) Data

The user can set the data condition to trigger the waveform. This is valid when the trigger mode is set to “Data” or “ID & Data”. The range can be set from 0 to 2047.



(4) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "LIN" section in the protocol decoding chapter.

FlexRay

FlexRay decoding triggering can configure the trigger mode, trigger coupling, and decoding parameters.

(1) Trigger mode

Set the frame header, indicating bit, ID, cycle number, data, ID and data, end of frame, or error for FlexRay.

- Frame header: The oscilloscope will generate a waveform at the transmission start sequence.
- Indicating bit: A waveform will be generated when the acquired information matches the specified indicating bit.

- ID: A waveform will be generated when the acquired data matches the set ID condition.
- Cycle number: A waveform will be generated when the acquired cycle number matches the specified cycle condition.
- Data: A waveform will be generated when the acquired data from the FlexRay protocol matches the user-defined condition, enabling quick identification of specific transmission signals of interest.
- ID & Data: A waveform will be generated when both the acquired ID and data match the specified conditions.
- End of frame: The oscilloscope will generate a waveform at the end of the frame.
- Error: The oscilloscope will be generated on bus error, including end of frame, empty frame, static frame, dynamic frame, synchronizing frame, and start frame error (no synchronization).

(2) Indicating bit

Set the indicating bit to normal (01XX), net load (11XX), empty frame (00XX), synchronization frame (XX10), or start frame (XX11) for the FlexRay protocol trigger.

(3) ID

The trigger condition is valid when the trigger mode is set to “ID” or “ID & Data”. The ID range is from 0 to 65535.

(4) Cycle numbers

The trigger condition is valid when the trigger mode is set to “cycle number” or “header field”. The range of cycle numbers can be set from 0 to 255.

(5) Data

The user can set the data condition to trigger the waveform. This is valid when the trigger mode is set to “Data” or “ID & Data”. The range can be set from 0 to 65535. Trigger data length: Set the byte length of the data to be triggered, and the range of "data" can be set differently for different byte lengths. Byte length range: 1~16.

(6) End of frame

The trigger condition is valid when the trigger mode is set to "end of frame". The frame type can be set to a static frame, dynamic frame, or all.

- Static state: It will be triggered on static frame.
- Dynamic state frame: It will be triggered on dynamic frame.
- All: It will be triggered on both static frames and dynamic frames.

(7) Error package

The trigger condition is valid when the trigger mode is set to “error”. The error package can be set to end of frame, empty static frame, empty dynamic frame, synchronizing frame, or start frame error (no synchronization).

- End of frame: Triggers on end-of-frame errors on the bus.
- Empty static frame: Triggers on empty static frames on the bus.
- Empty dynamic frame: Triggers on empty dynamic frames on the bus.
- Synchronization frame: FlexRay frame has a dedicated indicating bit in the frame header. The data frame is synchronization frame when the indicating bit is valid.
- Start frame (no synchronizing): FlexRay frame has a dedicated indicating bit in the frame header. The data frame is considered a start frame when the indication bit is valid.

(8) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "FlexRay" section in the protocol decoding chapter.

AudioBus

AudioBus decoding triggering can configure the trigger mode and decoding parameters.

(1) Trigger mode

Set the trigger mode for AudioBus.

Date: The waveform will be triggered when the data acquired through the AudioBus protocol matches the data condition set by the user. This feature helps users quickly locate specific transmission signal data of interest.

(2) Decoding parameter

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "AudioBus" section in the protocol decoding chapter.

MIL-STD-1553

MIL-STD-1553 decoding triggering can configure the trigger mode and decoding parameters.

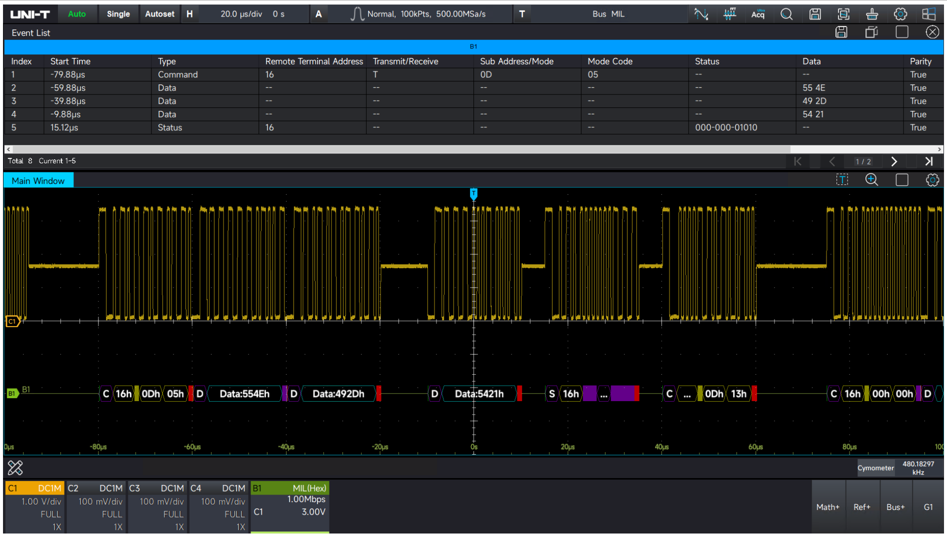
(1) Trigger mode

Set the command frame, data frame, state frame, or CRC error for MIL-STD-1553.

- Command frame: The oscilloscope triggers on the command frame of the MIL-STD-1553 signal.
- Data frame: The oscilloscope triggers on the data frame of the MIL-STD-1553 signal.
- State frame: The oscilloscope triggers on the status frame of the MIL-STD-1553 signal.
- CRC error: In an odd-even parity check, the oscilloscope triggers when the data within the character is incorrect.

(2) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "MIL-STD-1553" section in the protocol decoding chapter.



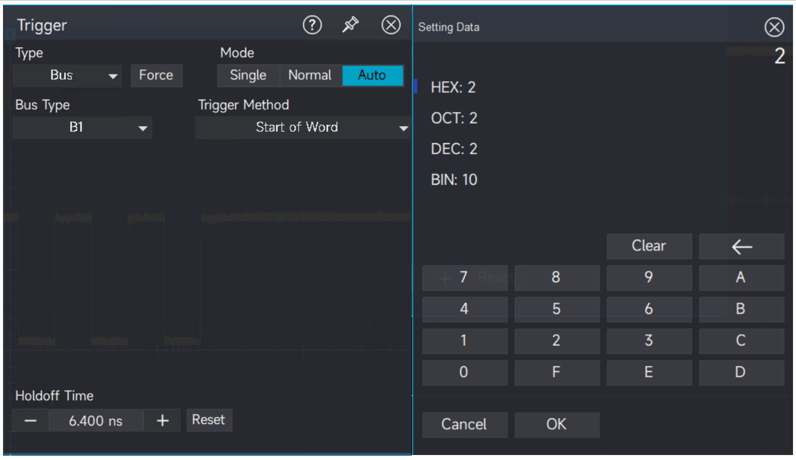
ARINC429

ARINC429 decoding triggering can configure the trigger mode and decoding parameters.

(1) Trigger mode

Set the start frame, source or destination identifier, data, indicating and state bit, or CRC error for ARINC429.

- Start frame: The oscilloscope triggers on the start bit of the ARINC429 signal sequence.
- Source or destination identifier: The oscilloscope triggers on the source or destination identifier bit, which specifies the origin and destination of the data.
- Data: The waveform triggers when the data acquired through the ARINC429 protocol match the user-defined data condition. This feature helps users quickly locate specific transmission signal data of interest.
- Indicating and state bit: The oscilloscope triggers when the indicating and state bit acquired through the ARINC429 protocol matches the user-defined SSM condition. SSM (Sign/Status Matrix) represents the symbolic state matrix, which specifies data properties such as direction (e.g., north or south), polarity (e.g., positive or negative), or state.
- CRC error: In an odd-even parity check, the oscilloscope triggers when the data within the character is incorrect.



(2) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "ARINC429" section in the protocol decoding chapter.

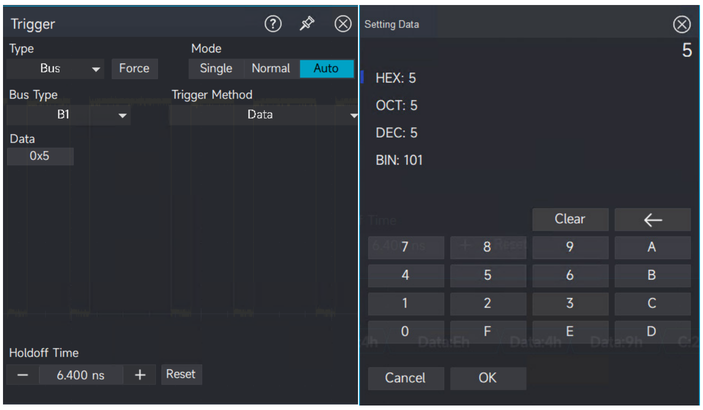
SENT

SENT decoding triggering can configure the trigger mode and decoding parameters.

(1) Trigger mode

Set synchronous bit, start frame, data, or CRC error for SENT.

- Synchronous bit: The oscilloscope triggers when it detects the synchronous pulse of the SENT signal, fixed at 56 ticks.
- Start frame: The oscilloscope will trigger on the start bit of the SENT signal sequence.
- Data: The oscilloscope triggers when the data acquired through the SENT protocol matches the user-defined data condition.
- CRC error: In an odd-even parity check, the oscilloscope triggers when the data within the character is incorrect.



(2) Decoding parameters

The trigger decode parameter settings will be synchronized with the protocol decode parameter settings. For more details, please refer to the "SENT" section in the protocol decoding chapter.

5.14 Zone Triggering

For complicated and volatile circuit signal in circuit debugging, the oscilloscope with high waveform capture rate can easily observe fleeting accidental abnormal signal. If users want to separate the abnormal signal from complicated and volatile circuit and to stable triggered. It may take a lot of time to learn the use of some advanced trigger, and even so, some more powerful advanced trigger also cannot be fully triggered.

UPO7000L series introduces an area trigger function that allows users to operate with a mouse on an external screen, making it easier to use this function. In UltraAcq mode, simply click the trigger icon  in the top right corner of the screen. In the zone trigger settings menu, select Zone A or Zone B, and one or two rectangular areas can be generated for quickly separating and observing signals.

Zone triggering can be combined with basic trigger, advanced trigger, and protocol trigger function. It also supports decoding, waveform recording, and pass/fail test, making it an effective tool for debugging complex signals.

Zone triggering provides two rectangular areas: Zone A and Zone B. Both regions support setting the trigger condition to either intersection or non-intersection.

Both regions allow setting the corresponding enable sources C1, C2, and C4.

If there is an area frame on the screen: ON (Displays the area frame), OFF (Closes the area frame).

The area frame settings can be configured as intersecting or non-intersecting.

For Area A:

Intersecting: The currently drawn area is for Area A triggering.

Condition: Triggering occurs when Area A intersects with the waveform. If there is no intersection, triggering does not occur.

Non-intersecting: The currently drawn area is for Area A triggering.

Condition: Triggering occurs when Area A does not intersect with the waveform. If there is an intersection, triggering does not occur.

For Area B:

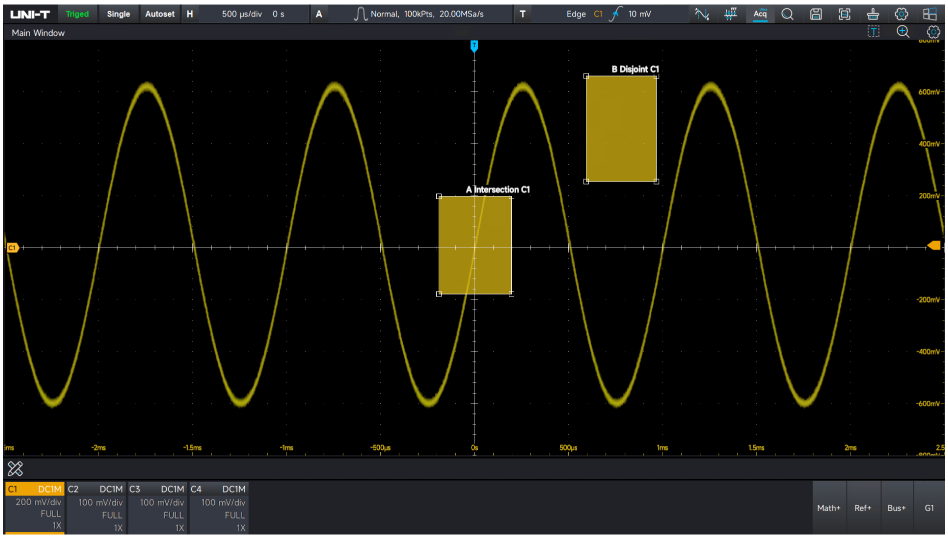
Intersecting: The currently drawn area is for Area B triggering.

Condition: Triggering occurs when Area B intersects with the waveform. If there is no intersection, triggering does not occur.

Non-intersecting: The currently drawn area is for Area B triggering.

Condition: Triggering occurs when Area B does not intersect with the waveform. If there is an intersection, triggering does not occur.

Dots are displayed within the intersecting area, and slashes are displayed within the non-intersecting area. Clicking on the area trigger frame on the screen will also bring up the settings menu, allowing the horizontal and vertical positions of the area trigger frame to be adjusted by touch.



Zone Trigger Enabling in UltraAcq Mode

6. Protocol Decoding (Option)

6.1 RS232

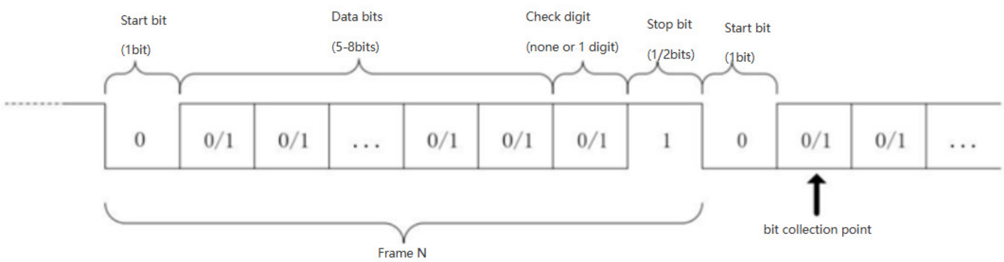
RS232 is an asynchronous transmission standard interface established by the Electronic Industries Association. It typically includes two application formats: DB-9 or DB-25. It is suitable for communication with data transmission rates ranging from 0 to 29,491,200bps.

It is widely used in microcomputer interfaces. The data to be transmitted is combined into a specified set of serial bits according to the protocol rules and sent in an asynchronous serial manner.

Each data transmission follows these rules:

- One start bit is sent first, followed by 5 to 8 data bits.
- An optional parity check bit may be included, and one or two stop bits are sent last.
- The number of data bits is agreed upon by both communicating parties, ranging from 5 to 8 bits. The parity check bit may be omitted or set to odd or even parity. The stop bit can be set to one or two bits.

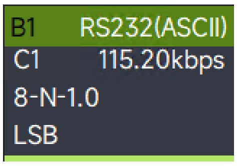
In the following description, the transmission of a data string is referred to as a frame.

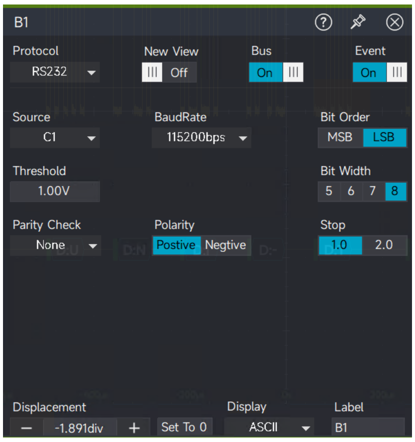


RS232 Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the RS232 protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the − and + icons. The range can be set from -5.5 to 5.5div.

(2) Trigger source

Select the trigger source, which can be C1, C2, C3, or C4.

Note: Only the channel that is connected to the signal and selected as the trigger source can provide stable triggering and correct decoding.

(3) Baud rate

The baud rate can be set to user-defined, 2400bps, 4800bps, 9600bps, 19200bps, 38400bps, 57600bps, or 115200bps.

It is recommended that users set the baud rate for RS232 according to both hardware and software specifications. RS232 is based on the fundamental model of the communication protocol and is used for short-distance (below 20 meters) and low-speed (below 1Mbps) transmission. Beyond these ranges, communication can be susceptible to interference and may become unreliable.

In RS232 communication, which is asynchronous, there is no accompanying clock signal during data transmission. To resolve the determination of data bits, the protocol requires that the baud rate be agreed upon by both communicating parties.

Generally, the baud rate defines the number of bits that can be transmitted in one second. For example, 9600bps means that 9600bits can be transmitted within one second. It is important to note that the start bit, data bits, parity bit, and stop bit are all counted as bits, meaning the baud rate is not directly equivalent to the effective data transmission rate. The oscilloscope will sample the bit values based on the set baud rate.

(4) Polarity

- Negative: Adverse logical level polarity, where a high level is represented by 0, and a low level is represented by 1.
- Positive: Adverse logical level polarity, where a high level is represented by 0, and a low level is represented by 1.
- Threshold: The voltage level that determines the signal state. If the signal voltage exceeds the threshold, it is regarded as a high level; if it is below the threshold, it is regarded as a low level.

(5) Bit sequence

To specify the data bit order for decoding the RS232 signal, choose whether the most significant bit (MSB) or the least significant bit (LSB) is transmitted first.

- MSB: The most significant bit is transmitted first.
- LSB: The least significant bit is transmitted first.

(6) Bit width

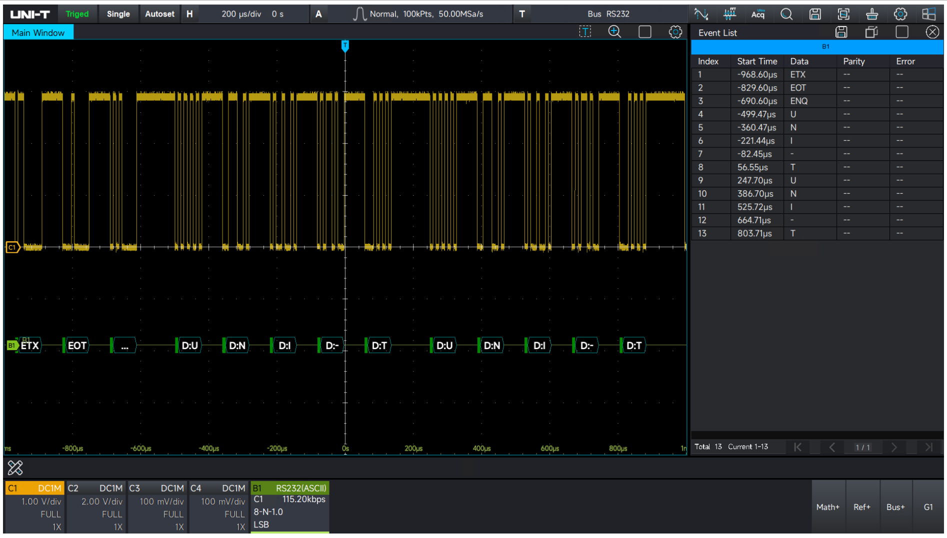
To appoint the data bit for RS232 signal to be decoding, it can select 5bits, 6bits, 7bits, or 8bits.

(7) Stop bit

Set the stop bit for each data, it can be set to 1bit or 2bits.

(8) Odd-even check

Set the odd-even check for data transmission. It can be set to no parity check, even check, or odd check.

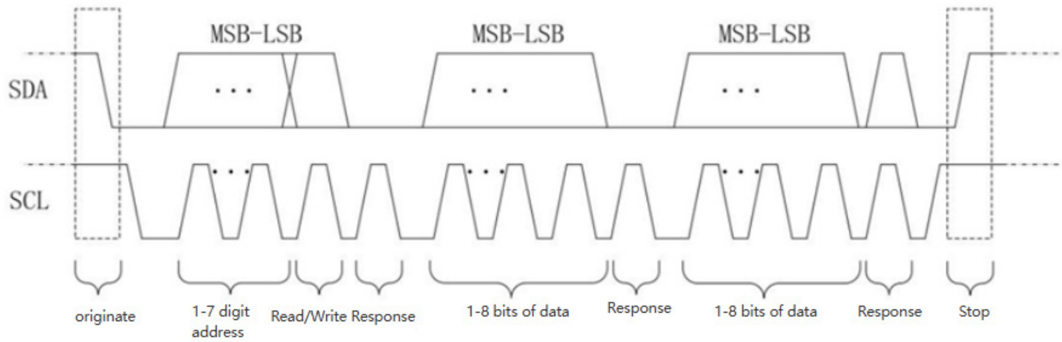


6.2 I²C

I²C triggering is commonly used to connect microcontrollers and peripheral devices, and it is widely applied in the microelectronics field. This bus protocol uses two lines for transmission: one for serial data (SDA) and the other for serial clock (SCL). It operates on a master-slave communication system, where both master and slave devices can communicate bidirectionally.

I²C bus supports multiple master devices and prevents data corruption through conflict resolution and arbitration mechanisms. Notably, the I²C bus can have two address bit widths: 7bits and 10bits. These address formats are compatible and can be used together.

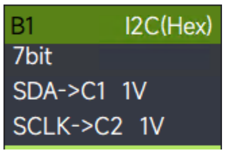
Both SCL and SDA lines in the I²C bus are pulled up to the positive power supply via resistors. When the bus is idle, both lines are at a high level. When any device on the bus outputs a low level, it pulls the bus signal low, creating a logical "AND" between the signals of multiple devices. This special logical relationship is crucial for bus arbitration. According to the protocol, the SDA data line must remain stable while the SCL clock line is high, and data is typically transmitted in MSB (Most Significant Bit) first.

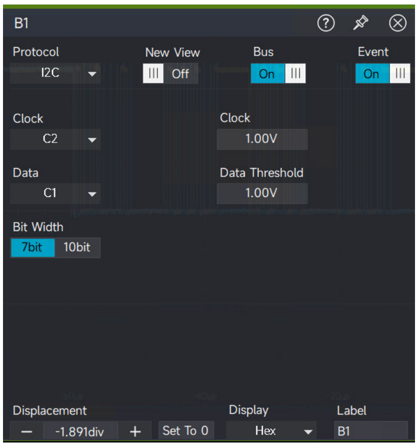


I²C Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the I²C protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the  and  icons. The range can be set from -5.5 to 5.5 div.

(2) Trigger source

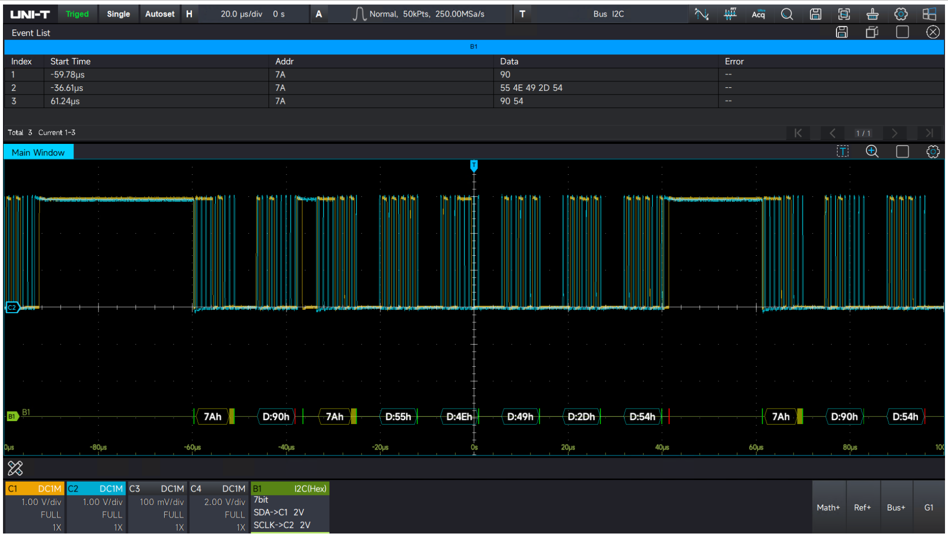
Select the trigger source, which can be C1, C2, C3, C4

Data input: Select a channel to be the data channel, which can select C1, C2, C3, C4.

Clock input: Select a channel to be the clock channel, which can select C1, C2, C3, C4.

Threshold: Set the voltage for judging the trigger level of the signal, which can set data threshold and clock threshold.

Bit width: 7bits or 10bits.

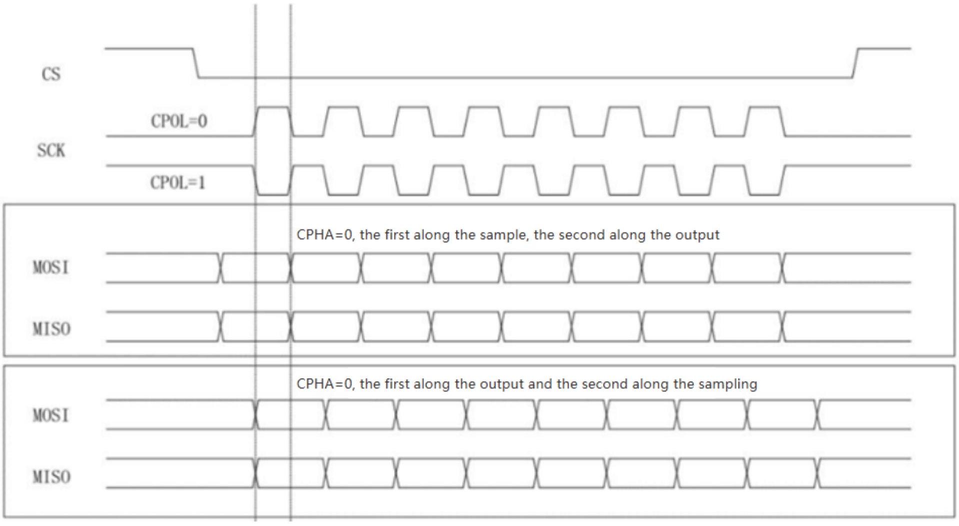


6.3 SPI

SPI (Serial Peripheral Interface) is a communication protocol that connects a host with peripheral devices in a serial manner. It is a full-duplex, synchronous communication bus. Typically, it uses four signal lines:

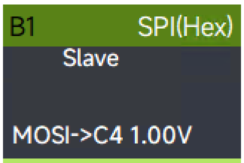
- MOSI (Master Out, Slave In): Data output from the master device and data input to the slave device.
- MISO (Master In, Slave Out): Data input to the master device and data output from the slave device.
- SCLK (Serial Clock): Clock signal generated by the master device.
- CS (Chip Select): Chip select signal from the master device to enable communication with the slave device.

The SPI interface is commonly used for synchronous serial data transfer between the host and low-speed peripheral devices. Data is transmitted bit by bit under the shift pulses of the master device, with the transmission format being MSB (Most Significant Bit) first. SPI is widely used due to its simplicity, as it does not require slave address addressing, supports full-duplex communication, and has a straightforward protocol.

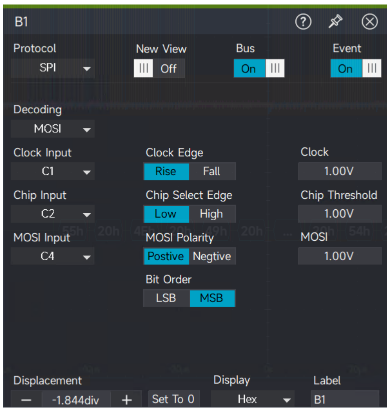


SPI Decoding Setup

(1) Bus+



Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the SPI protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.



- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the  and  icons. The range can be set from -5.5 to 5.5div.

(2) Decoding channel

MOSI (Master Out, Slave In): Data transfers from the master device to the slave device (output from master, input to slave).

MISO (Master In, Slave Out): Data transfers from the slave device to the master device (input to master, output from slave).

MOMI: Data transfers from the master device to the slave device (output from master, input to slave).

(3) Clock input

Any one of C1-C4 can be selected as the clock signal input for SPI decoding.

Clock Edge: Select either rising edge or falling edge for clock signal triggering.

Clock Threshold: The voltage level that determines the trigger level for the clock signal.

(4) Chip selection

Any one of C1-C4 can be set as the chip selection input for SPI decoding.

Chip selection edge: Select either high level or low level for chip selection triggering.

Chip selection threshold: The voltage level that determines whether the chip select signal is high or low. If the voltage is above the threshold, the signal is high; if below, it is low.

(5) MOSI input

Any one of C1-C4 can be set as the MOSI input for SPI decoding.

MOSI polarity: Select between negative or positive polarity for the MOSI signal.

MOSI threshold: The voltage level that determines the polarity of the MOSI data. If the voltage is above the threshold, the signal is positive; if below, it is negative.

(6) MISO input

Any one of C1-C4 can be set as the MISO input for SPI decoding.

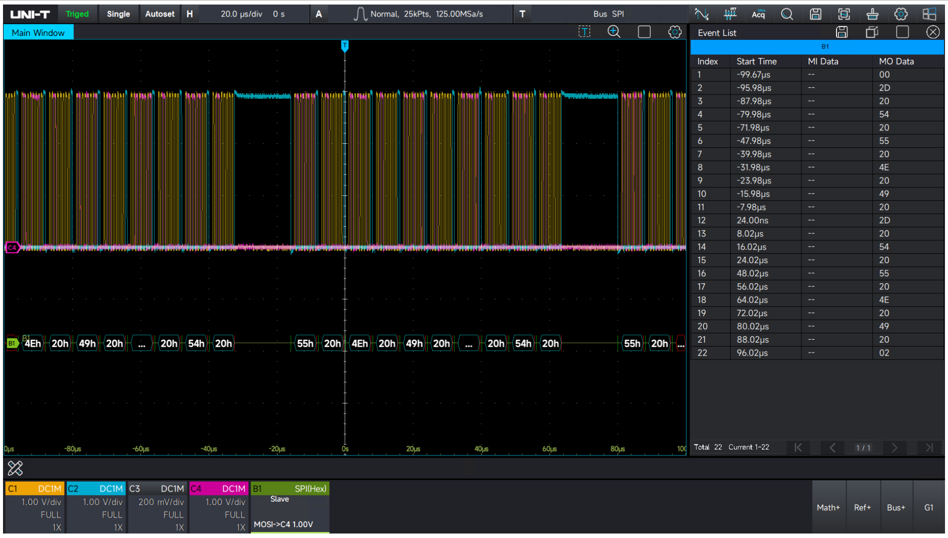
MISO Polarity: Select between negative or positive polarity for the MISO signal.

MISO threshold: The voltage level that determines the polarity of the MISO data. If the voltage is above the threshold, the signal is positive; if below, it is negative.

(7) Bit sequence

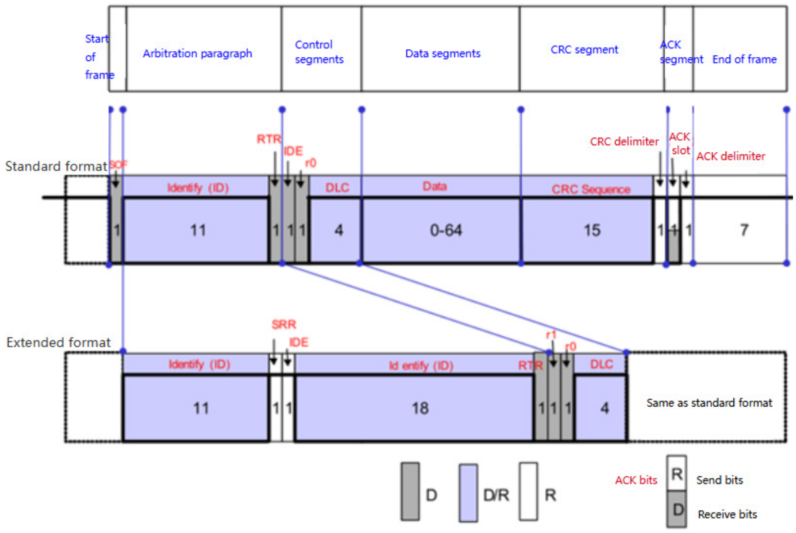
Set the bit sequence of the SPI protocol signal, choosing between MSB or LSB:

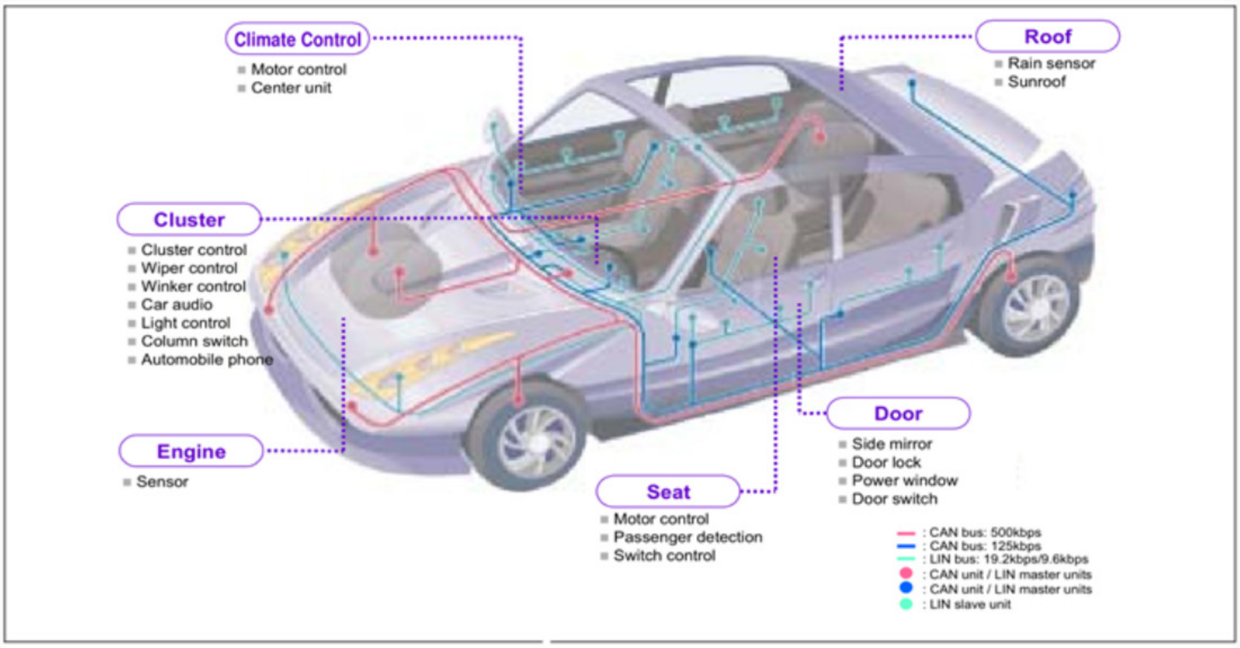
- MSB (Most Significant Bit): The high-order data bit is transmitted first.
- LSB (Least Significant Bit): The low-order data bit is transmitted first.



6.4 CAN

CAN is gaining increasing attention due to its high performance, reliability, and specialized design. It is commonly used as a single-wire or two-wire system, employing either unshielded or shielded twisted pairs for data transmission. The signal types in CAN communication are CAN_H (High) and CAN_L (Low).



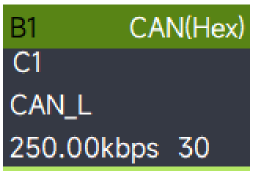


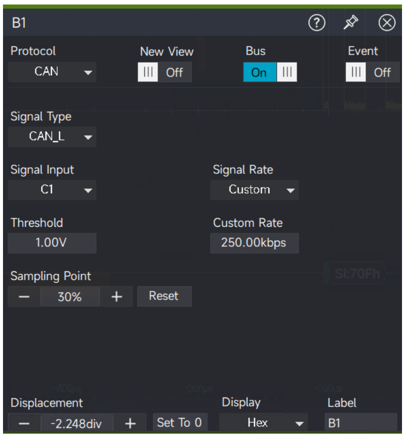
Application examples of CAN

CAN Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the CAN protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer

decoded data.

- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the – and + icons. The range can be set from -5.5 to 5.5div.

(2) Signal type

Select whether the connected signal is a high data line signal (CAN_H) or a low data line signal (CAN_L).

(3) Signal input

Any one of C1-C4 can be set as the signal input for CAN decoding.

(4) Signal rate

The signal rate can be set to 10kbps, 20kbps, 33.3kbps, 50kbps, 62.5kbps, 83.3kbps, 100kbps, 125kbps, 1Mbps, or a user-defined value.

CAN protocols and standard specifications

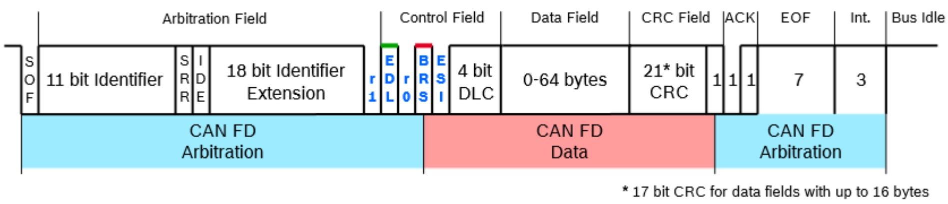
name	**baud rate**	**specification**	**Fields of application**
SAE J1939-11	250k	Double-wire, shielded twisted pair	Trucks, buses
SAE J1939-12	250k	Double-wire,shielded twisted pair，12V power supply	Agricultural machinery
SAE J2284	500k	Dual-wire, twisted-pair (unshielded)	Automotive (high-speed: powertrain, drivetrain)
SAE J24111	33.3k、83.3k	Single-line	Automotive (Low Speed: Body System)
NMEA-2000	62.5k、125k、250k、500k、1M	Double-wire, shielded twisted pair	shipping
DeviceNet	125k、250k、500k	Double-wire, shielded twisted pair，24V power supply	Industrial equipment
CANopen	10k、20k、50k、125k、250k、500k、800k、1M	Dual-wire, twisted-pair optional (shielded, powered)	Industrial equipment
SDS	125k、250k、500k、1M	Dual-wire, twisted-pair optional (powered)	Industrial equipment

(5) Sampling point

The sampling point is the point in time where the oscilloscope samples the bit level. It is expressed as a percentage of the "time from the start bit to the sampling point" relative to the "bit time" and can be set between 30% and 90%.

6.5 CAN-FD

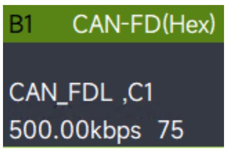
CAN_FD (CAN-Flexible Data Rate) is a new standard developed by Bosch in 2011, designed to increase the bandwidth of the CAN bus while retaining the core characteristics of the traditional CAN protocol. Compared to the original CAN protocol, CAN_FD offers higher transmission rates and longer data lengths. It can be considered an upgraded version of CAN, where the protocol has been enhanced, but the physical layer remains unchanged. CAN_FD significantly improves data transmission efficiency and increases load capacity.

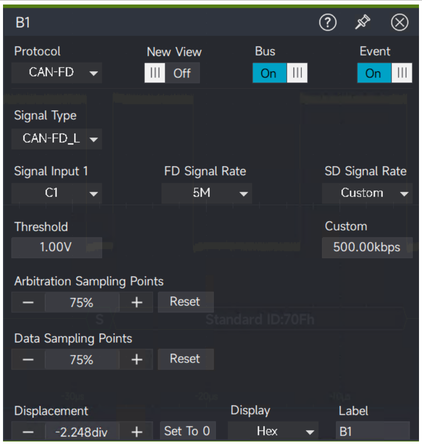


CAN-FD Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the CAN_FD protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the  and  icons. The range can be set from -5.5 to 5.5div.

(2) Signal type

Select whether the connected signal is a high data line signal (CAN-FDH) or a low data line signal (CAN-FDL).

(3) Signal input

Any one of C1-C4 can be set as the signal input for CAN-FD decoding.

(4) SD Signal rate

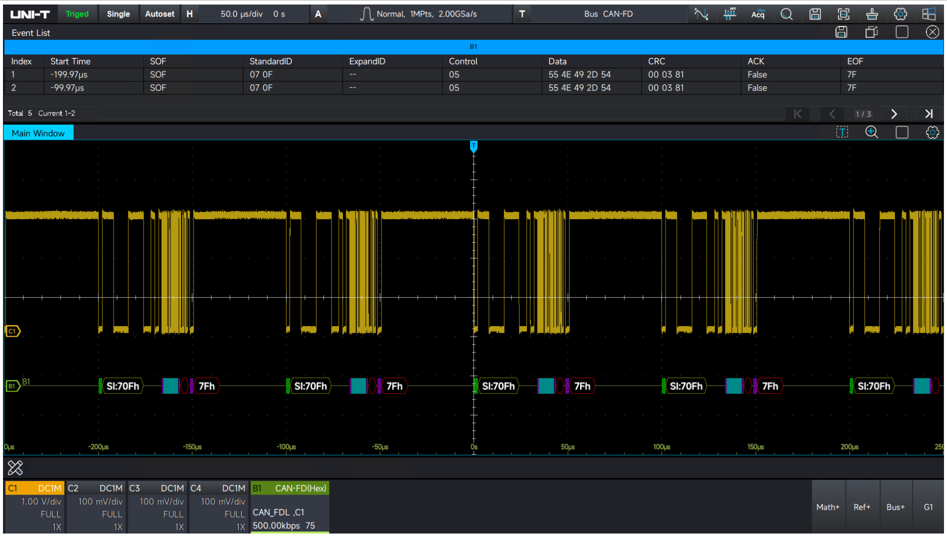
The signal rate can be set to 10kbps, 20kbps, 33.3kbps, 50kbps, 62.5kbps, 83.3kbps, 100kbps, 125kbps, 1Mbps, or a user-defined value.

(5) FD Signal rate

The signal rate can be set to 1Mbps, 2Mbps, 3Mbps, 4Mbps, 5Mbps, 6Mbps, 7Mbps, 8Mbps, or a user-defined value.

(6) Sampling point

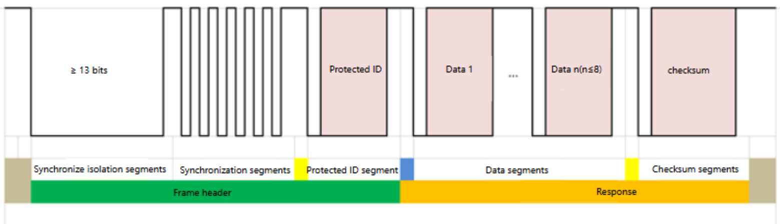
The sampling point is the point in time where the oscilloscope samples the bit level. It is expressed as a percentage of the "time from the start bit to the sampling point" relative to the "bit time" and can be set between 30% and 90%.



6.6 LIN

LIN (Local Interconnect Network) bus is a low-cost serial communication protocol based on UART/SCI (Universal Asynchronous Transceiver/Serial Communication Interface). Compared with CAN bus, LIN bus protocol is simpler and does not require high requirements for microcontrollers, which can be realized with basic serial ports, thus the cost is lower.

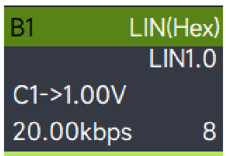
As the auxiliary bus of CAN bus, LIN bus is widely used in the field of car body control, such as doors, windows, lights and central locking. The following figure shows the message structure of LIN.

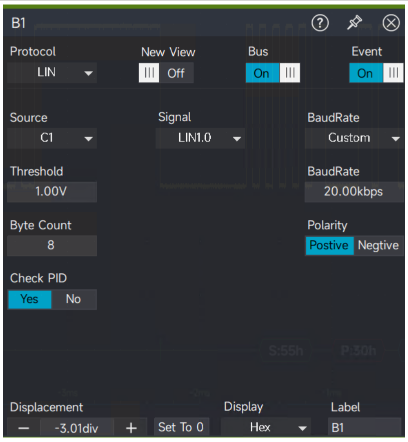


LIN Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the LIN protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the  and  icons. The range can be set from -5.5 to 5.5div.

(2) Signal input

Any one of C1-C4 can be set as the signal input for LIN decoding.

(3) Polarity

- Negative: Adverse logical level polarity, where a high level is represented by 0, and a low level is represented by 1.
- Positive: Adverse logical level polarity, where a high level is represented by 0, and a low level is represented by 1.
- Threshold: The voltage level that determines the signal state. If the signal voltage exceeds the threshold, it is regarded as a high level; if it is below the threshold, it is regarded as a low level.

(4) Baud rate

Set the signal rate for LIN. It can be configured to 2400bps, 4800bps, 9600bps, 19200bps, or a user-defined rate.

(5) Signal standard

Set the LIN signal standard. It can be configured to LIN 1.0 or LIN 2.0.

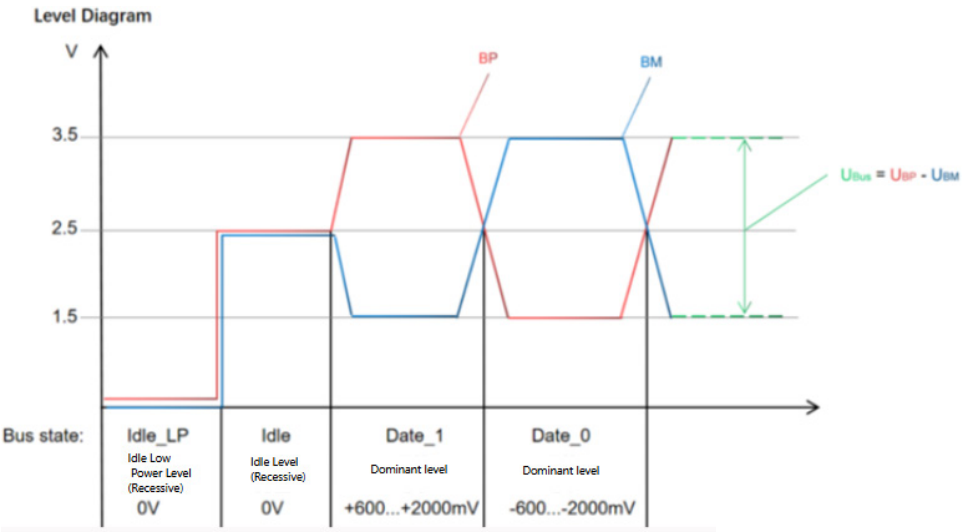
(6) Byte number

Set the data byte length for LIN. The available range is 1 to 8bits.

6.7 FlexRay

FlexRay is a differential serial bus consisting of three consecutive segments: the header, payload, and trailer. An oscilloscope samples the FlexRay signal at a specified sample position and determines whether each data point represents a logic "1" or a logic "0" based on a predefined threshold level.

FlexRay decoding requires both a specified signal type and signal rate. As a high-speed, deterministic, fault-tolerant bus technology, FlexRay is widely used in automotive applications. It combines both event-triggered and time-triggered modes to provide efficient network utilization and system flexibility.



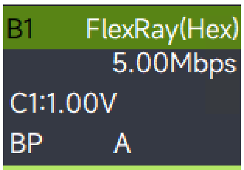
Dominance: The differential voltage is not 0 V (i.e., Data_0 and Data_1).

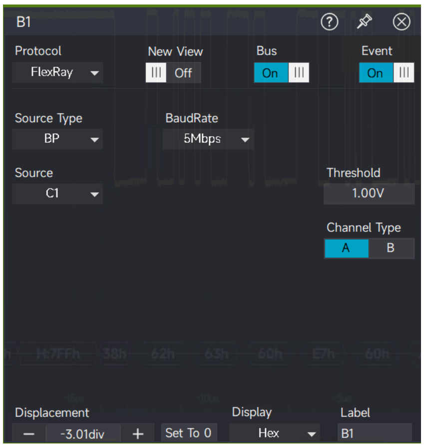
Recessive: The differential voltage is 0V (i.e., Idle_Lp, Idle).

FlexRay Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the FlexRay protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the  and  icons. The range can be set from -5.5 to 5.5div.

(2) Source type

Set the source type for FlexRay. It can be set to BP (Bus Negative), BM (Bus Positive), RX/TX, or Differential.

(3) Signal rate

Set the transmission rate for FlexRay. It can be set to 1Mbps, 5Mbps, 10Mbps, or a user-defined value.

(4) Channel type

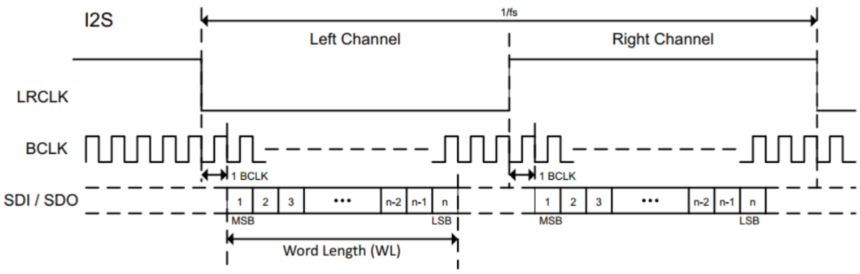
Set the channel type to either A or B.

6.8 AudioBus

The full name of I²S is Inter-IC Sound or Integrated Interchip Sound, abbreviated as IIS. It is a bus standard developed by Philips Semiconductors (now NXP Semiconductors) for audio data transmission between digital audio devices. The bus is specifically designed for data transfer between audio devices and is widely used in various multimedia systems.

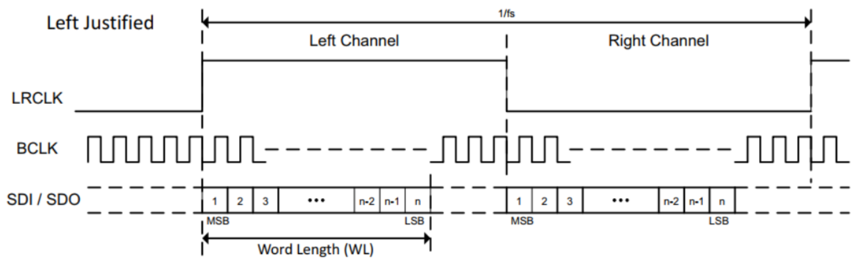
I²S supports three different operation modes depending on the relative positions of SD (Serial Data), SCK (Serial Clock), and WS (Word Select). These modes are I²S mode, left-justified mode, and right-justified mode.

I²S Mode:



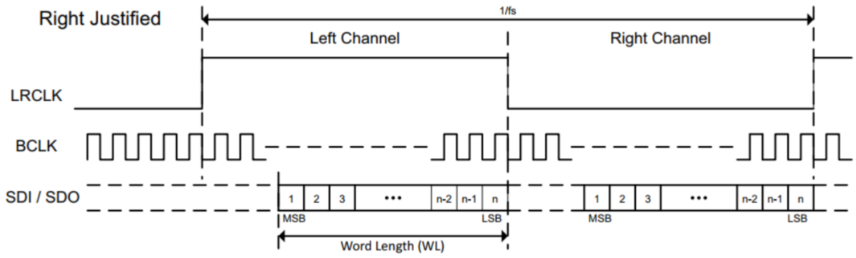
Left-Justified Mode (LJ): In this mode, data transmission begins as soon as the LRCLK (Left-Right Clock) is flipped. This standard is rarely used.

Note: When LRCLK is 1, the left audio channel data is transmitted, which is the opposite of the standard defined by I²S Philips.

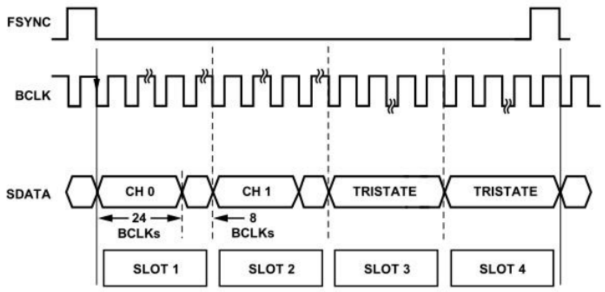


Right-Justified Mode (RJ): In this mode, the LRCLK is flipped a second time at the same moment the least significant bit (LSB) of the sound data is transmitted. Since the LSB and LRCLK are right-aligned in this configuration, it is referred to as the right-justified standard.

Note: When LRCLK is 1, the left audio channel data is transmitted, which is the opposite of the I^2S Philips standard.



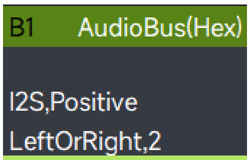
TDM (Time-division multiplexing)

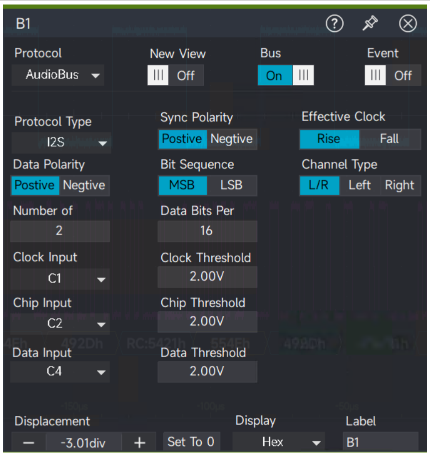


AudioBus Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the AudioBus protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the − and + icons. The range can be set from -5.5 to 5.5div.

(2) Protocol type

I^{2}S, LJ (Left-Justified), RJ (Right-Justified), TDM.

(3) I^{2}S signal setting

To set up the I2S signals, connect the oscilloscope to the chip selection (WS), serial clock (CLK), and serial data output (SDO) signals. Then, set the threshold level for each input channel, followed by the configuration of other signal parameters. The process for specifying the signal source and threshold level is similar to the procedure outlined in "I^{2}C".

(4) Synchronous polarity

Set the synchronous polarity to positive or negative.

(5) Clock feature

Specify the valid clock edge for the clock signal (CLK)

- Rising edge: Data is locked and saved on the rising edge of the clock signal.
- Falling edge: Data is locked and saved on the falling edge of the clock signal.

(6) Number of sound channels, number of clocks, and data bits

The chip selection (WS) signal should specify the number of sound channels, the number of clocks for each channel, and the number of data bits for each channel.

(7) Chip selection (WS) signal-sound channel type

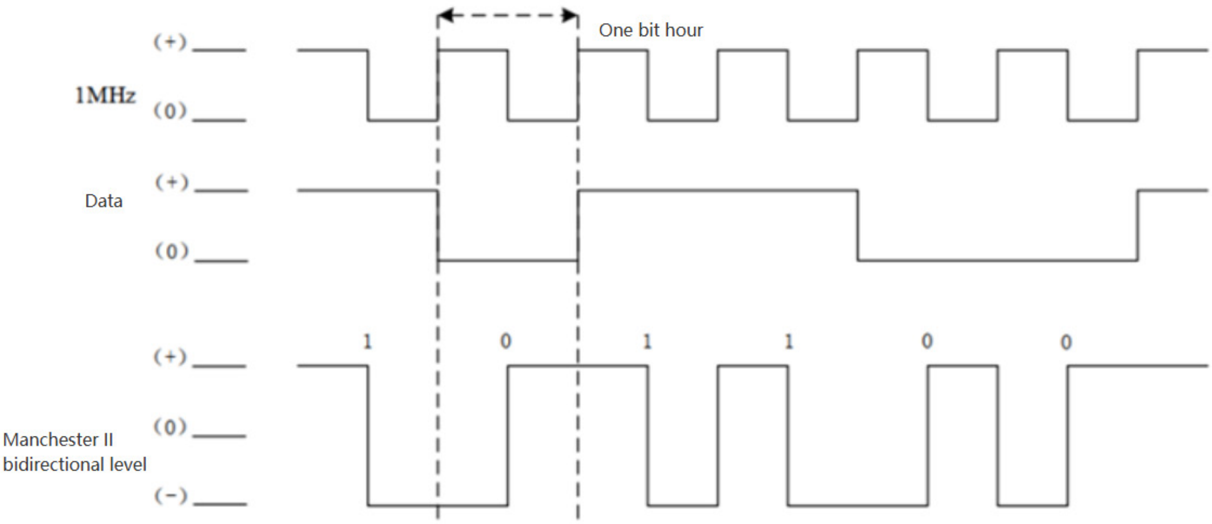
Specify whether the sound channel is for the left or right channel.

Left sound channel: Choose to assign the sound to the left channel.

Right sound channel: Choose to assign the sound to the right channel.

6.9 MIL-STD-1553

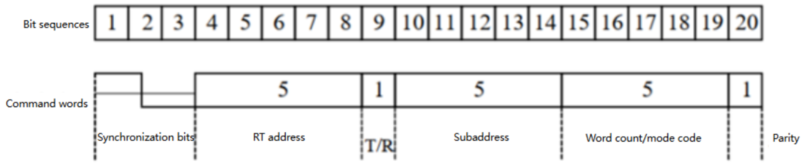
MIL-STD-1553 is a military standard issued by the U.S. Department of Defense, defining the functional characteristics of mechanical, electrical, and serial data buses. The MIL-STD-1553B bus standard is widely used in integrated avionics systems for airplanes, armored vehicles, and ships. Typically, the transmission speed of the 1553B bus is 1Mbps, although 4 Mbps is also supported using Manchester II encoding.



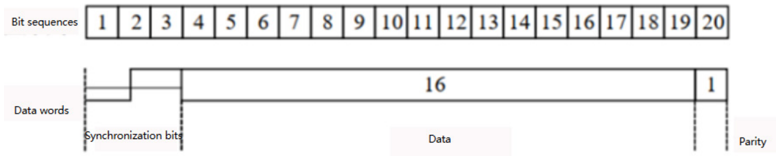
The MIL-STD-1553 message stream consists of a sequence of 1553B messages. Each 1553B message is composed of a command word, a data word, and a state word. The smallest unit in a

1553B message is a bit, with every 20bits forming a word. Each word contains 16 valid information bits, preceded by a 3-bit synchronization header (the synchronization header is divided into two 1.5-bit sections) and followed by a 1-bit parity bit (odd parity is used).

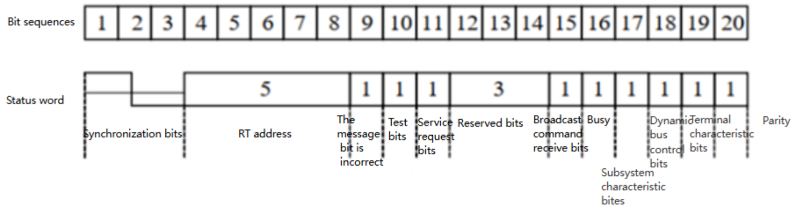
(1) Command word



(2) Data word



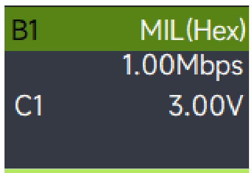
(3) State word

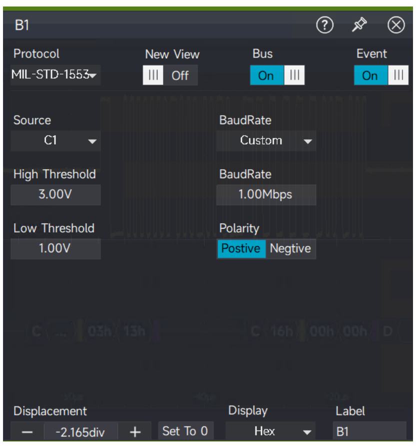


MIL-STD-1553 Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the MIL-STD-1553 protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to

hexadecimal, decimal, binary system, ASCII, or Auto.

- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the − and + icons. The range can be set from -5.5 to 5.5div.

(2) Trigger source

C1-C4 can be configured as the signal input for MIL-STD-1553.

(3) Baud rate

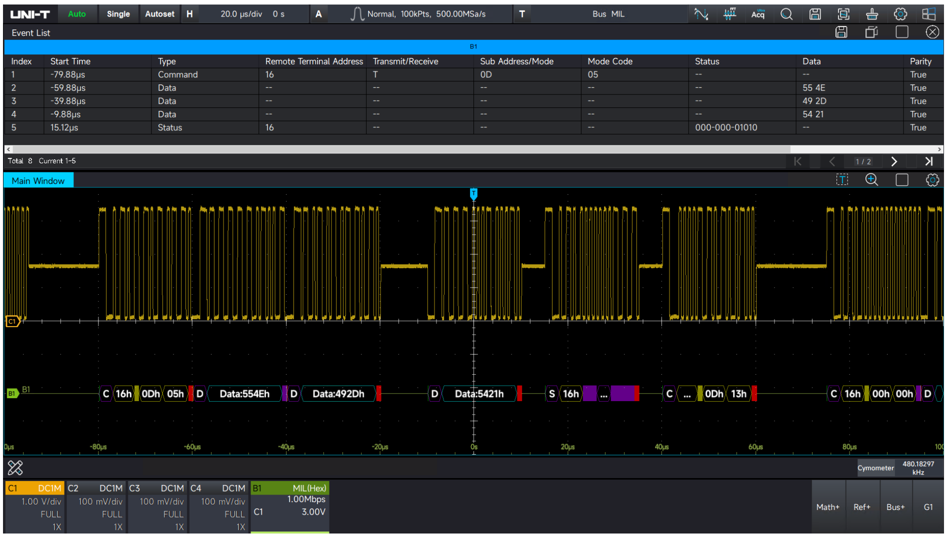
Set the transmission rate of the signal. It can be set to 1Mbps, 10Mbps, or a user-defined value.

(4) High/low level threshold

Set the threshold for distinguishing between high and low levels. The threshold range is ±30V.

(5) Data polarity

Set the data polarity to positive or negative.



6.10 ARINC429

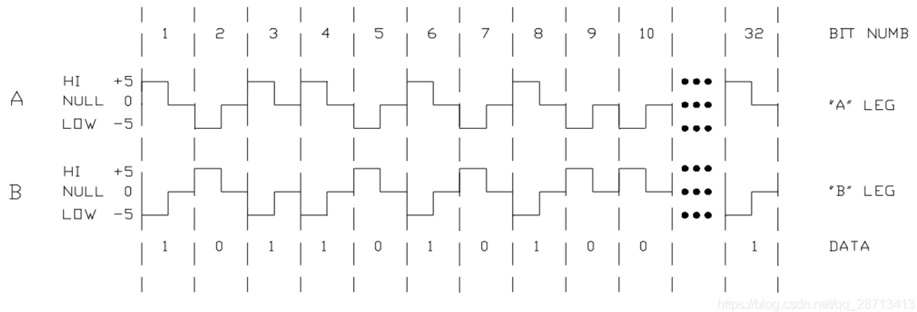
ARINC 429 bus protocol was proposed and approved by the U.S. Airlines Engineering Committee in July 1977. ARINC, which stands for Aeronautical Radio Incorporated, specifies the Digital Information Transfer System (DITS). This protocol standard defines the requirements for digital information transfer between avionics and related systems and is widely used in advanced civil airliners.

The ARINC 429 bus protocol was proposed and approved by the U.S. Airlines Engineering Committee in July 1977. ARINC, which stands for Aeronautical Radio Incorporated, specifies the Digital Information Transfer System (DITS). This protocol standard defines the requirements for digital information transfer between avionics and related systems and is widely used in advanced civil airliners.

ARINC 429 transmission rate: Slow speed: 12.5kb/s; high speed: 100kb/s. High-speed and slow-speed transmissions cannot occur on the same bus.

Threshold voltage: High level: +5V; low level: -5; null level: 0V (the difference between +5V and -5V).

ARINC 429 employs bipolar zero clearing tri-state code modulation, where the modulating signal has three possible states: "high," "zero," and "low."



In the ARINC 429 protocol layer, a data packet transmits a 32-bit message, with the Least Significant Bit (LSB) transmitted first and the Most Significant Bit (MSB) transmitted last.

32	31	30	29	11	10	9	8	1
P	SSM		DATA → ← PAD ←	DISCRETES	SDI		LABEL	
			MSB	LSB				

- Bit 1 to Bit 8: Tab Domain indicates the data type, which relates to the subsystem on the vehicle that the transmitted data pertains to.
- Bit 9 to Bit 10: SDI (Source/Destination Indicator) indicates the destination of the data or, more commonly, the source of the data.
- Bit 11 to Bit 29: Data Domain indicates in either BCD (Binary Coded Decimal) or BNR (Binary Number Representation) format. These two formats can be used together within the same message.
- Bit 30 to Bit 31: SSM (Signal/Status Matrix) describes the data character within the transmission, providing information on the signal or status.
- Bit 32: Parity (P), this is a parity check bit. ARINC 429 uses odd parity for error checking.

The parity check method works as follows:

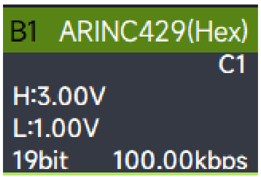
If the sum of the bits from Bit 1 to Bit 31 (counting how many 1s are present) is even, Bit 32 will display a "1."

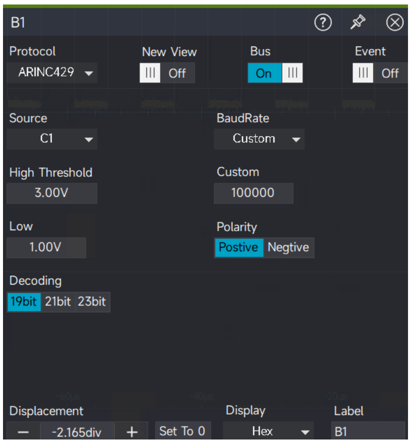
If the sum of the bits is odd, Bit 32 will display a "0."

ARINC429 Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the ARINC429 protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.
- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the – and + icons. The range can be set from -5.5 to 5.5div.

(2) Protocol type

There are two types: single-ended and differential. When the differential signal is selected, two sources should be set for the H and L signals.

(3) Trigger source

C1-C4 can be configured as the trigger source for ARINC429.

(4) Signal rate

The signal rate can be set to 12.5kbps, 100kbps, or a user-defined value.

(5) High-Low Level Threshold

ARINC 429 uses a tri-state signaling system with three levels: high, zero, and low. Therefore, it is necessary to set the high-low level threshold for proper signal interpretation.

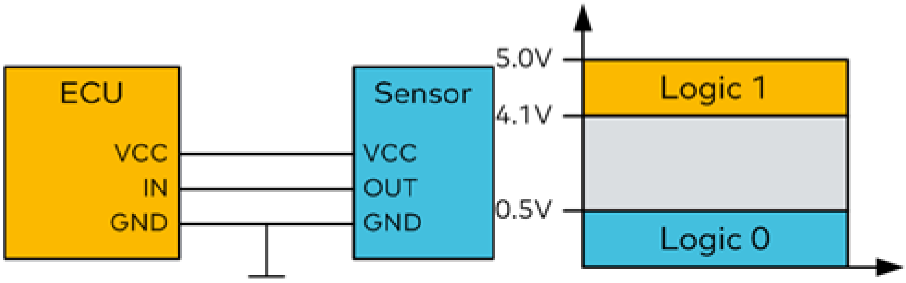
(6) Decoding mode

19-bit: data; 21-bit: data+SDI; 23-bit: data+SDI+SSM.

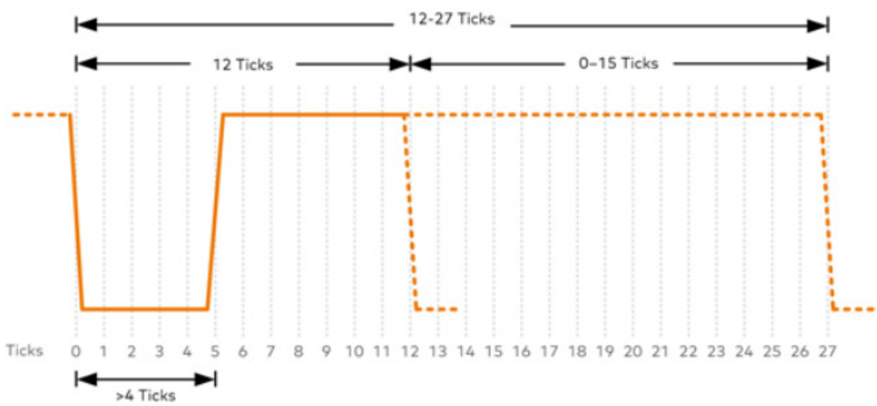
6.11 SENT

SENT (Single Edge Nibble Transmission) is a point-to-point, unidirectional transmission protocol introduced by the Society of Automotive Engineers (SAE). It is primarily used for data transmission between vehicle sensors and electronic control units (ECUs).

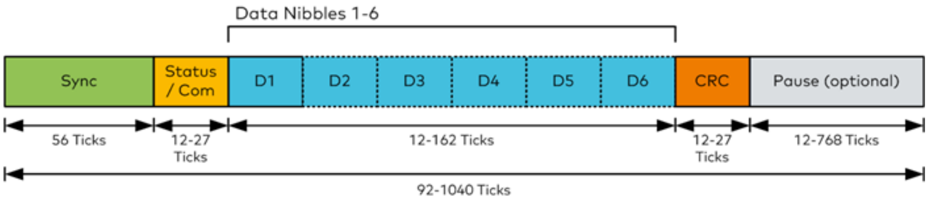
Signal level requirements of SENT: Logic level 0: 0 to 0.5V; Logic level 1: 4.1 to 5V.



The SENT protocol encodes data in 4-bit units, known as half-byte nibbles. Each nibble is defined by the time interval between two falling edges.



Frame Structure



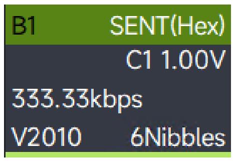
The SENT protocol uses a nibble (4bits) as its basic unit, with the following components:

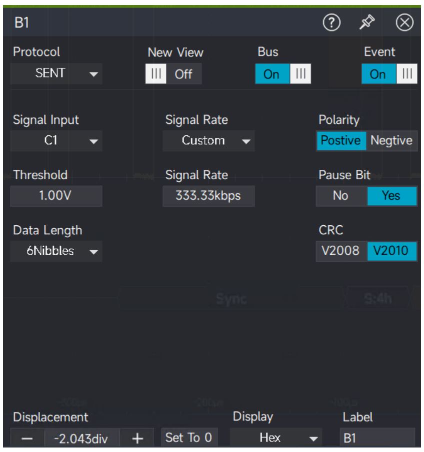
- Sync: A synchronization pulse, fixed at 56 Ticks.
- Status/Com: Represents the status and communication state, ranging from 12 to 27 Ticks, that is 1 Nibble (4-bit).
- Data: The data field, ranging from 12 to 162 Ticks (equivalent to 1 to 6 nibbles).
- CRC: The checksum field, ranging from 12 to 27 Ticks (equivalent to 1 nibble).
- Pause pulse: Ranges from 12 to 768 Ticks. In earlier SENT protocols, this field either did not exist or had a fixed number of Ticks. Starting from SENT2010, the length of this field can be dynamically adjusted based on the number of Ticks, allowing the entire SENT protocol frame to maintain a fixed total length.

SENT Decoding Setup

(1) Bus+

Click Bus+ label at the bottom right corner of the screen to open the bus decoding menu. Select the SENT protocol and configure the bus display, independent window, display format, event list, label, offset, and decoding parameters. The Bus label at the right bottom of screen will display the set values and state.





- Bus display: Set whether to turn the decoding bus on or off.
- Display format: Set the display format for the decoding bus, which can be set to hexadecimal, decimal, binary system, ASCII, or Auto.

- Event list: The event list displays the decoded data, corresponding line number, time, data, and verified data on the data line in a table format, making it easier to observe longer decoded data.
- Label: Set the label name of bus. This name will appear on the bus signal after the setting is completed, making it easier to distinguish between different bus types.
- Offset: Adjust the bus display position by using the  and  icons. The range can be set from -5.5 to 5.5div.

(2) Signal input

C1-C4 can be configured as the trigger source for SENT.

(3) Signal polarity

Set the signal polarity to positive or negative.

(4) Threshold level

The threshold level is used to distinguish logic levels for “1” and “0.”

(5) Signal rate

The signal rate can be set to 10kbps, 20kbps, 33.3kbps, 50kbps, 62.5kbps, 83.3kbps, 100kbps, 125kbps, 1Mbps, or a user-defined value.

(6) Data length

The data length can be set to 1Nibbles, 2Nibbles, 3Nibbles, 4Nibbles, 5Nibbles, or 6Nibbles.

(7) Stop bit

Set whether a stop bit is included in the transmission protocol.

(8) CRC check

Supports CRC standards V2008 and V2010.

7. Automatic Measurement

- Parameter Measurement
- Parameter Snapshot
- Add Measurement Parameter
- Measurement Statistics
- Threshold Measurement

7.1 Parameter Measurement

UPO7000L series oscilloscope can automatically measure 48 kinds of parameters, such as vertical, horizontal, and other parameters.

Vertical Parameter

Maximum (Max): The voltage from the highest point of the waveform to GND.

Minimum (Min): The voltage from the lowest point of the waveform to GND.

Peak-to-peak (Pk-Pk): The voltage value from the highest point to the lowest point of the waveform.

High: The voltage value from the flat top of the waveform to GND.

Low: The voltage value from the bottom of the waveform to GND.

Middle: Half of the sum of the voltage values at the top and bottom of the waveform.

Amplitude (Amp): The voltage from top to bottom of the waveform.

Average (Mean): The average amplitude of the waveform on the screen.

Root mean square (RMS): The energy generated by the conversion of an AC signal, it corresponds to the DC voltage that generates the same energy.

Standard Deviation (AC RMS): The RMS value of the waveform after the DC component has been removed.

Positive overshoot (+OverSht): The difference between the maximum and the highest value is divided by the amplitude.

Negative overshoot (-OverSht): The difference between the minimum and the lowest value is divided by the amplitude.

Maximum cycle (CycMax): The maximum value of waveform in one cycle.

Minimum cycle (CycMin): The minimum value of waveform in one cycle.

RMS of period (CycRMS): The energy generated by the conversion of AC signal in one cycle, it corresponds to the DC voltage that generates equivalent energy.

Average of cycle (CycMean): The average amplitude of waveform in one cycle.

Peak-to-peak of cycle (CycPK-PK): The peak-to-peak of waveform in one cycle.

Middle of cycle (CycMid): The middle value of waveform in one cycle.

Positive overshoot (+PreSht): Preshoot before the rising edge.

Negative overshoot (-PreSht): Preshoot before the falling edge.

Horizontal Parameter

Period: Time between two consecutive, same-polarity edges of a repetitive waveform.

Frequency (Freq): The reciprocal of the cycle.

Rising time (Rise): Time needed for waveform amplitude rising from 10% to 90%.

Falling time (Fall): Time needed for waveform amplitude falling from 90% to 10%.

Positive pulse width (+Width): The pulse width of a positive pulse at amplitude of 50%.

Negative pulse width (-Width): The pulse width of a negative pulse at amplitude of 50%.

Positive duty ratio (+Duty): The ratio of positive pulse width to cycle.

Negative duty ratio (-Duty): The ratio of negative pulse width to cycle.

 Time@Max: The point corresponding to the first maximum.

 Time@Min: The point corresponding to the first minimum.

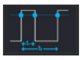
 Rising time@Lv: Rising edge duration between user-defined levels.

 Falling time@Lv: Falling edge duration between user-defined levels.

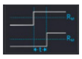
 Period@Lv: The time for each cycle at the specified level of the waveform.

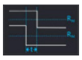
 Frequency@Lv: The frequency for each cycle at the specified level.

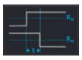
 Pulse width@Lv: The width measured at the specified level of the waveform.

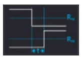
 Duty ratio@Lv: The duty ratio of the specified position.

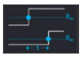
 Phase difference@Lv: Calculating the phase difference at 50% of the first rising edge between the two waveforms.

 RRD@Lv: Calculating the time difference at the specified level of the first rising edge between the two waveforms.

 FFD@Lv: Calculating the time difference at the specified level of the first falling edge between the two waveforms.

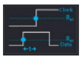
 RFD@Lv: Calculating the time difference at the specified level from the rising edge of the first waveform to the falling edge of the second waveform.

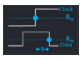
 FRD@Lv: Calculating the time difference at the specified level from the falling edge of the first waveform to the rising edge of the second waveform.

 Offset: Calculating the time difference from the first edge of 50% to the system trigger position.

 Data count: The number of sample points of the waveform data participating in the measurement.

 Periodic number: The cycle number within the periodic waveform.

 Setup time: Time from exceeding the specified intermediate reference level on the data source to the recently exceeding the specified intermediate reference level on the clock source.

 Hold time: Time from exceeding the specified intermediate reference level on the clock source to the recently exceeding the specified intermediate reference level on the data source.

Other Parameters

 Area: Algebraic sum of all point voltage and time product on the screen.

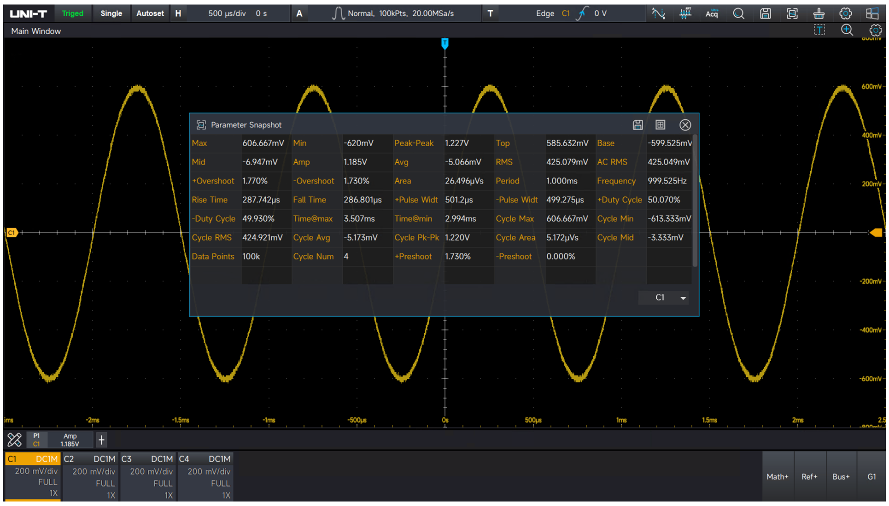
 Cycle area (CycArea): Algebraic sum of all point voltage and time product in one cycle of waveform.

7.2 Parameter Snapshot

Click the icon of measurement bar  to enable the parameter snapshot function.

The parameter snapshot is always marked with a color consistent with the current measurement channel (the primary source).

If it displays "----", indicating no signal input connect to the current measurement source or the measured result is not within the valid range (too large or too small).



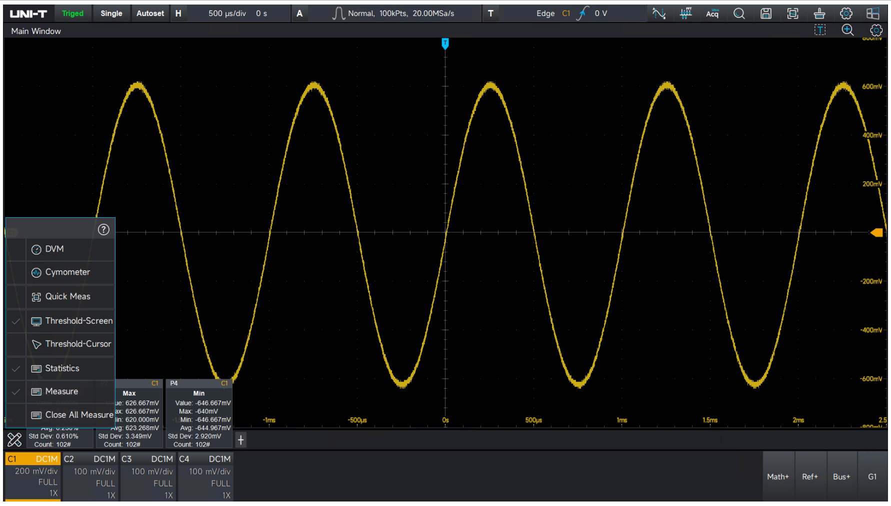
7.3 Add Measurement Parameter

UPO7000L allows users to select parameters of interest for long-term observation. Click the measurement icon at the bottom left corner to open the parameter measurement menu. By default, the previously measured parameters are restored. To access the parameter measurement page, click the icon and select the desired "Parameter" for observation. The measurement parameters are categorized into "Vertical", "Horizontal", and "Other". Selected parameters are highlighted with a blue dashed box. Click "Select" to add or modify the measurement parameters. Once added, the real-time measured value of the parameter will be displayed at the bottom left corner. These parameters can then be used for subsequent analysis, such as measurement statistics, histograms, or trend chart analysis. Parameters already added cannot be added again.

Note: Double-clicking can directly add the parameter measurement item.

7.4 Measurement Statistics

UPO7000L calculates and analyzes the currently added measurement parameters in real time based on the number of samples. To view the measurement statistics, click the measurement icon at the bottom left corner. The system supports the calculation of maximum, minimum, average, standard deviation, and sample statistics.

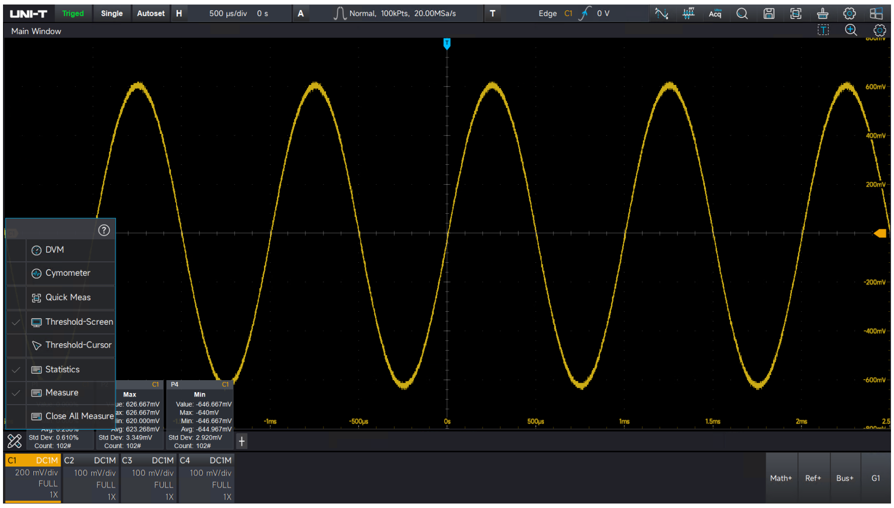


7.5 Threshold Measurement

UPO7000L allows the user to limit the test range when using automatic parameter measurement.

Threshold Measurement - Screen: The current parameter measurement range uses the entire screen as the measurement domain.

Threshold Measurement - Cursor: The current parameter measurement range uses the samples within the cursor as the measurement domain.



7.6 Close Measurement

UPO7000L enables the user to close all enabled measurement items with a single key press for convenience.

8. Cursor Measurement

- Time-domain Cursor
- Frequency-domain Cursor

Click the icon in the top right corner to enter the cursor measurement.

(1) Display: "ON" indicates the cursor measurement is enabled. "OFF" indicates the cursor measurement is disabled.

(2) Source: C1, C2, C3, C4, Math, Ref.

(3) Type: Horizontal indicates the measured result of time or frequency, while vertical indicates the measured result of voltage or power.

(4) A position: The position of cursor A on the screen, with unit of div. It can adjust by clicking the − and + icons to adjust the A position in the cursor measurement window or clicking A position to open the numeric keyboard to adjust.

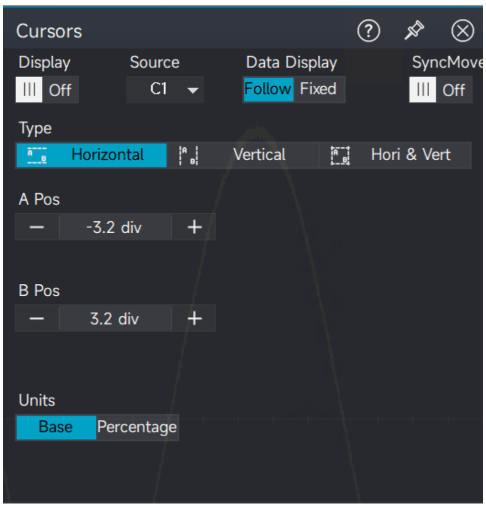
(5) B position: The position of cursor B on the screen, with unit of div. It can adjust by clicking the − and + icons to adjust the B position in the cursor measurement window or clicking B position to open the numeric keyboard to adjust.

(6) Data display: Suspend or fixed

(7) Synchronous Movement: This function is disabled by default. Moving cursor A or cursor B will not affect the position of the other cursor. When synchronous movement is enabled, an icon will appear next to cursor B. Moving cursor B will cause cursor A to follow, maintaining the relative distance between them. Moving cursor A will not affect cursor B position.

(9) Horizontal unit: s, %.

(10) Vertical unit: s, °, %.



8.1 Time-domain Cursor

Source: C1-C4, Math, REF

Vertical Measurement

Select the cursor type to "Vertical" in the cursor measurement menu.

"X" indicates the measured result of channel time.

"Y" indicates the measured result of voltage at the intersection of the open channel and the cursor.

"△X" indicates the absolute value of the time difference measured by the two cursors A-B.

"△Y" indicates the absolute value of the voltage difference measured by the two cursors A-B.

"1/△X" indicates the reciprocal of the time difference measured by the two cursors A-B (indicating the waveform frequency between the two cursors A-B).

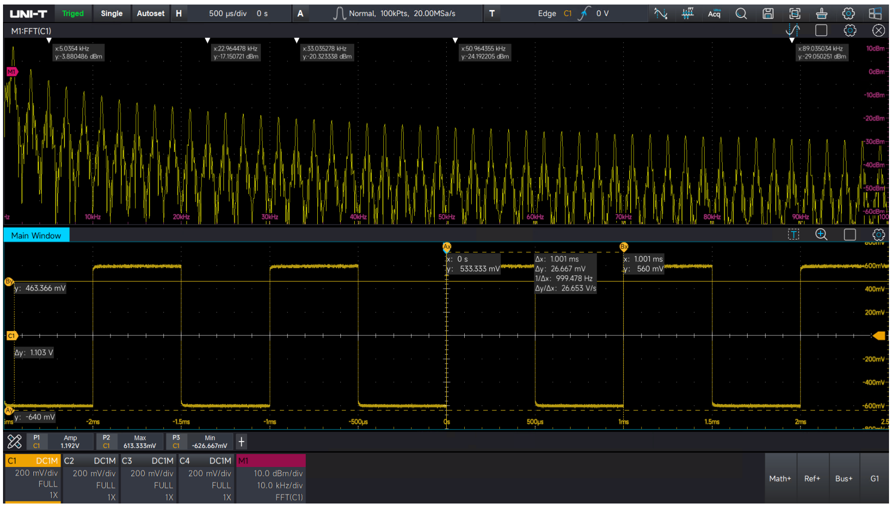
"△Y/△X" indicates the absolute value of voltage variation at two points A-B in unit interval.

Horizontal Measurement

Select the cursor type to "Horizontal" in the cursor measurement menu.

"Y" indicates the measured result of cursor voltage.

"△Y" indicates the absolute value of the voltage difference measured by the two cursors A-B.



8.2 Frequency-domain Cursor

Source: Math

Vertical Measurement

Amplitude spectrum

"X" indicates the measured result of channel frequency.

"Y" indicates the measured result of amplitude/power at the frequency intersection of the open channel and the cursor.

"△X" indicates the absolute value of the frequency difference measured by the two cursors A-B.

"△Y" indicates the absolute value of the amplitude/power difference measured by the two cursors A-B.

"1/△X" indicates the reciprocal of the frequency difference measured by the two cursors A-B (indicating the time difference between the two cursors A-B).

"△Y/△X" indicates the absolute value of amplitude/power variation at two points A-B in frequency interval.

Horizontal Measurement

Select the cursor type to "Horizontal" in the cursor measurement menu.

"Y" indicates the measured result of cursor amplitude/power

"△Y" indicates the absolute value of the amplitude/power difference measured by the two cursors A-B.

9. Mathematical Operation

- Basic Operation
- FFT
- Filter
- ERes
- Advanced Operation
- User-defined Operation

UPO7000L series digital phosphor oscilloscope offers a variety of mathematical operations, including basic operations, FFT, digital filters, Eres, advanced operations, and user-defined functions.

To access the mathematical operation menu, click the "Math +" label. In this menu, users can adjust the vertical scale, vertical position, horizontal scale, and horizontal position of the math waveform. Additionally, the math waveform tab can be configured, and the unit for the mathematical operation can be customized.

Math operation cursor 

marks the result of a mathematical operation.

9.1 Basic Operation

The waveforms involved in the operation can be analog waveforms, mathematical waveforms, or reference waveforms. Mathematical operations can be performed on the channel waveforms using the following operators: "+", "-", "×", and "÷", which produce the final MATH waveform.

(1) Operator: "+", "-", "×", "÷":

- +: The waveforms of source 1 and source 2 are added point by point.
- -: The waveform of source 2 is subtracted from source 1 point by point.
- ×: The waveforms of source 1 and source 2 are multiplied point by point.
- ÷: The waveform of source 1 is divided by source 2, point by point.

9.2 FFT

FFT (Fast Fourier Transform) operation can convert time-domain signal (YT) to frequency-domain signal. The following types of signals can be easily observed by using FFT.

Harmonic content and distortion in measurement system

- Noise feature in DC power supply

- Vibration analysis

Vertical Unit

The unit of the FFT operation results.

Amplitude spectrum: **Vrms** and **dBm**. **Vrms** and **dBm** respectively display the vertical amplitude size in linear and decibel volts. If the FFT spectrum needs to display in a large dynamic range, dBm is recommended.

Frequency Range

- Center frequency: Set the frequency for the center frequency point of the FFT spectrum view
- Span: Set the scanning range of FFT spectrum, the center frequency point is used as the reference, and the left and right bandwidths take each half of the span.

Count

The number of points processed by the FFT spectrum, it can be set to Num1k, Num2k, Num4k, Num8k, Num16k, Num32k, or Num1M.

Window

Window functions help reduce spectral leakage, making the time-domain signal better conform to the periodicity required for FFT processing. A good window function should ideally have a narrow main lobe in its frequency spectrum and high side lobe attenuation. However, it's important to note that these two attributes cannot be maximized simultaneously. Therefore, the selection of the window function should be based on the specific requirements of the measurement. A narrower main lobe improves frequency resolution, while higher side-lobe attenuation improves amplitude accuracy. The following window functions are available for different measurements: Hamming, Blackman, Rectangle, Hanning, and flat-top. Each window function has distinct characteristics and is suited for different types of waveforms. The choice of window function should be based on the specific properties of the waveform being measured and the requirements of the measurement.

1. Rectangle: It has the best frequency resolution and the worst amplitude resolution, which is similar to the one with no window. It is suitable for measuring the following waveforms.
   - Transient or short pulse, the signal level is almost equal to before and after
   - Equal amplitude sine wave with very similar frequency
   - Wide-band random noise in a slowly changing spectrum
2. Hanning: Compared with the rectangle window, it has better frequency resolution, but poorer amplitude resolution. It is suitable for measuring sine, periodic and narrow-band

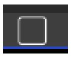
random noise waveforms.

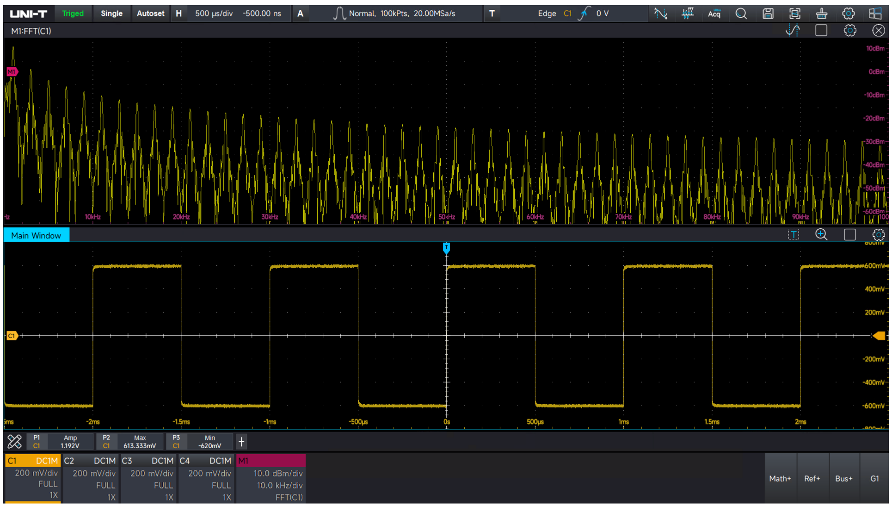
3. Hamming: The frequency resolution is slightly better than that of Hanning window. It is suitable for measuring transient or short pulse, and waveform in which the signal level varies considerably before and after.
4. Blackman: It has the best amplitude resolution, and the worst frequency resolution. It is suitable for measuring the single frequency signals or seeking higher harmonics.
5. Flat-top window: Accurate measurement signal, it is suitable for measuring the signal without precise reference substance but requires accurate measurement.

Output Mode

- Amplitude spectrum
- Power spectrum
- Psd (Power spectral density)
- Real part
- Imaginary part
- Phase spectrum

Display Mode

Open FFT default separate window, click the icon  with the mouse in the top right corner to display full screen.



FFT Operating Tip: The signal with DC component or deviations can cause errors or deviations in the FFT waveform components. To reduce the DC component, the user can set the channel to AC coupling mode.

Peak Marker

click the icon with the mouse in the top right corner to enable the peak marker function.

(1) Select marker source

UPO7000L supports 2 FFT windows, which can be enabled simultaneously. The sources can be selected from any FFT channels.

(2) Select threshold and number of markers

The threshold determines the display position of the peak marker. The number of markers specifies how many peaks can be marked, with a range of 1 to 11 markers.

(3) Select marker reading

Absolute Value: Select the absolute value of the marker.

Increment: Select the absolute value of the frequency difference and amplitude difference between the first and second markers, and so on.

(4) Enable automatic marker

When the automatic marker is enabled, the oscilloscope will mark the peaks in real-time.

9.3 Filter

Filter Type

The filter type can be low-pass, high-pass, band-pass, and band-resistant.

- Low pass: Only signals with a source frequency lower than the current “cut-off frequency 1” are allowed to pass.
- High pass: Only signals with a source frequency higher than the current “cut-off frequency 1” are allowed to pass.
- Band pass: Only signals with a source frequency higher than the current “cut-off frequency 1” and lower than the current “cut-off frequency 2” are allowed to pass.
- Band limit: Only signals with a source frequency lower than the current “cut-off frequency

1” or higher than the current “cut-off frequency 2” are allowed to pass.

Cut-off frequency 1

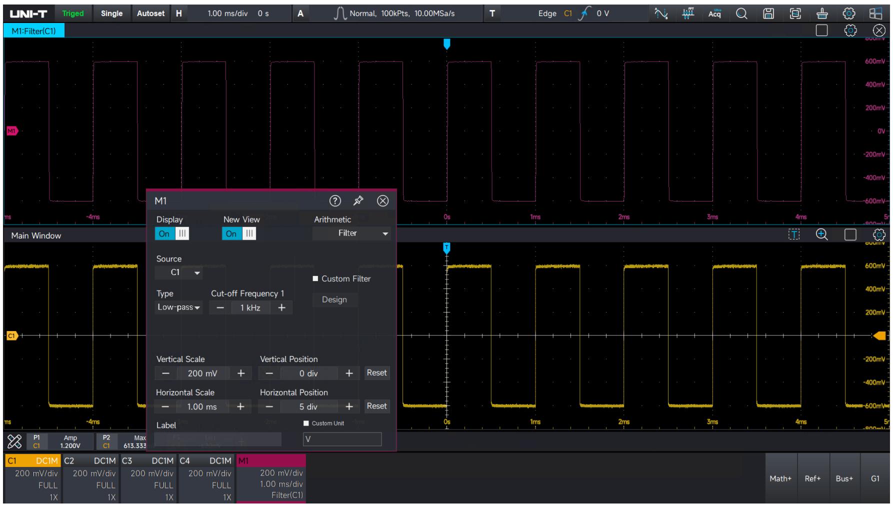
It can be set by clicking the − and + icons in the menu or using the numeric keyboard to enter the cut-off frequency.

Cut-off frequency 2

It can be set by clicking the − and + icons in the menu or using the numeric keyboard to enter the cut-off frequency.

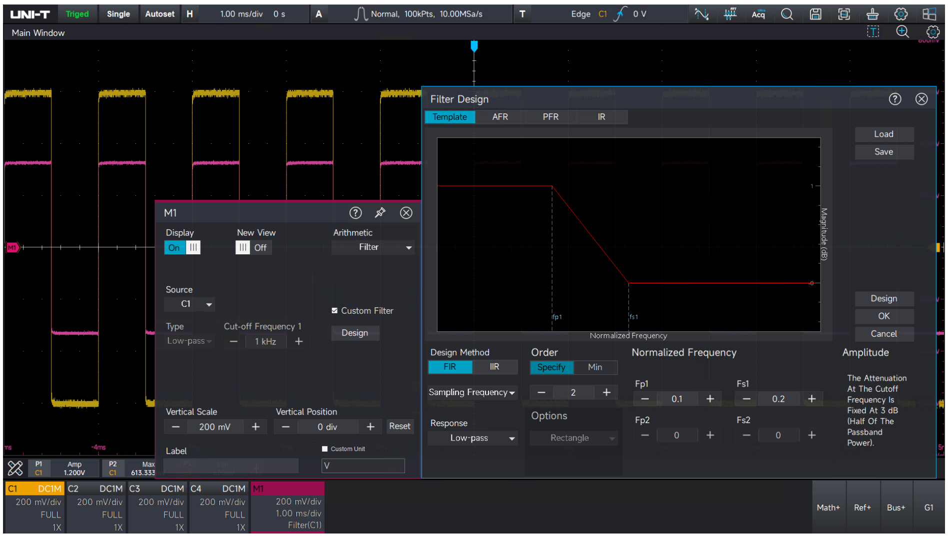
In the low/high pass filter, if the cut-off frequency 2 is invalid, the menu will be hidden.

Note: The range of the cut-off frequency is dependent on the current horizontal time base.



Custom Filter Designer

UPO7000L series enables custom filter design, supporting both FIR (Finite Impulse Response) and IIR (Infinite Impulse Response) filter designs. During the filtering process, select the custom filter option and click "Design" to open the custom filter designer.



Design interface:

1) Response types: Low pass, high pass, band pass, and band limit filters.

2) Design methods: IIR and FIR.

3) Filter order: The order can be configured based on your requirements. For FIR filters, the range is from 2 to 50. For IIR filters, the range is from 2 to 1000; the user can also directly select the minimum order for IIR.

4) Frequency: Configure these settings based on user-specific requirements.

5) Filter feature: Amplitude-frequency, phase-frequency, and pulse response.

The following example introduces the design parameters for a filter:

1) Design a Butterworth low-pass filter.

2) Set the order to 10.

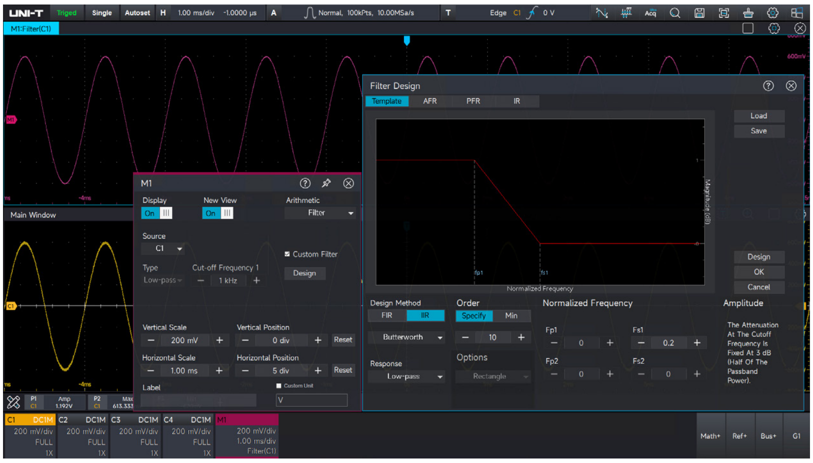
3) Set the sampling frequency to 500kHz.

4) Set the cut-off frequency to 50kHz.

5) Using the calculation formula for the normalized frequency: fp/fs/2, the normalized cut-off frequency coefficient is calculated as 0.2.

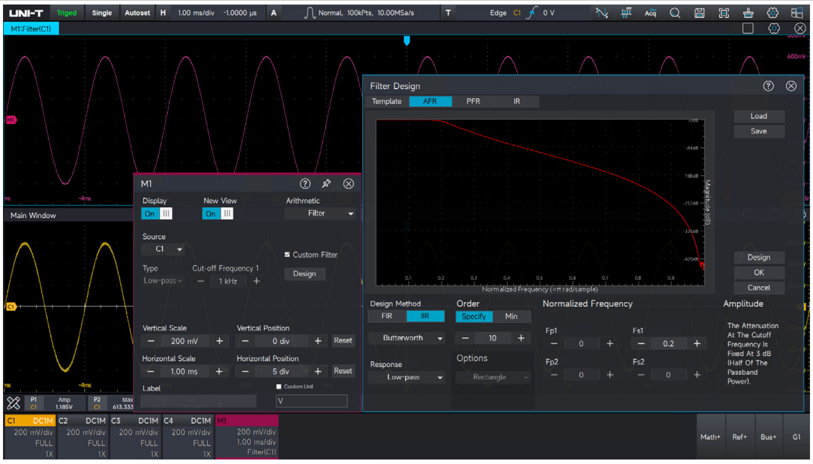
Set the parameters as described above and click "Design."

Template reference:

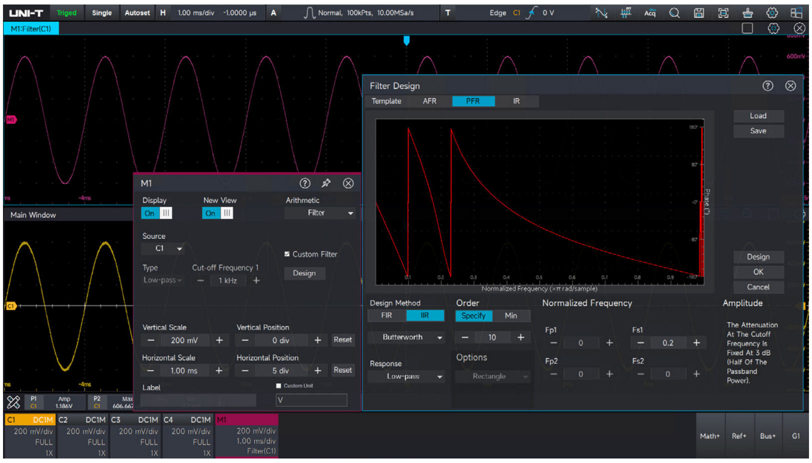


After the design is complete, the user can view the characteristics of the designed filter.

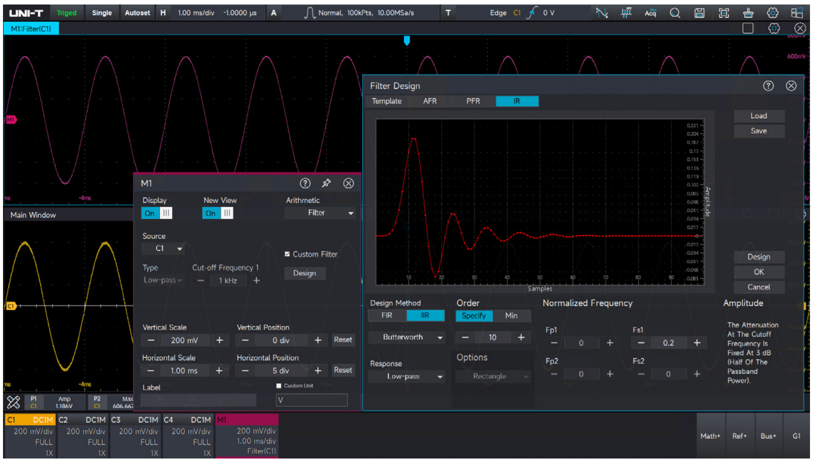
Amplitude-frequency feature:



Phase-frequency feature:



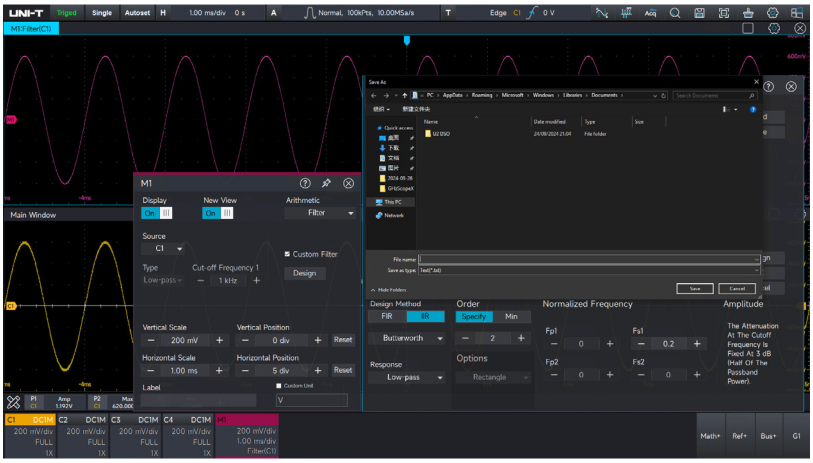
Pulse response feature:



Click "OK" to apply the designed filter to waveform processing.

Save and Load filter design:

The user can click "Save" in the upper-right corner of the designer to save the filter design as a script. To use it again, click "Load" to load the saved design.



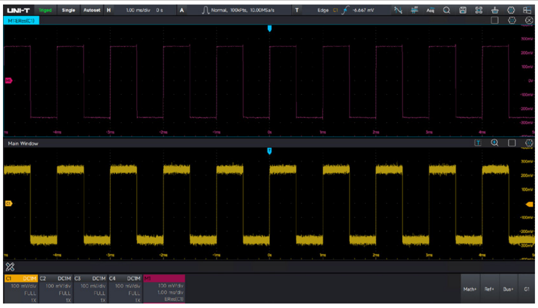
9.4 ERes

Enhanced resolution (ERes) mode has two features.

(1) In all cases, each filter uses a fixed value to improve the resolution (i.e., the ability to distinguish between closely spaced voltage levels). This effectively enhances resolution regardless of whether the signal contains noise, is a single-shot signal, or is a repetitive signal.

(2) The Signal-to-Noise Ratio (SNR) can also be improved, depending on the noise characteristics in the original signal. The Enhanced Resolution (ERes) mode reduces the signal bandwidth, allowing certain noise components to be filtered out.

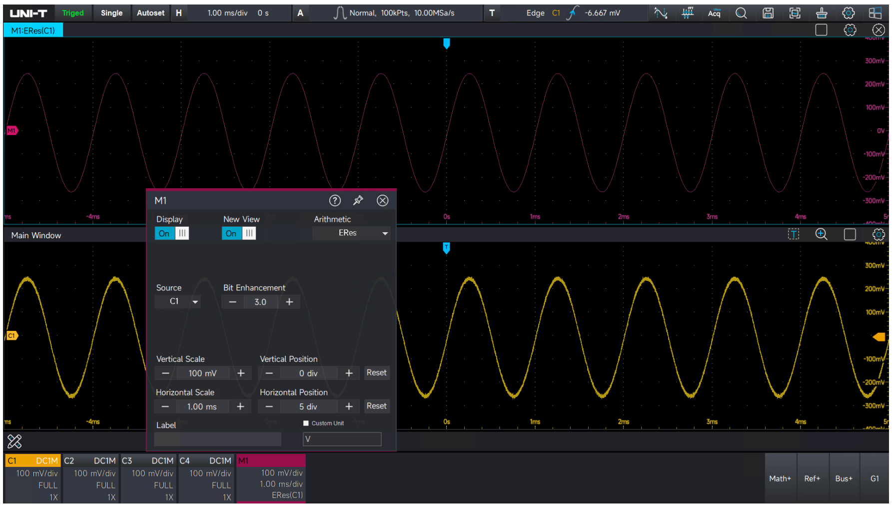
Unprocessed waveform: noise 70mV



Enhance resolution 3bits: noise about 10mV

Set ERes Mode

Select “ERes” in Math menu, and select source to set enhance bit (0.5 to 3).



9.5 Advanced Operation

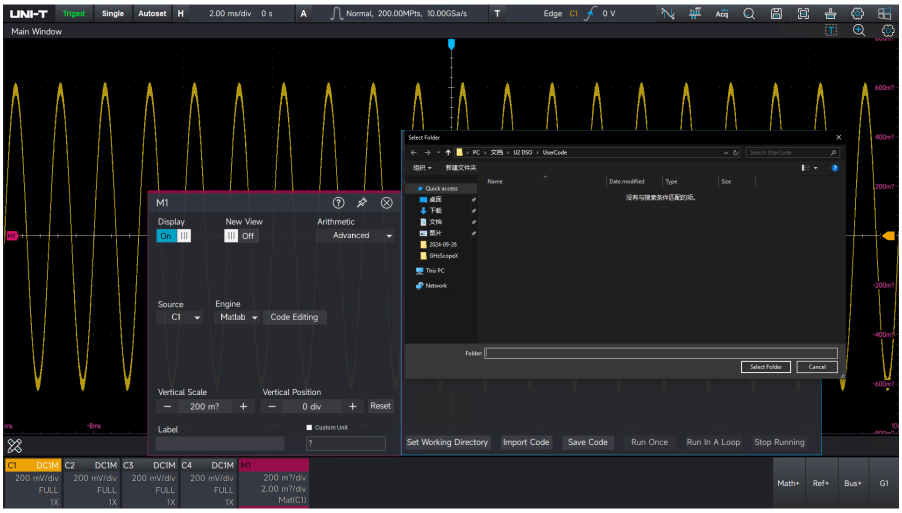
UPO7000L supports embedded Matlab and data presentation, the programming results are run directly on the oscilloscope.

Running Matlab

In the Math menu, select “Advanced Operation”, set the engine library type to “Matlab”, click the code compiler to open the Matlab, connect a keyboard, and then either directly input or import the Matlab code. Click “Run” to execute the code and obtain the programming result.

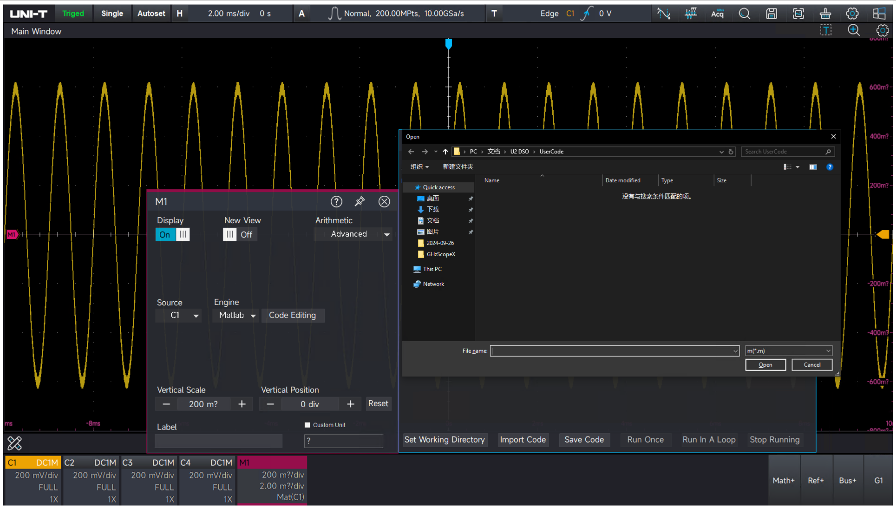
Set Working Directory

The default path for the code compiler to compile files can be adjusted by clicking "+" or "-" to change the working directory level. Click the “folder” icon to select a folder as the compiler's working directory. This directory will be opened by default for saving and importing code.



Import Matlab

Click “Import”, locate the saved Matlab code file with the “.m” suffix in the working directory, select it, and click “Confirm” to load it into the code compiler. Ensure the file is copied to the working directory in advance, or import it from other removable storage devices (e.g., a USB flash drive).

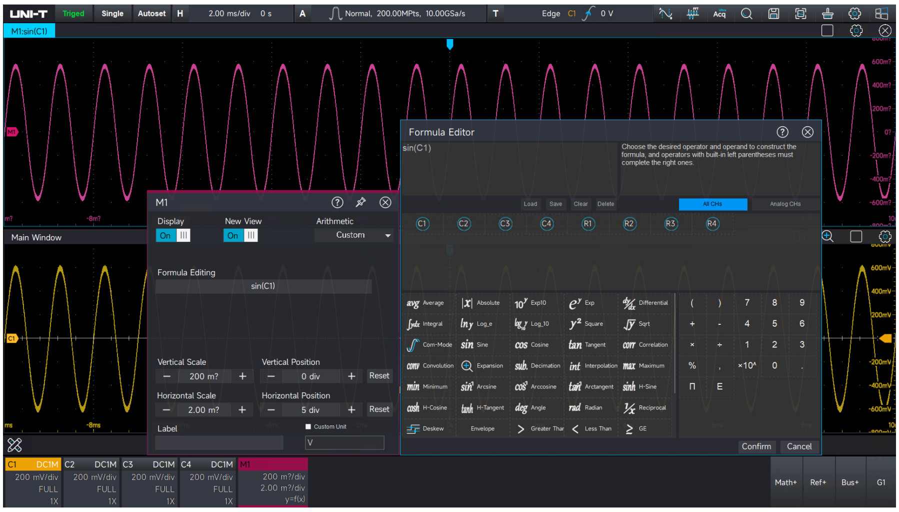


Save Matlab File

Create a blank ".m" file in the working directory in advance. Then, click "Save" in the code compiler and select the created blank file to save the code. The saved file can later be recalled by importing it.

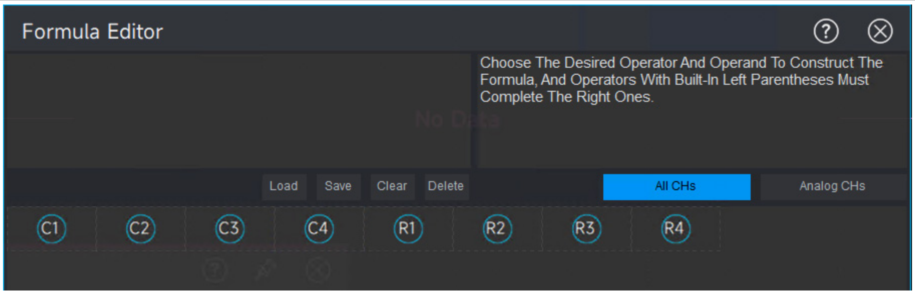
9.6 User-defined Operation

The operation for each signal input channel can be freely set to obtain the MATH waveform with different operation results. While editing the expression, the user can "Delete", "Clear", "Load", or "Save" the expression. After applying the expression, the oscilloscope performs the operation and displays the result. The following figure shows the result of five-times averaging for the C1 channel.

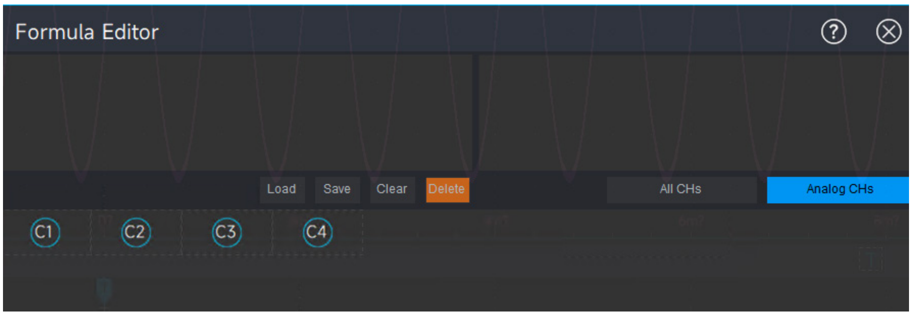


Select Channel

All Channels: Four analog channels (C1–C4), four reference waveforms (R1–R4).

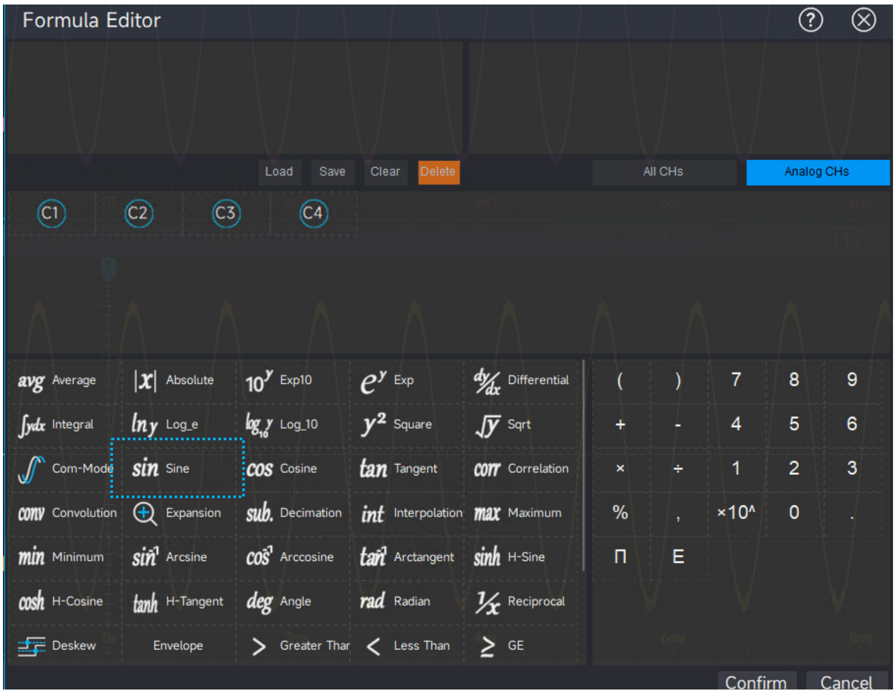


Analog channel C1-C4



Formula Editing Area

When an expression is selected, a blue dashed box will appear, and the mathematical formula analysis along with the definition of the variable will be displayed in the expression dialog box. Please strictly follow the analysis requirements when entering the variable assignment; the calculation cannot be performed if the input does not meet the requirements. Meanwhile, the oscilloscope will display “Input Format Error.”



10. Reference Waveform

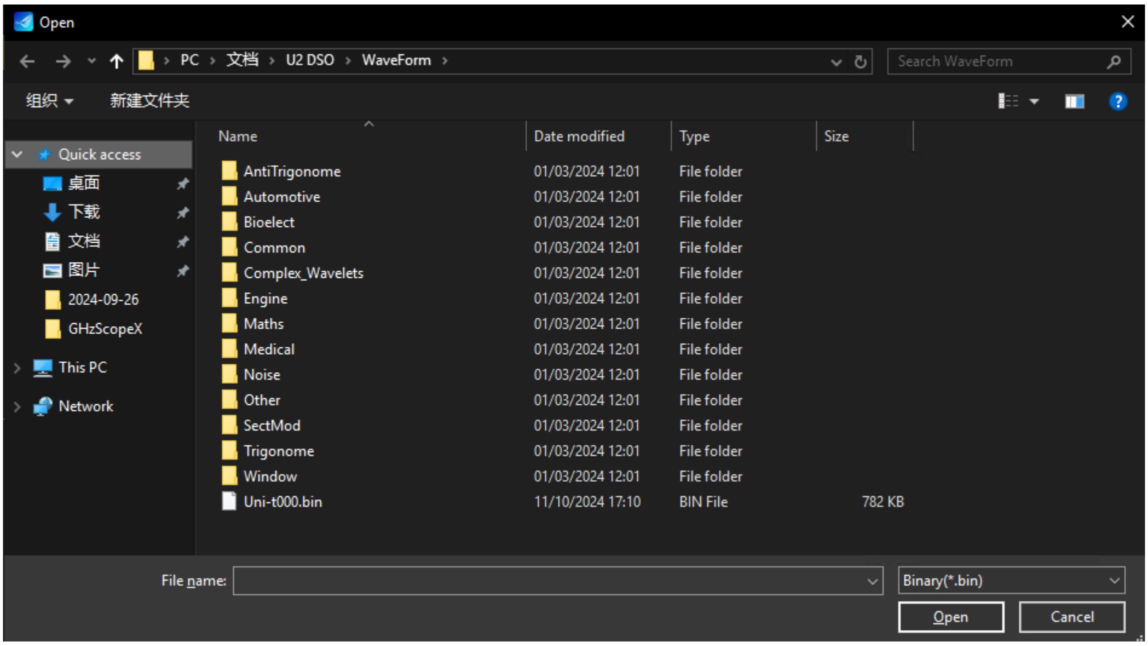
- Open Ref Function
- Adjust Ref Waveform
- Close Ref Waveform

UPO7000L supports loading waveform files from the internal system or external storage and allows loading four reference waveforms to compare with other waveforms. This comparison helps analyze the differences between the waveforms to identify the cause of the failure.

10.1 Open Ref Function

Click the "Ref+" icon at the bottom right corner to open the reference waveform file. Select the corresponding path and waveform file to load the reference waveform.

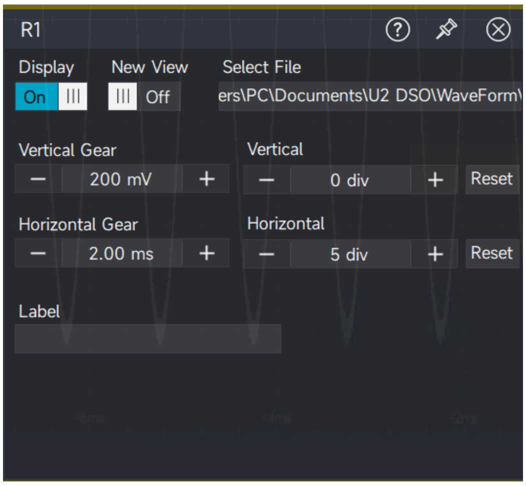
UPO7000 currently supports loading reference waveform files only in .bin and CSV format.



10.2 Adjust Ref Waveform

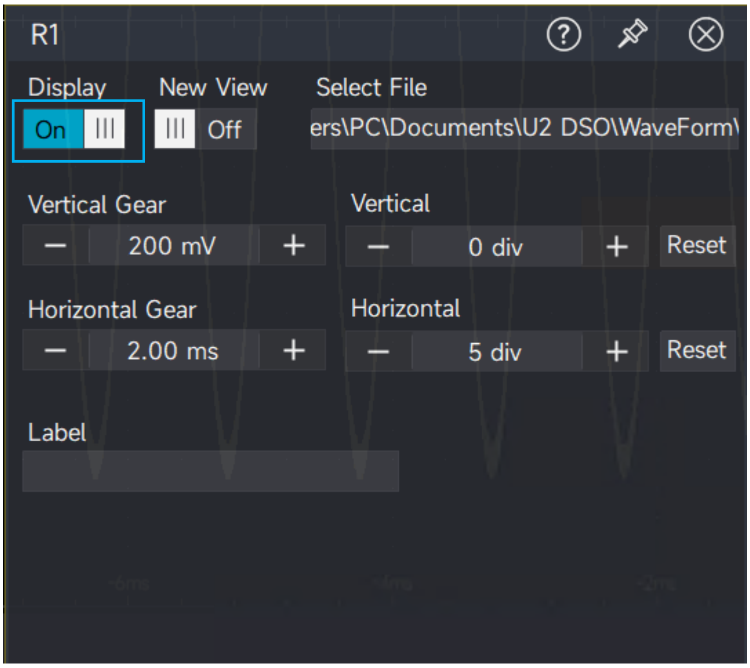
The user can configure the reference waveform using the following steps.

1. Open or close the display of the current reference waveform.
2. Set the current reference waveform to display in a separate window.
3. Switch the reference waveform file.
4. Adjust the vertical scale and position of the reference waveform.
5. Adjust the horizontal scale and position of the reference waveform.
6. Add a label for the current reference waveform.



10.3 Close Ref Waveform

1. Close the display of the reference waveform, which deletes the reference waveform.
2. Use the mouse to slide down the channel label of the reference waveform to close it.



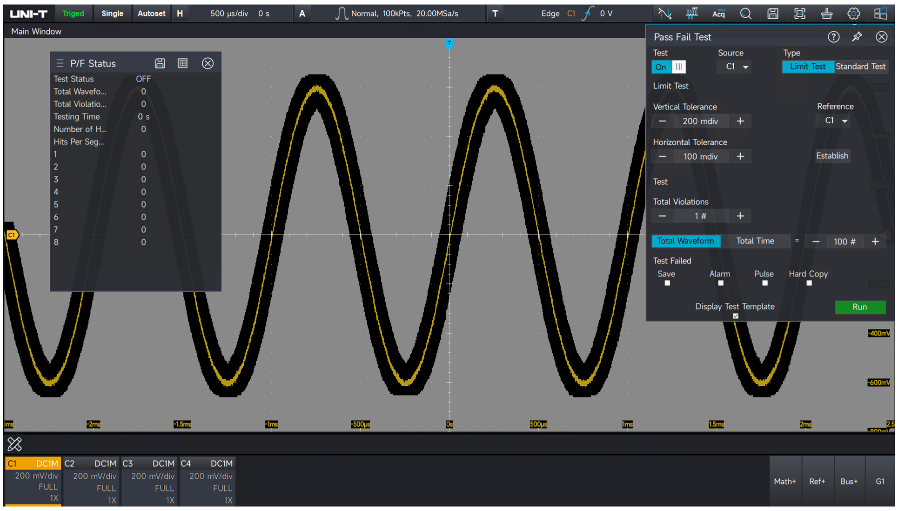
11. Pass/Fail Test

- Limit Test
- Standard Test Template

In the process of product design and production, it is often necessary to monitor signal fluctuations or determine whether the product meets the required specifications. This oscilloscope features a pass/fail test function that can greatly facilitate this task.

By checking whether the input signal falls within the template, the oscilloscope determines signal compliance. It can be used to detect abnormal waveforms or conduct production line tests. The test results can be displayed on the screen or indicated by the pulse signal output from the "AUX OUT" on the rear panel.

11.1 Limit Test

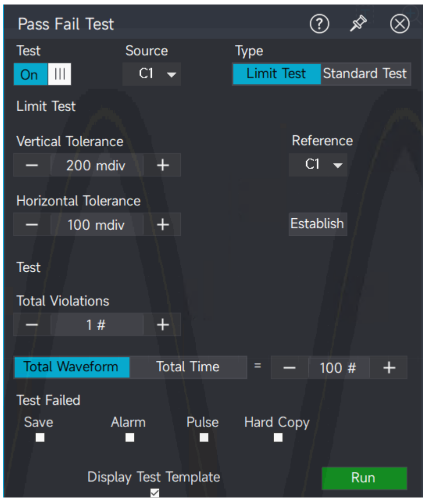


(1) Create template for limit test

Click the Start menu > P/F Test > Pass/Fail test menu.

1. Turn on/off the test.
2. Select the test source (C1-C4).
3. Select the test type – limit test.
4. Set the template for the limit test:

- Select the reference source (C1-C4).
- Set the vertical tolerance (range: 1mdiv to 1div).
- Set the horizontal tolerance (range: 1mdiv to 500mdiv).
- Click "Create".



(2) Set the end condition of the test

- Set the number of violations (range: 1 to 1k).
- Select the total waveform count and the time for the test.

  Allowable testing range for total waveforms: 1 to 100k.

  Allowable testing range for total time: 100ms to 1Ms.

(3) Set test failure actions: Save, Pulse, Alarm, Hard Copy

- Save: Save the waveform screenshot of the failed test.
- Alarm: An alarm will be triggered when the number of violations reaches the set limit.
- Pulse signal: A pulse signal will be output from the Aux Out on the rear panel when the test fails.
- Hard copy: Save the waveform screenshot of the failed test.

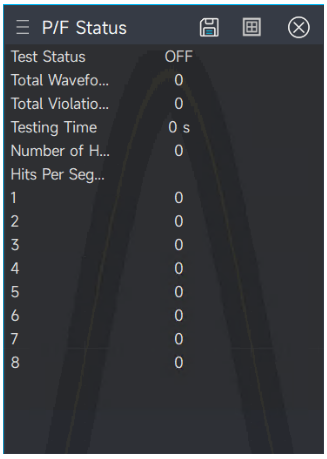
(4) Test template

Set whether to display the test template: ☑ indicates display the test template; ☐ indicates the test template will not be displayed.

(5) Click "Run"

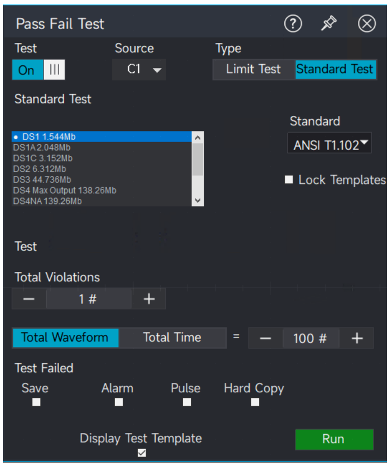
(6) P/F state

The P/F status label displays the current test state (Running/Off), the current total waveform count, the number of violations, test time, the target number, and the target value in each field.



Note: After setting the template conditions, click “Run” to start the P/F template test, or click “Stop” to end the operation.

11.2 Standard Test Template



(1) Create test template

Click the Start menu > P/F Test > Pass/Fail test menu.

1. Turn on/off the test.
2. Select the test source (C1-C4).
3. Select the test type – standard test.
4. Select the standard： ANSI T1.102, ITU-T, USB

(2) Set the end condition of the test

- Set the number of violations (range: 1 to 1k).
- Select the total waveform count and the time for the test.

  Allowable testing range for total waveforms: 1 to 100k.

  Allowable testing range for total time: 100ms to 1Ms.

(3) Set the operation for fail test: Save, Alarm, Pulse Output, Hard Copy, and LabNoteBook.

(4) Test template

Set whether to display the test template: ☑indicates display the test template; ☐ indicates the test template will not be displayed.

(5) Click “Run”

(6) P/F state

The P/F status label displays the current test state (Running/Off), the current total waveform count, the number of violations, test time, the target number, and the target value in each field.

Note: After setting the template conditions, click “Run” to start the P/F template test, or click “Stop” to end the operation.

12. Digital Voltmeter and Frequency Meter

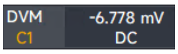
- Digital Voltmeter
- Frequency Meter

UPO7000L series features a built-in 4-digit digital voltmeter and an 8-digit high-precision frequency meter for accurate measurements. This chapter introduces how to use the digital voltmeter and frequency meter.

12.1 Digital Voltmeter

The digital voltmeter measurements in the UPO7000L series oscilloscopes are asynchronous with the oscilloscope's acquisition system and are always active.

(1) Turn on/off digital voltmeter

Click the Measurement Menu>Digital Meter with the mouse at the bottom left corner to enable/disable the digital measurement function. The measured results of the digital voltmeter 

will be displayed at the bottom right corner on the screen. Click the digital measurement menu to toggle the digital voltmeter display ON or OFF.

Note: The digital voltmeter shares the same probes as the oscilloscope, so the unit for the digital voltmeter measurement is consistent with the channel's unit.

(2) Select source

The measurement source can be selected from C1 to C4.

(3) Select measurement mode: DC, AC RMS, and DC+AC RMS.

DC: Displays the average value of the acquired data.

AC RMS: Displays RMS of acquired data with the DC component removed.

DC+AC RMS: Displays RMS of acquired data.

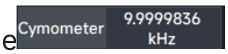
(3) Auto range

Enable the auto range function.

12.2 Frequency Meter

The counter measurement of frequency meter can be performed on the analog channels C1-C4.

Turn on/off frequency meter:

Click the Measurement Menu>Frequency Meter with the mouse at the bottom left corner to enable/disable the counter measurement function. The 

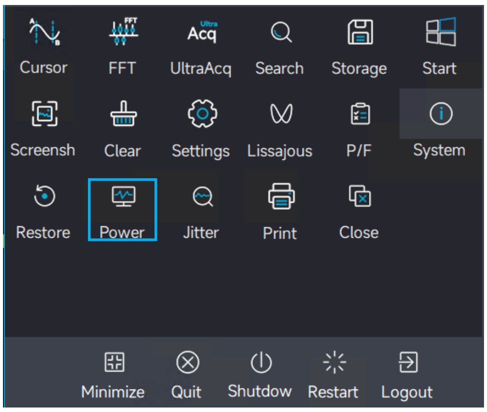
measured results of the frequency meter will be displayed at the bottom right corner on the screen.

13. Power Analysis (Option)

UPO7000L series supports power supply analysis, helping engineers assess the efficiency and reliability of switching power supplies. It also supports power quality analysis, including harmonic analysis, ripple analysis, switching loss, safety operation area, and loop analysis.

The power analysis function requires the differential voltage probe (UT-P3X series high-voltage differential probe), current probe (UT-P4X series), test fixture, and the oscilloscope's advanced measurement and analysis option (UPO7000L-PWR). For more details on this option, refer to the *UPO7000L Series Digital Phosphor Oscilloscope-Data Sheet.*

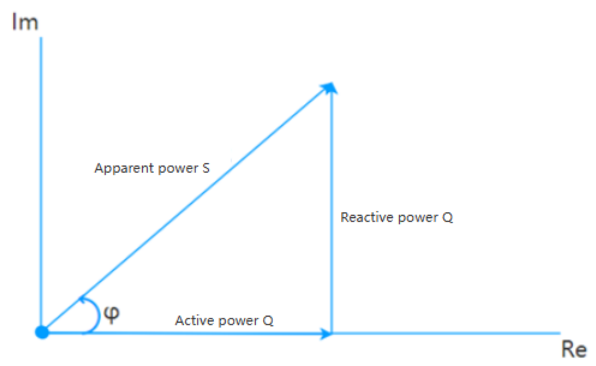
To access the power analysis function, click the Start Menu in the top right corner, open the function menu, and select "Power Analysis."



13.1 Power Quality Analysis

By measuring the input voltage, current, and generated power, the test results provide insights into the quality of the input AC line.

The analysis parameters include voltage RMS, voltage crest factor, frequency, current RMS, current crest factor, active power, apparent power, reactive power, power factor, and phase angle.



Voltage: Measurement of the voltage at the power input terminal.

Voltage RMS: The root mean square (RMS) voltage of the input AC power.

Voltage crest factor: The ratio of the peak value of the input AC power voltage to its RMS voltage. The crest factor affects the accuracy of AC measurements.

Voltage frequency: The frequency of the input AC power voltage.

Current: Measurement of the current at the power input terminal.

Current RMS: The root mean square (RMS) current of the input AC power.

Current crest factor: The ratio of the peak value of the input AC power current to its RMS current. The crest factor affects the accuracy of AC measurements.

Power: Measurement of the power at the power input terminal.

Effective power: The actual amount of power consumed by a power supply per unit of time. It represents the electrical power that converts electrical energy into other forms of energy, measured in watts (W).

Apparent power: The product of the input voltage RMS and the input current RMS. It represents the power capacity delivered to the switching power supply, measured in volt-amperes (V/A).

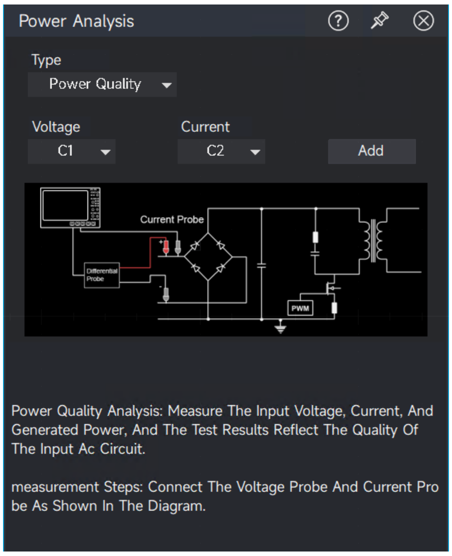
Reactive power: In AC circuits containing reactive components (capacitors and inductors), this is the electrical power required to establish alternating magnetic fields and induced flux. This energy is transferred between the power supply and the inductive elements but does not produce mechanical or thermal energy. The unit is volt-amps reactive (VAR).

Power factor: The ratio of effective power to apparent power. It represents the efficiency with which the switching power supply utilizes power. The lower the power factor, the higher the reactive power. In addition to reactive power generated by reactive components, high-frequency harmonic components from nonlinear devices also contribute to reactive power.

Phase angle (φ): The phase difference between voltage and current in an AC power supply line, reflects the operating state of the switching power supply.

(1) Add power analysis

To access the power analysis function, click the Start Menu  in the top right corner, open the function menu, and select "Power Analysis". Then, choose "Power Quality" as the analysis type. Set the input voltage source and input current channel, and click "Add".

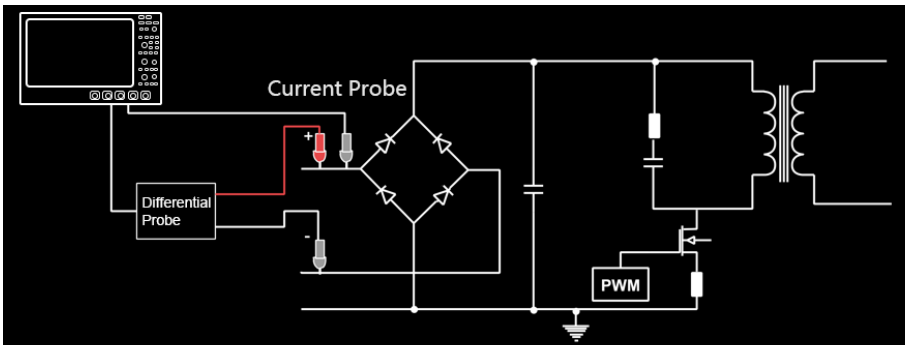


(2) Frequency reference

The input voltage source and the frequency of the input current serve as the frequency reference for calculating the phase angle (φ).

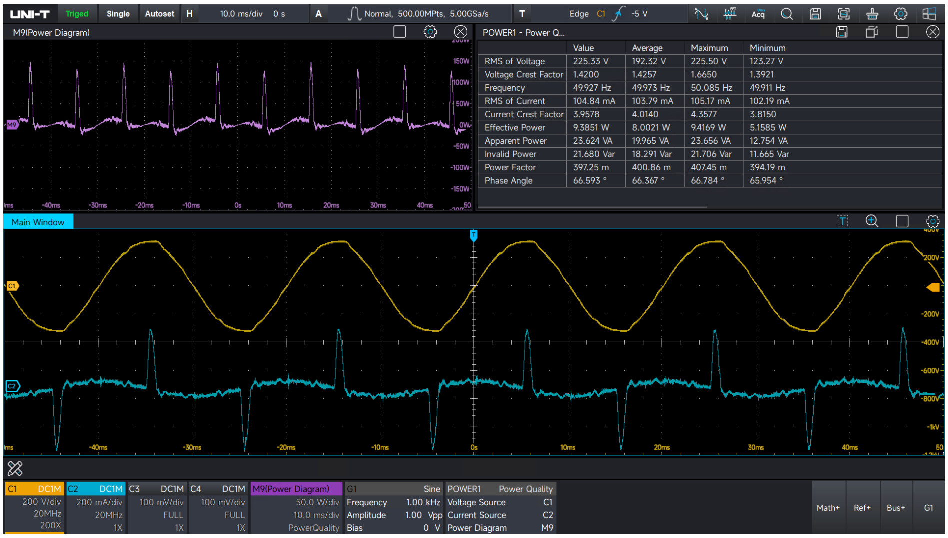
(3) Schematic diagram of signal connections

Connect the differential probe and current probe to the circuit under test as shown in the figure below.



(4) Power display

The oscilloscope displays the voltage and current waveforms, along with the calculated power waveform.



(5) Measured result table

The input test results include the current value, average, maximum, and minimum values.

POWER1 - Power Quality

	Value	Average	Maximum	Minimum
RMS of Voltage	224.77 V	188.02 V	225.80 V	23.563 V
Voltage Crest Factor	1.4236	1.4413	2.4870	1.3908
Frequency	49.965 Hz	49.938 Hz	50.025 Hz	33.313 Hz
RMS of Current	103.97 mA	104.36 mA	106.43 mA	102.62 mA
Current Crest Factor	3.8624	3.9977	4.3577	3.7599
Effective Power	9.3130 W	7.8153 W	9.4169 W	969.42 mW
Apparent Power	23.371 VA	19.615 VA	23.826 VA	2.4571 VA
Invalid Power	21.436 Var	17.990 Var	21.898 Var	2.2525 Var
Power Factor	398.47 m	398.38 m	405.27 m	283.02 m
Phase Angle	66.517 °	66.522 °	73.558 °	66.091 °

13.2 Harmonic Analysis

Harmonic analysis measures the harmonics of the current and voltage at the power supply input. The harmonic values are obtained by testing the current and voltage on the input power line, where an FFT (Fast Fourier Transform) is performed to extract the harmonic components of the signal.

(1) Add harmonic analysis

To access the power analysis function, click the Start Menu  in the top right corner, open the function menu, and select "Power Analysis". Then, choose "Harmonic Analysis" as the analysis type. Set the input voltage source and input current channel, and click "Add".

(2) Harmonic type

The harmonic type can be set to odd harmonics, even harmonics, or all harmonics.

(3) Harmonic source

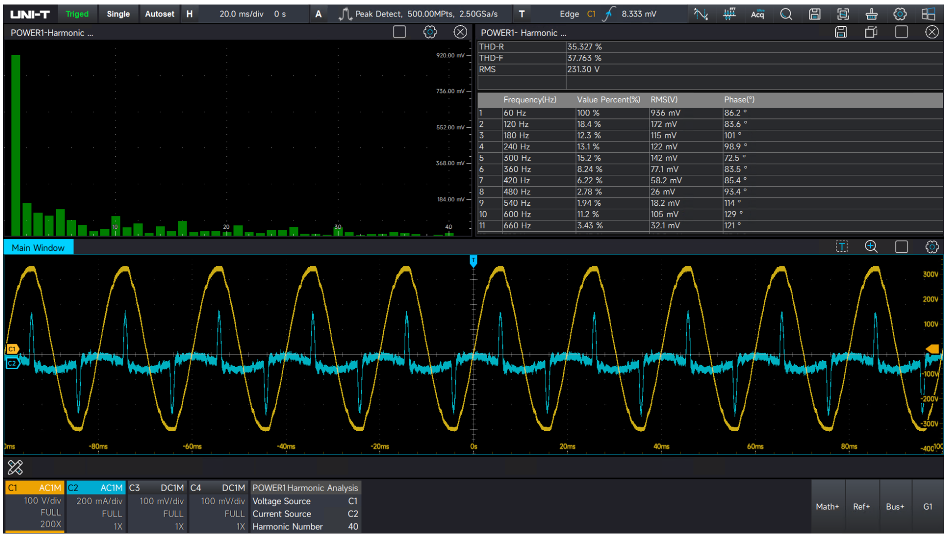
The harmonic source can be set to either the voltage source or the current source.

(4) Number of harmonics

Select the number of harmonics to be displayed. After selecting the number, the measured result table and histogram will be updated with the measurement results.

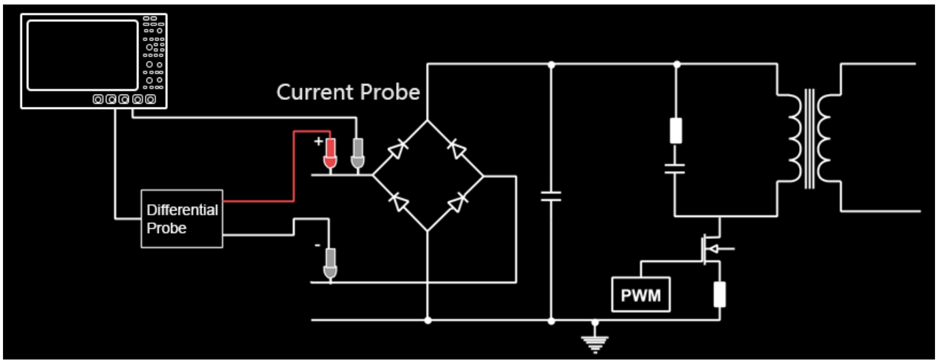
(5) Measured result table

The harmonic analysis test results include the current value, average, maximum, and minimum values.



(6) Schematic diagram of signal connections

Connect the voltage probe and current probe to the circuit under test as shown in the figure below.



13.3 Ripple Analysis

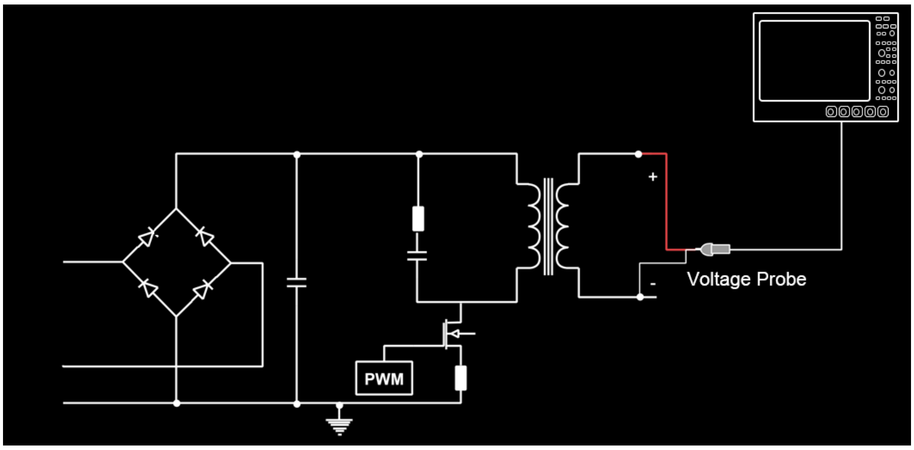
Power supply ripple is a critical parameter for evaluating DC power supplies. By measuring the peak-to-peak value of the DC signal output, the voltage regulation capability and filtering quality of the DC power supply can be assessed. Ripple analysis parameters include the current value, average, maximum, and minimum values.

(1) Add ripple analysis

To access the power analysis function, click the Start Menu  in the top right corner, open the function menu, and select "Power Analysis". Then, choose "Ripple Analysis" as the analysis type. Set the input voltage source and input current channel, and click "Add".

(2) Schematic diagram of signal connections

Connect the voltage probe and current probe to the circuit under test as shown in the figure below.

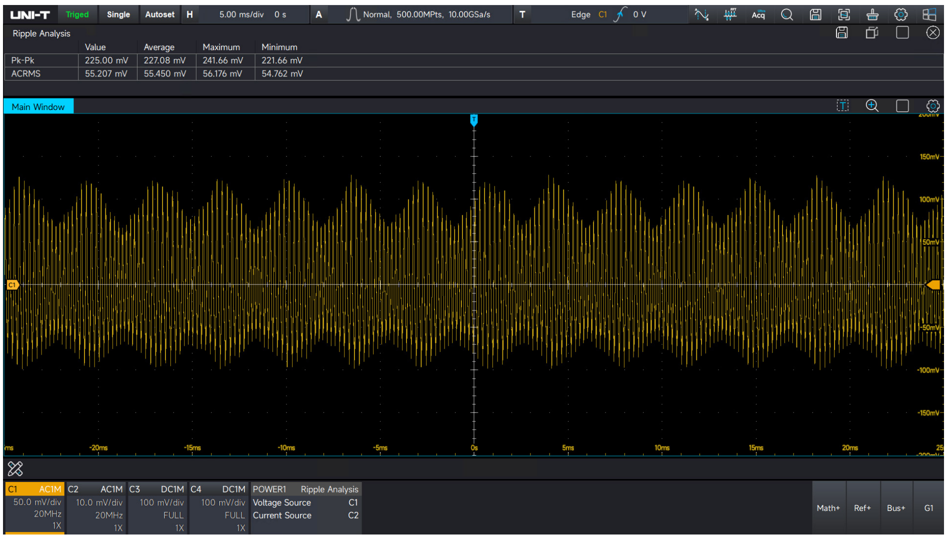


(3) Ripple source

The ripple source can be set to either the voltage source or the current source.

(4) Measured result table

The ripple analysis test results include the current value, average, maximum, and minimum values.



13.4 Switching Loss

The internal losses of a switching power supply can be categorized into switching loss, conducting loss, additional loss, and resistance loss. These losses typically occur simultaneously in lossy components, with power switches being one of the most significant sources of loss in a typical switching power supply.

Switching loss analysis measures the power and energy losses of a switching device during both the switching and conduction phases of a transistor. The analysis parameters include open power loss, conducting power loss, close power loss, non-conducting power loss, total power loss, open energy loss, conducting energy loss, close energy loss, non-conducting energy loss, total energy loss, and number of switching cycles.

(1) Probe degaussing and zero clearing

Before performing switching loss analysis, the current probe must be degaussed and zeroed to ensure accurate measurements.

(2) Time-delay calibration

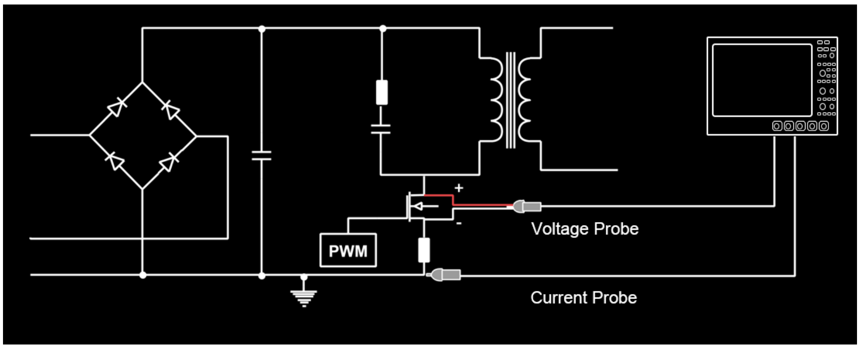
A smaller time delay can result in larger switching loss measurement errors. Time-delay calibration corrects the time delay of the oscilloscope or probe. This calibration only needs to be performed once; however, it should be repeated if any part of the hardware setup is changed (e.g., changing probes, switching oscilloscope channels) or if there is a significant change in the environmental temperature.

(3) Add switching loss

To access the power analysis function, click the Start Menu in the top right corner, open the function menu, and select "Power Analysis". Then, choose "Switching Loss" as the analysis type. Set the input voltage source and input current channel, and click "Add".

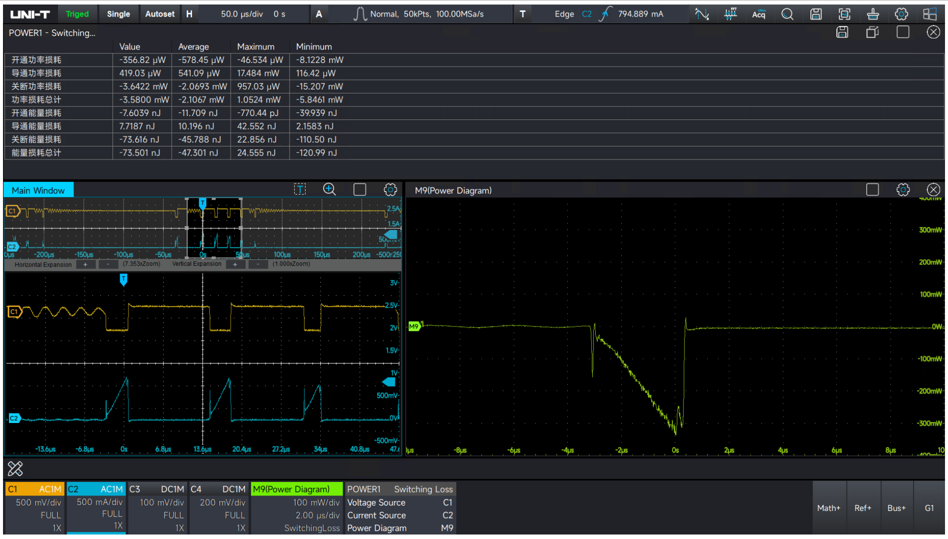
(4) Schematic diagram of signal connections

Connect the voltage probe and current probe to the circuit under test as shown in the figure below.



(5) Power display

The oscilloscope displays the voltage and current waveforms, along with the calculated power waveform.



(6) Measured result table

The switching loss test results include the current value, average, maximum, and minimum values.

POWER1 - Switching...

	Value	Average	Maximum	Minimum
Turn-on Power Loss	-460.05 μW	-541.43 μW	-187.79 μW	-16.528 mW
Conduction Power Loss	413.58 μW	544.98 μW	23.356 mW	292.60 μW
Shutdown Power Loss	-3.9633 mW	-2.1071 mW	-270.99 μW	-17.484 mW
Total Power Loss	-4.0098 mW	-2.1036 mW	979.27 μW	-10.656 mW
Turn-on Energy Loss	-9.6564 nJ	-10.819 nJ	-4.3809 nJ	-44.835 nJ
Conduction Energy Loss	7.5924 nJ	10.217 nJ	35.854 nJ	6.4440 nJ
Shutdown Energy Loss	-79.121 nJ	-46.463 nJ	-5.4415 nJ	-104.73 nJ
Total Energy Loss	-81.185 nJ	-47.064 nJ	-6.0681 nJ	-111.12 nJ

13.5 Safety Operation Area

The safety operation area (SOA) is defined by the X-Y mode of the switching device's voltage and current. The SOA template test provides Pass/Fail results, allowing power device violations at different loads to be directly observed within the safety operation area.

(1) Add SOA

To access the power analysis function, click the Start Menu  in the top right corner, open the function menu, and select "Power Analysis". Then, choose "SOA" as the analysis type. Set the input voltage source and input current channel, and click "Add".

(2) SOA coordinate type

The SOA coordinate type can be set to either linear or logarithm.

(3) SOA template

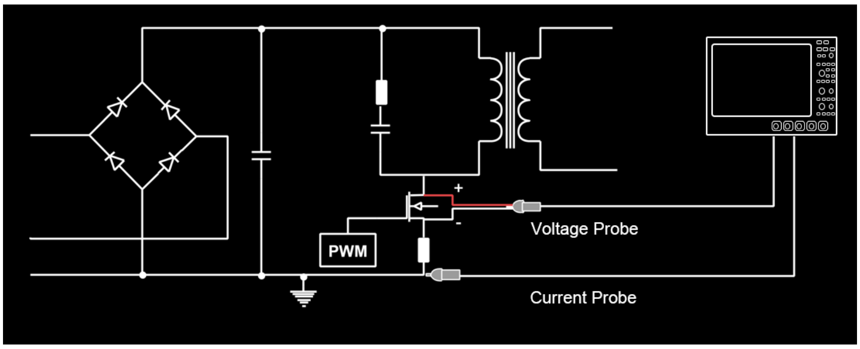
Set the maximum voltage, current, and power.

(4) SOA template display

Set the X-axis maximum voltage, X-axis minimum voltage, Y-axis maximum voltage, and Y-axis minimum voltage.

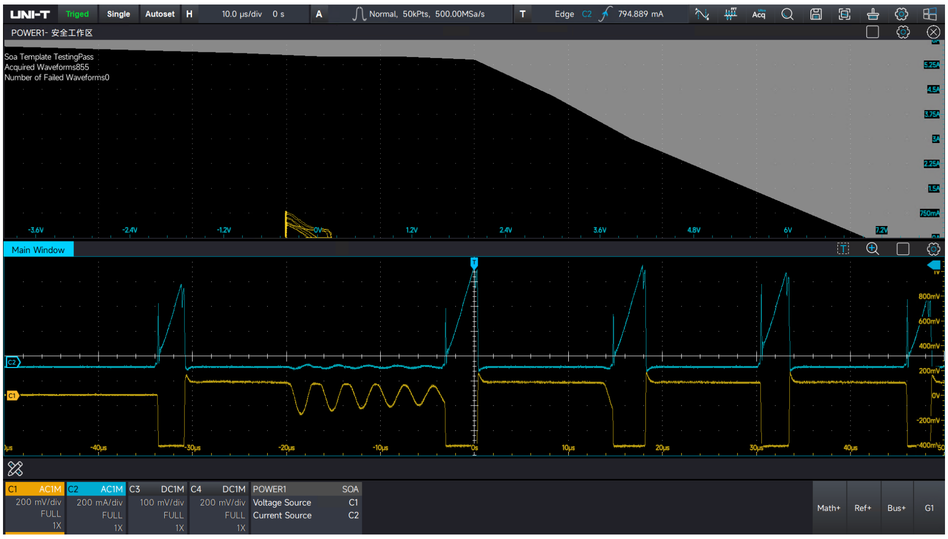
(5) Schematic diagram of signal connections

Connect the voltage probe and current probe to the circuit under test as shown in the figure below.



(6) SOA waveform display

When SOA is enabled, SOA waveforms are displayed in the template test area. The user can visualize whether the SOA waveforms violate the template, as well as view the SOA template test results, the number of captured waveforms, and the number of failed waveforms.



(7) Stop and reset

In SOA test, set to automatically stop and resume the test when the violation occurs.

13.6 Loop Analysis

The loop analysis involves inputting a positive wave signal with varying frequencies into the switching power supply circuit as an interference signal. The dynamic adjustment ability of the circuit system to each frequency of the interference signal is evaluated based on its output response.

This test requires the UPO7000L-AWG option (function/arbitrary waveform generator) to output interference signals at different frequencies. Additionally, a signal injector or an isolation transformer is needed to input the signal into the circuit system, and two voltage probes are used to detect the input and output signals in the loop circuit.

During the scan, the oscilloscope will automatically configure the function/arbitrary waveform generator to output the signal and connect it to the DUT. It then compares the DUT's input signal to its output signal, measures Gain and Phase at each frequency, and polts them on a Bode diagram.

(1) Add loop analysis

To access the power analysis function, click the Start Menu  in the top right corner, open the function menu, and select "Power Analysis". Then, choose "Loop Analysis" as the analysis type. Set the input voltage source and input current channel, and click "Add".

(2) Sweep mode

The sweep mode can be set either continuous or single sweep.

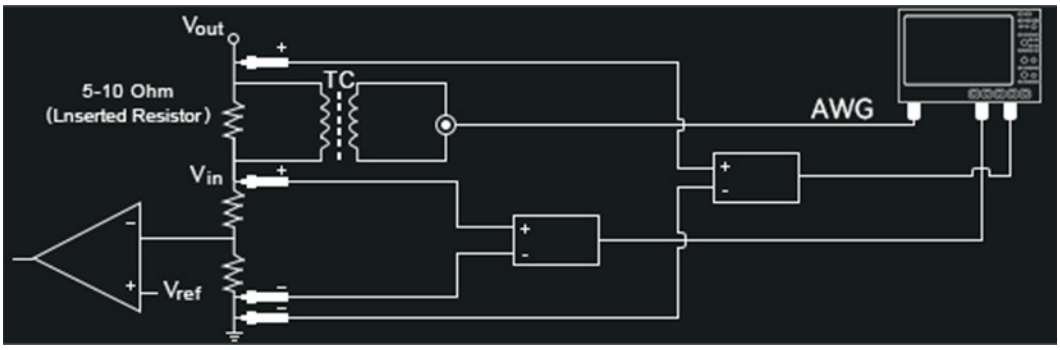
(3) Impedance

The impedance can be set either 50Ω or high impedance. The setting depends on the resistance value of the loop.

Note: The injection resistor is inserted into the loop and must not affect the steady-state value of the loop.

(4) Schematic diagram of signal connections

Connect the voltage probe 1, voltage probe 2, and AWG output signal to the circuit under test as shown in the figure below.



(5) Start/cut-off frequency

The start frequency and cut-off frequency should not exceed the output frequency of the function signal generator. The cut-off frequency of the loop system is recommended to be set between 1/20 and 1/6 of the switching frequency. Within this range, the crossing frequency point of the loop can generally be found. The crossing frequency should not be too low, as it will prevent the loop from responding to high-frequency load fluctuations, which may cause noise in the output voltage.

(6) Sweep point

The sweep point can be set from 1 to 1000. The higher the point, the slower the sweeping speed.

(7) Output amplitude of signal

The output amplitude of the signal can be set to either constant amplitude or adjustable amplitude. The amplitude of the injected signal can be set to 5% of the output voltage. If the amplitude is too small, the oscilloscope may not be able to detect it. Conversely, if the amplitude is too large, it may cause nonlinearities in the system, leading to measurement distortion.

(8) Click to run analysis

(9) Bode plot

The Bode plot provides the frequency response curve of the DUT.

(10) Measured result table

The loop analysis test results include frequency, amplitude, gain, and phase displays.

14. Jitter Analysis and Eye-diagram (Option)

Eye diagram and jitter analysis are essential tools for assessing signal integrity in high-speed interconnect systems, providing valuable information.

An eye diagram is a statistical distribution graph formed by stacking data bits at different positions of high-speed digital signals based on clock intervals. It reflects the overall characteristics of all digital signals in the transmission link.

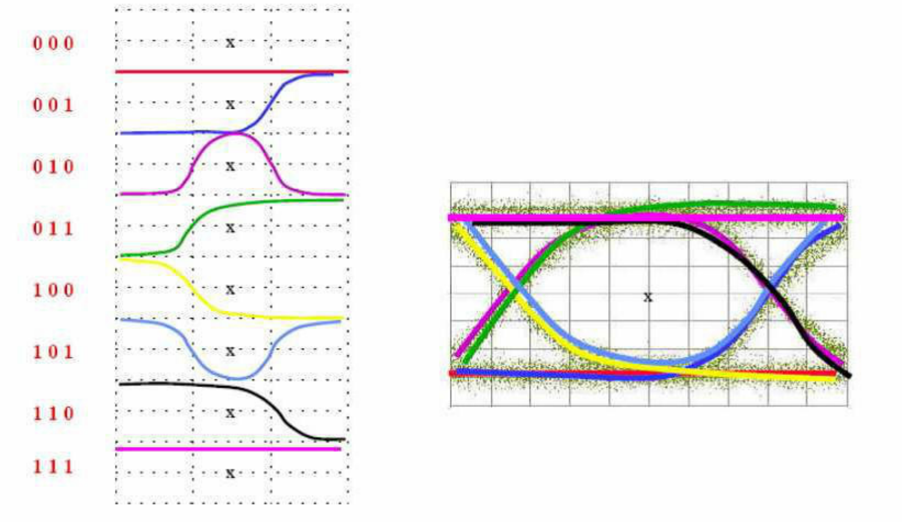
Jitter analysis refers to the noise and phase changes that occur on the edges of a signal, which can cause timing errors. As signal rates increase, interference factors in the data transmission link also rise, making signal loss and transmission quality more critical. Therefore, designers must monitor signal quality during transmission from TX to RX. Jitter analysis is a tool that helps engineers perform this type of testing.

UPO7000L series provides a suite of tools for visual jitter analysis and eye diagram testing, delivering a comprehensive analysis of signal quality in the time, frequency, and statistical domains. Through eye diagram tests, users can gain insight into the complete characteristics of a digital signal. The TIE (Time Interval Error) histogram shows the distribution of jitter, while the TIE tendency chart displays its trend. Users can identify jitter at specific points using the TIE spectrogram, allowing for targeted design adjustments. Additionally, the bathtub curve assists in further evaluating jitter behavior.

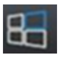
This chapter will introduce jitter and eye diagram testing in detail.

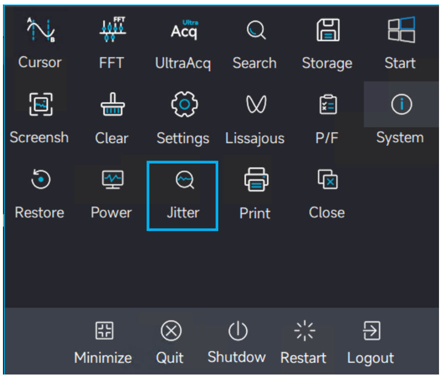
14.1 Eye-diagram

An eye diagram is a method of analyzing high-speed digital signals. The oscilloscope separates all the code elements of a digital signal and superimposes them on the screen at clock intervals, creating an eye-like effect.



Rapid generating an eye diagram

Click the start menu  at the bottom right corner to open the function menu and select "Jitter Analysis".



(1) Click to enter the jitter analysis measurement menu, enable jitter analysis, and connect the digital signal to the oscilloscope's channel.

(2) Select the analysis source: C1–C4.

(3) Select the analysis signal: Data signal, clock signal.

(4) Set the comparison threshold, delay, and data mode length.

Comparison threshold: 45%-55%, default is 50%.

Delay: 0%-30%, default is 20%.

Data mode length: 0-4.295Gbit, default is 127bits.

(5) Figure: Eye diagram

(6) Search for the bit rate or manually input the bit rate. Finding the bit rate automatically may not be accurate due to signal complexity, so manual input is recommended.

(7) Clock recovery

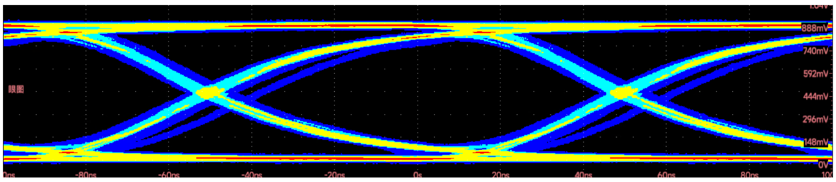
Constant clock: The least squares method is used to fit the collected data linearly, with a constant recovered clock frequency.

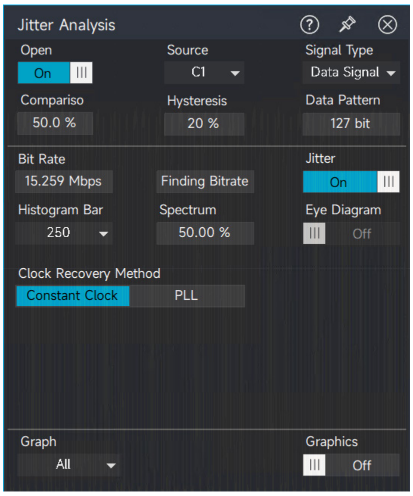
PLL (Phase-Locked Loop): Uses a software-based phase-locked loop to calculate the position of each reference clock edge. The PLL has tracking ability for clock changes, removes low-frequency jitter components, and supports Type 1 and Type 2.

Type 1 (First-order Golden PLL): Requires configuration of the cut-off frequency and coefficient of the phase-locked loop to set the loop bandwidth. If the cut-off frequency is less than 5 Gbps, the coefficient should be set to 1667 (empirical value); for cut-off frequencies greater than 5 Gbps, set it to 2500 (empirical value).

Type 2 (Second-order Golden PLL): Requires setting the natural frequency and damping factor.

(8) Open the eye diagram. The eye diagram is generated by superimposing long data bits. As a result, memory depth and sampling rate affect both the quality of the eye diagram and the drawing time.



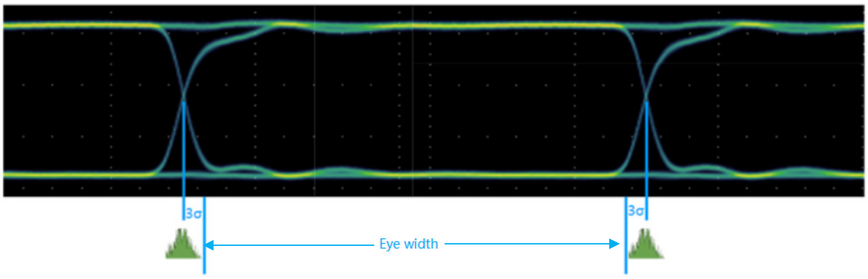
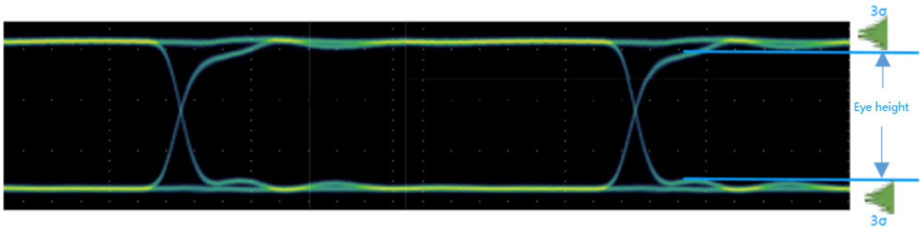


14.2 Measurement Parameter of Eye-diagram

Click the measurement parameter of eye-diagram to enable the eye-diagram function. Open the eye-diagram table for more details.

Eye

Items	Value
0 Level	26.667 mV
1 Level	966.667 mV
Eye Amplitude	940 mV
Eye Height	862.976 mV
Eye Width	51.541 ns
Extinction Ratio	14.718
Q-factor	36.612
Eye-crossing Rat	71.63 %

Parameter	Description
Eye Width	The width of the eye diagram is measured in the horizontal direction, based on the probability distribution of eye intersections in this direction. 
Eye Height	The height of the eye diagram is measured in the vertical direction, based on the probability distribution of the 1 level and 0 level within the 40% to 60% UI range. 
1 Level	The "1" level of the eye diagram is determined by taking the middle 20% of the UI in the vertical direction and calculating the average of the high points.
0 Level	The "0" level of the eye diagram is determined by taking the middle 20% of the UI in the vertical direction and calculating the average of the low points.
Eye Amplitude	The difference value between 1 level and 0 level.
Cross Ratio of Eye	The ratio of the amplitude at the intersection with the "0" level to the eye amplitude.

Extinction ratio	It reflects the noise immunity of the transmitted signal, defined as the ratio of the average power of the "1" level to the average power of the "0" level.
Q Factor	The ratio of the eye amplitude to the noise amplitude at the "1" level and "0" level.

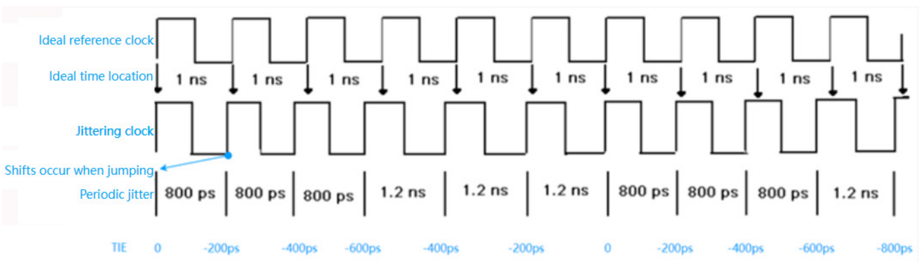
14.3 Jitter Analysis

The jitter analysis function is primarily used to analyze the signal integrity of high-speed serial systems. TIE jitter is a common measurement indicator.

UPO7000L provides a TIE tendency chart, TIE spectrogram, histogram, and bathtub curve for visualizing jitter, helping to further identify jitter conditions.

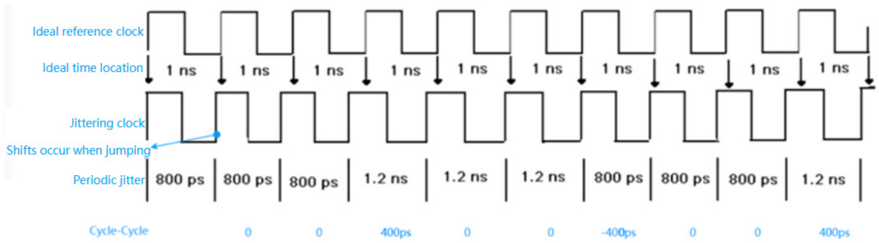
The operation of jitter analysis is the same as drawing an eye diagram, refers to the "Eye-diagram" section for more details.

TIE (Time Interval Error) represents the timing error of a signal relative to the reference clock. In high-speed digital systems, TIE refers to jitter, and in this oscilloscope, it specifically refers to the $TIE_{peak\text{-}peak}$. The measured signal's edge is compared with the ideal edge established by the clock recovery, and all signal intervals are measured based on the ideal data rate.



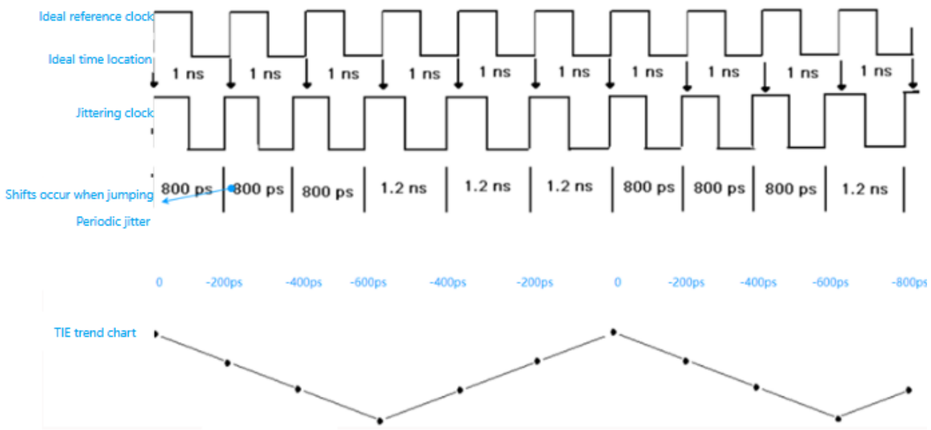
Cycle-Cycle

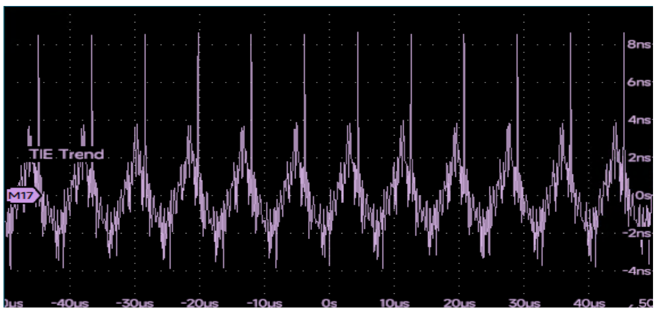
Measure the period of the first signal, then subtract the period of the first signal from the period of the second signal, and continue this process for subsequent signals.



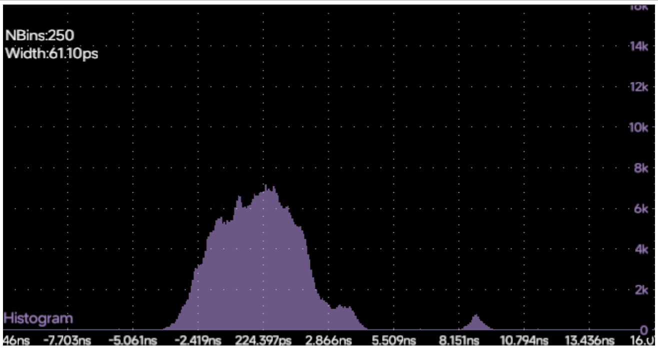
TIE tendency chart: The TIE tendency chart is derived from the time trend statistics of TIE jitter measurements and belongs to the time-domain analysis of jitter. The horizontal axis represents the time when the measurement occurred, while the vertical axis shows the value of the TIE jitter measurement.

The TIE tendency chart visualizes the jitter offset for each cycle, allowing for an understanding of the temporal trend of the signal's jitter.

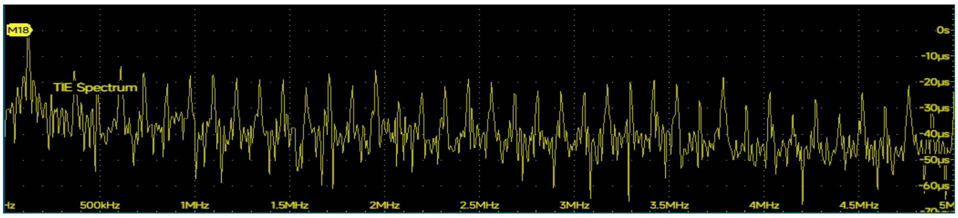




Jitter histogram: By statistically analyzing jitter offsets, the histogram presents the distribution of different jitter offsets, providing insight into the distribution of jitter in the clock signal. The horizontal axis represents the jitter offset, while the vertical axis shows the number of measurements accumulated at each offset.

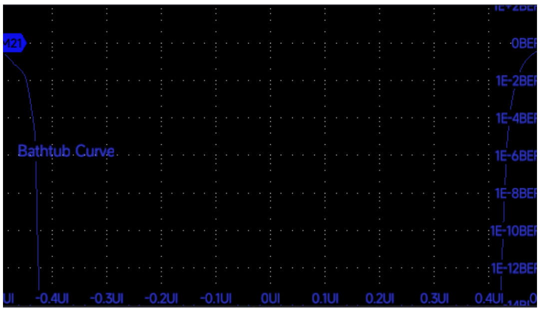


TIE spectrogram: The spectrum of the jitter is obtained by performing an FFT on the jitter, accurately localizing the distribution of jitter in the frequency domain. The horizontal axis represents the signal frequency, while the vertical axis represents the measured value of signal jitter.



Bathtub curve: This curve shows the variation of BER (bit error ratio) with the judgment moment, illustrating how BER changes with the cumulative number of occurrences. It allows for the analysis of the degree of eye diagram opening under different BER values. Typically, the degree of eye diagram opening is observed at a BER^{-12}, as jitter becomes meaningful only in relation to BER.

The horizontal axis represents the degree of eye diagram opening, measured in UI (Unit Interval), while the vertical axis represents the accumulated number of bits.

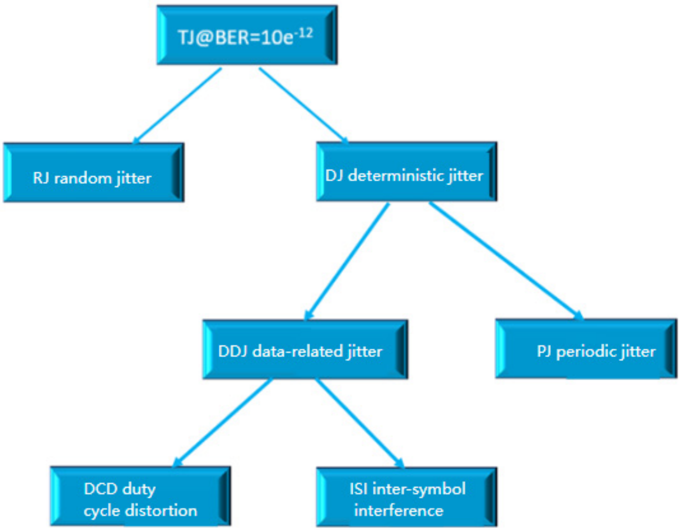


14.4 Clock Recovery

The method for setting clock recovery is the same as for the eye diagram, refers to the "Eye-diagram" section for more details.

14.5 Jitter Resolving

The decomposition of jitter is based on the acquired TIE (Time Interval Error) measurement data and is performed according to the different jitter components (as shown in the figure below).



Jitter Parameter	Description
TJ@BER=e^{-12}	The total jitter (TJ) is estimated based on the Bit Error Rate (BER): $TJ = DJ + 2Q_{BER} * \sigma_{RJ}$ The value of Q_{BER} varies depending on the BER. For example, when BER =10^{-12}, Q_{BER} = 7.05
RJ	Generally speaking, the probability density function (PDF) of random jitter (RJ) follows a Gaussian normal distribution. Theoretically, as the number of samples increases, the distribution range of the test also increases. If the sample size is sufficiently large, the distribution range approaches infinity, making it unbounded. The magnitude of the jitter is expressed by the standard deviation σ Random jitter primarily arises from internal thermal phenomena, vibrations of thermal molecules and atoms, mechanical noise, external cosmic rays, and other factors that cannot be eliminated.
DJ	The distribution of deterministic jitter is bounded.
DCD	Duty cycle distortion refers to the asymmetry between the rising and falling edges of a clock signal. When the clock duty cycle deviates from 50%, the rising and falling edges become unequal, and improper selection of the reference level can also contribute to duty cycle distortion.
DDJ	Data-dependent jitter (DDJ) primarily arises from Inter-symbol interference (ISI). Due to the non-ideal frequency response characteristics of the digital

	signal transmission channel, different code sequences passing through the channel can result in variations in the rising and falling edge amplitudes, leading to inconsistent crossing points. For example, a high-frequency signal pattern like “1, 0, 1, 0…” experiences significantly more attenuation than a continuous high-frequency signal such as “1, 1, 1, 1, 0, 0, 0, 0…”. As a result, it takes longer for the signal to reach higher levels when the code pattern changes, and more time to cross the threshold during transitions, leading to signal jitter. Additionally, impedance mismatches can cause signal reflections. The reflected signal, when combined with the original signal, can increase the amplitude and cause premature threshold crossing, further contributing to jitter. Consequently, the probability density function (PDF) of DDJ is represented as a series of discrete lines, reflecting the distinct variations in jitter.
ISI	Inter-symbol interference
PJ	Periodic jitter is primarily caused by periodic signal interference on the board, such as power ripple from the switching power supply and clock crosstalk. Periodic jitter can be considered equivalent to phase modulation.

Meaning of Jitter Dissolving:

Jitter dissolving allows the total jitter (TJ) at any bit error rate (BER) to be estimated from a finite sample of jitter measurements, thus saving test time, provided that the random jitter (RJ) is estimated accurately. By analyzing the cause behind each jitter component, it becomes possible to trace back to the source of any issue when a component is tested, which helps in quickly pinpointing the problem.

14.6 Measurement Parameter of Jitter

The measurement parameter of jitter refers to the section of "Jitter Resolving ".

14.7 Effect of Test System on Jitter Test

(1) Bandwidth of test system

The bandwidth of the test system encompasses the oscilloscope, probe, testing cable, and test fixture. Generally, the bandwidth of the test system (oscilloscope and probe) should be at least five times that of the signal being measured. This ensures that the fifth harmonic of the signal can be resolved. The testing cable and test fixture must meet the relevant standards and code requirements; failing to do so may introduce additional inter-symbol interference (ISI) jitter.

(2) Real-time sampling rate of the oscilloscope

Higher sample rates typically offer greater resolution and improved edge accuracy. These benefits are reflected in the jitter analysis results, where higher sampling rates correlate with higher measurement accuracy.

(3) Instrument ground noise and intrinsic jitter

Jitter, which refers to timing errors, can be influenced by the inherent jitter present in the instrument itself. Ground noise and the instrument's intrinsic jitter can affect the accuracy of the test results.

(4) Waveform sample number

The oscilloscope's memory depth is a critical factor in jitter analysis, as it directly impacts the number of samples available for jitter and eye diagram tests. Sufficient waveform data length allows for capturing lower frequency jitter and provides a longer clock period or data UI, which results in more accurate jitter analysis. This highlights the importance of an oscilloscope’s long memory storage capacity.

(5) Clock recovery

The method of clock recovery can influence the appearance of the eye diagram. With constant clock recovery, the clock frequency remains fixed, making it suitable only for signals with a stable clock. If there is a significant clock jitter or instability, measurement performance may degrade. In such cases, using PLL (phase-locked loop) clock recovery is recommended to ensure more accurate analysis.

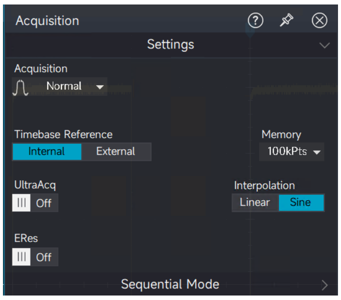
15. Sequence Mode

The sequence mode is a high-speed acquisition mode based on UltraAcq® technology, which divides the oscilloscope's memory depth into multiple segments, with a single triggered waveform stored in each segment. When the number of stored waveforms does not reach the preset frame count, the oscilloscope performs only acquisition and storage, without display or data processing. In sequence mode, the oscilloscope minimizes trigger dead time, significantly enhancing the waveform refresh rate. With UltraAcq® technology, the oscilloscope can achieve a minimum trigger interval of 1μs, corresponding to a waveform capture rate of 600,000wfms/s (waveforms per second).

15.1 Sequence Mode

(1) Enable sequence mode

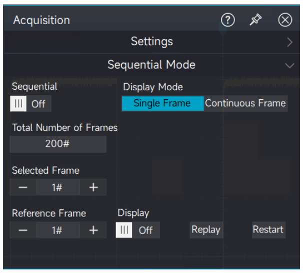
Click the acquisition, memory depth, and sampling labels to open the acquisition menu, then select "Sequence Mode".



(2) Display mode: Single frame, consecutive frame

Single frame: Displays one frame of the waveform on the screen.

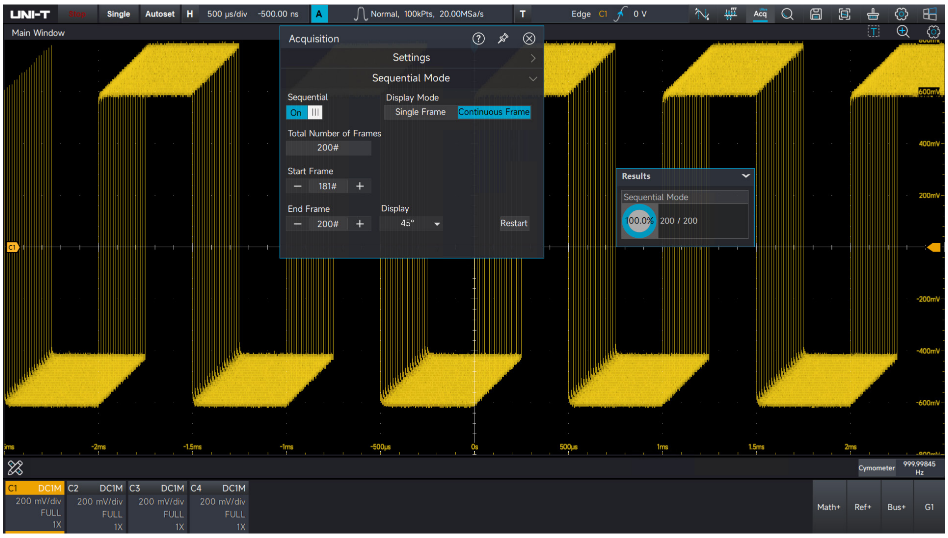
Consecutive frame: Set and display a range of frame numbers (up to 20 frames) on the screen simultaneously.



(3) Set total frame

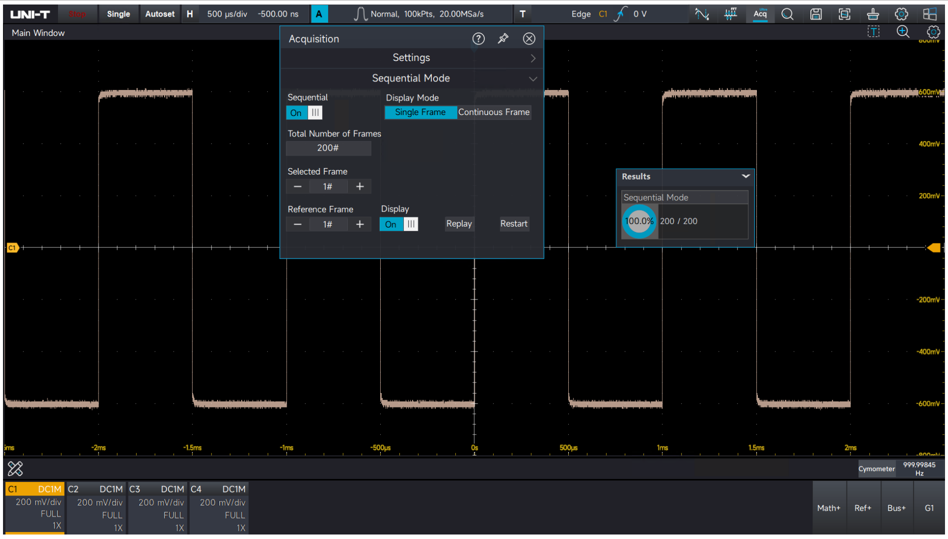
The total number of frames is related to the memory depth. The oscilloscope divides the memory depth into equal-length segments, with fewer segments created as the memory depth increases.

(4) Enable sequence mode



15.2 Single Frame

In single-frame mode, the reference frame can be displayed, the acquired waveform can be played back, and new acquisitions can be carried out. When the total frame is set or changes, the previously collected waveforms will be re-acquired. The oscilloscope will stop once the acquisition reaches the set total frame. At this point, a frame within the acquired range can be selected as the reference frame, which can then be played back. The reference frame is displayed in a different color to distinguish it from the other frames. During playback, the user can stop at any time if something unusual occurs, click the  and  icons or input the frame number to review the waveform.



15.3 Consecutive Frame

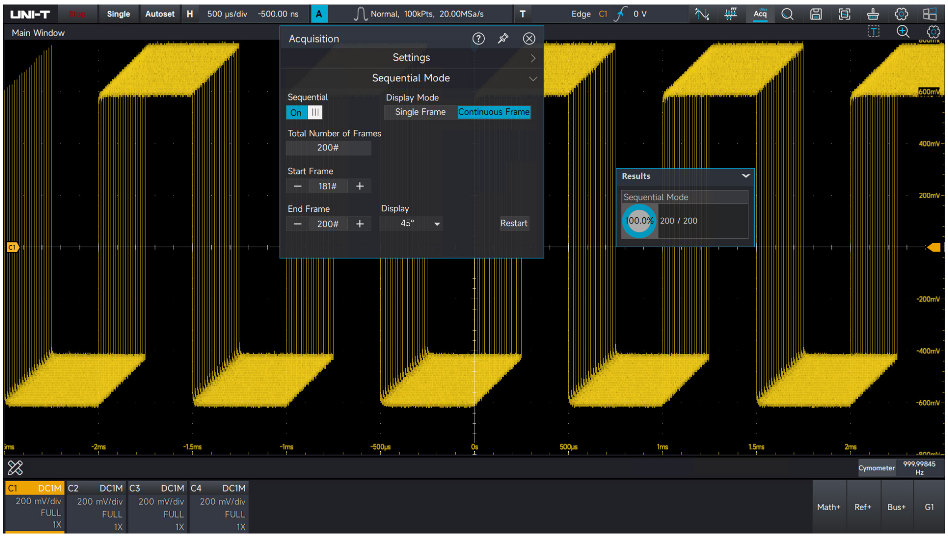
In consecutive frame mode, both the display range and the display type of the waveform can be set.

The display range of the waveform can be set from frame 1 to frame 20. The display mode can be selected from options such as 45°, stacking, superposition, and stitching.

For example, when setting the display range to a maximum of 20 frames, the waveform will be demonstrated in different display types.

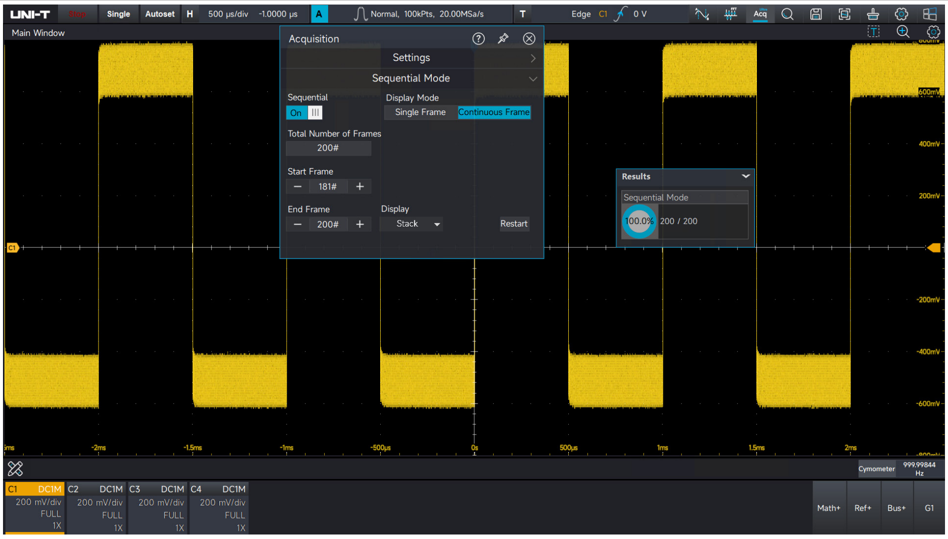
45°:

The waveform in the selected range is displayed at a 45° upward angle.



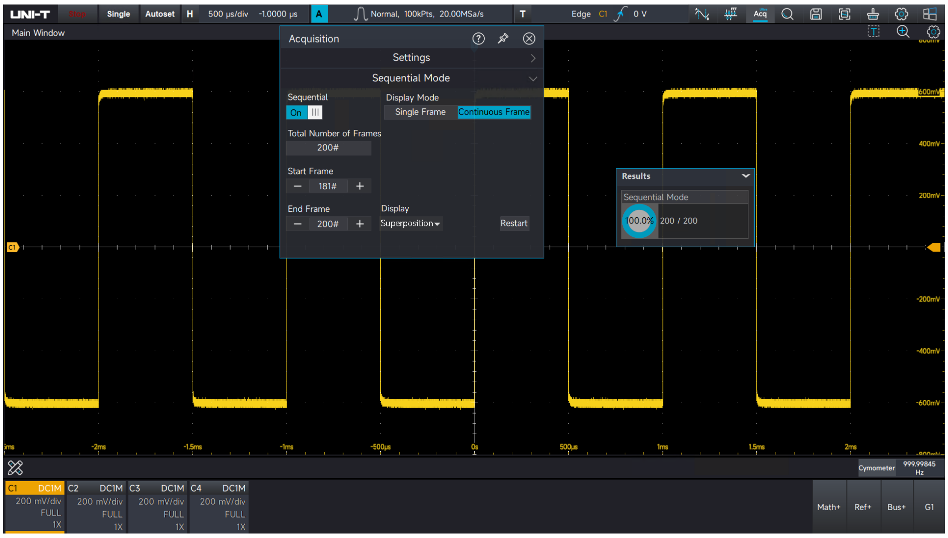
Stacking:

The waveforms within the range are stacked vertically for display.



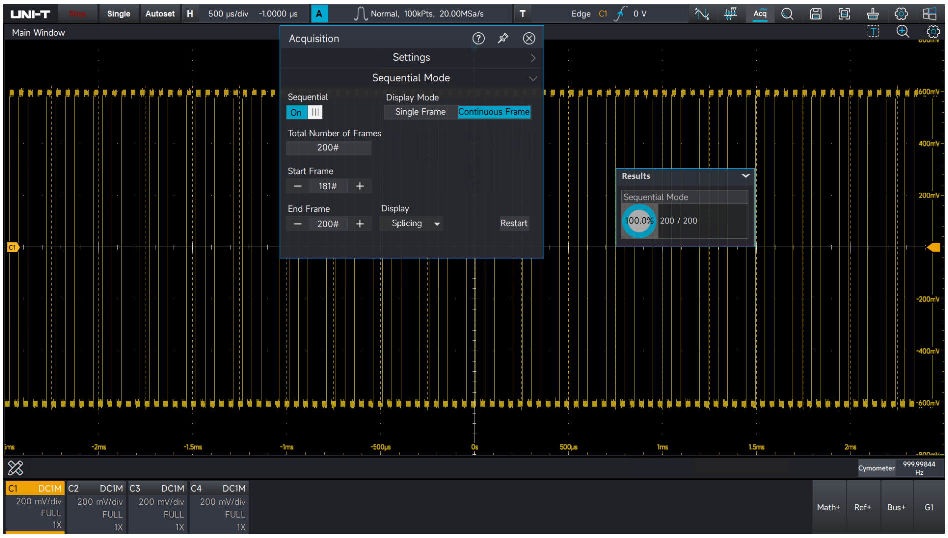
Superposition:

The waveforms within the range are overlaid on top of one another to display as a single waveform.



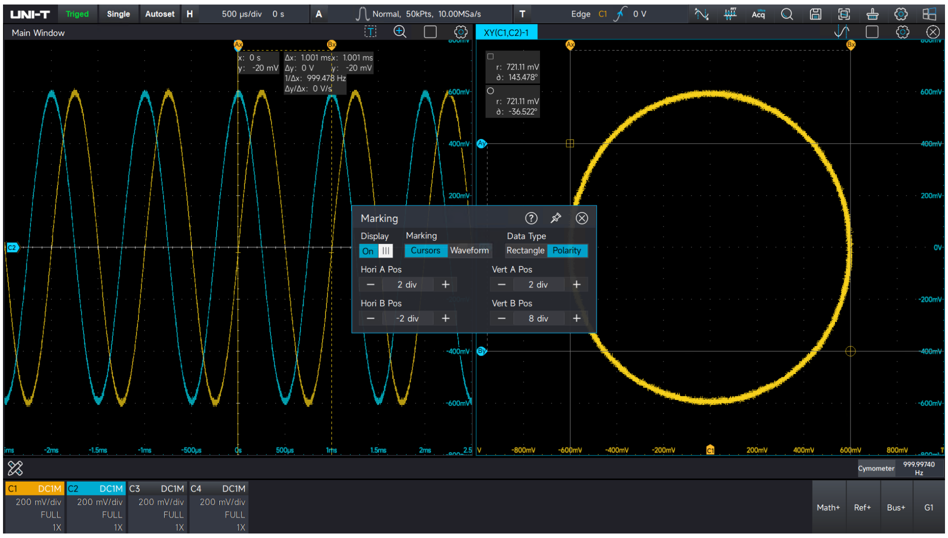
Stitching:

The waveforms within the range are displayed by stitching them together in sequence.

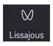


16. XY Mode

The waveform displayed in XY mode is referred to as a Lissajous curve.



(1) Quickly produce a Lissajous curve

Click the icon 

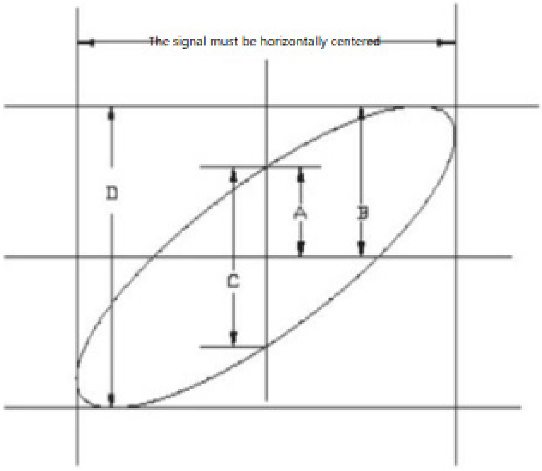
in the start menu to open Lissajous menu and quickly generate a Lissajous curve. Before generating the curve, set the input signals for the horizontal axis (X-axis) and the vertical axis (Y-axis). For example, assign C1 to the X-axis and C2 to the Y-axis.

(2) Adjust Lissajous waveform

- Horizontal movement: When the X-axis (C1) is selected, use the mouse to adjust and move the Lissajous curve horizontally.
- Vertical movement: When the Y-axis (C2) is selected, use the mouse to adjust and move the Lissajous curve vertically.
- Horizontal zooming: When the X-axis (C1) is selected, use the mouse to zoom in or out on the Lissajous curve horizontally.
- Vertical zooming: When the Y-axis (C2) is selected, use the mouse to zoom in or out on the Lissajous curve vertically.
- Centering the curve: Click the zero key to move the Lissajous curve to the center of the display, ensuring an optimal viewing effect for the curve.

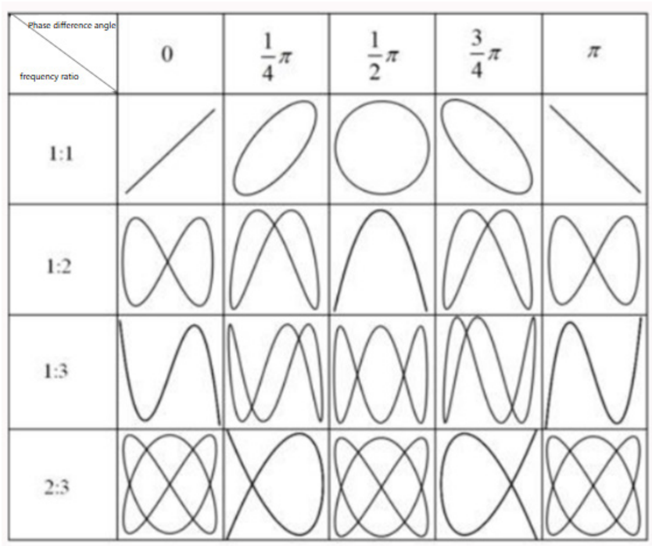
(3) Lissajous application

The phase difference between two signals with the same frequency can be easily observed using the Lissajous curve.



Based on sinθ=A/B or C/D, where θ is the phase angle between the channels (with A, B, C, and D defined in the figure above), the phase angle can be calculated as θ=±arcsin (A/B) or θ=±arcsin (C/D). If the major axis of the ellipse is in the I or III quadrant, then the resulting phase angle should be in the I or IV quadrant, i.e., within (0 - π/2) or (3π/2 - 2π). If the major axis of the ellipse is in the II or IV quadrant, then the phase angle should be within (π/2 − π) or (π - 3π/2).

Additionally, if the frequency or phase difference between the two signals is an integer multiple, calculate the frequency and phase relationship between the two signals based on the following figure.



17. Histogram

- Statistical Histogram
- Zone Histogram

UPO7000L series supports histogram analysis for trend judgment, providing an efficient tool for users to observe waveforms and analyze the probability distribution of measurement parameters. This functionality helps quickly identify potential signal anomalies. This functionality enables users to quickly identify potential signal anomalies.

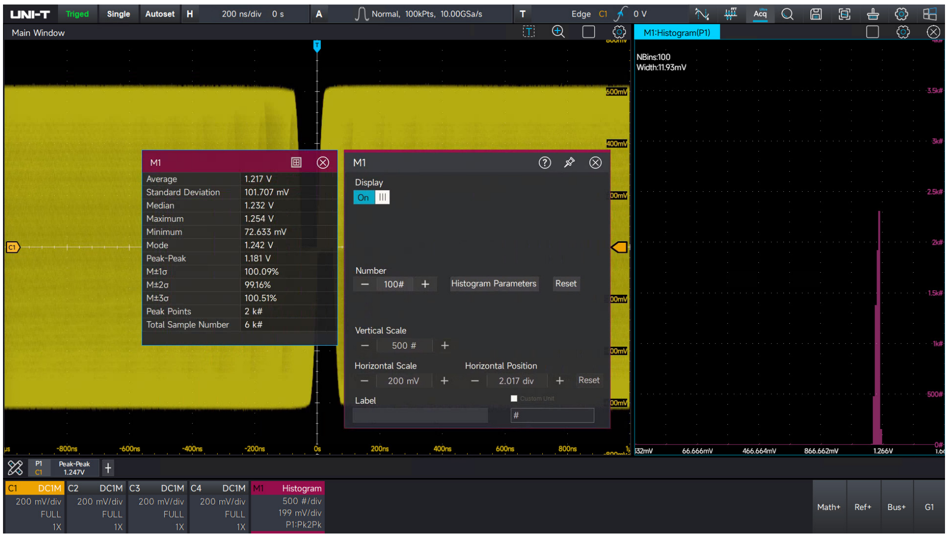
The histogram analysis is divided into two types:

Statistical histogram: Counts the number of samples for waveform measurement parameters, , providing a statistical overview of key measurement data.

Zone histogram: Analyzes waveform data distributions in both vertical and horizontal directions, offering insights into localized waveform behavior.

This chapter provides detailed instructions on how to use the histogram analysis function effectively.

17.1 Statistical Histogram

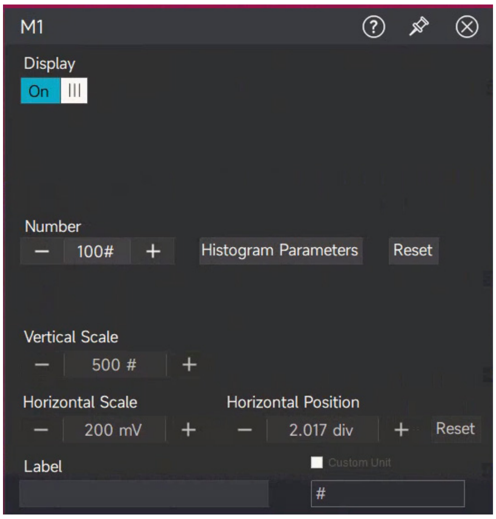


Turn on/ off statistical histogram

(1) Turn on statistical histogram: Click the parameter to be counted (ensure the parameter measurement is ticked), and select the histogram in the amplitude figure option.

(2) Turn off statistical histogram: Use the mouse to click on the histogram operation channel at the bottom to turn off statistical histogram function.

Alternatively, select the "None" option in the amplitude figure to turn off the statistical histogram.



Set the cylinder number of histogram

When the histogram is enabled, click the histogram operation channel at the bottom of the screen. In the settings menu, set the cylinder number. The range can be set from 2 to 2,000.

Set the vertical and horizontal scales

The vertical scale and horizontal scale of the histogram represent the vertical and horizontal axes of the statistical histogram.

The horizontal axis represents the unit of the current measurement parameter. For example, if frequency is being counted, the unit will be Hz; if amplitude is counted, the unit will be V.

The vertical axis represents the number of cylinders. The more cylinders there are, the higher the probability for that unit.

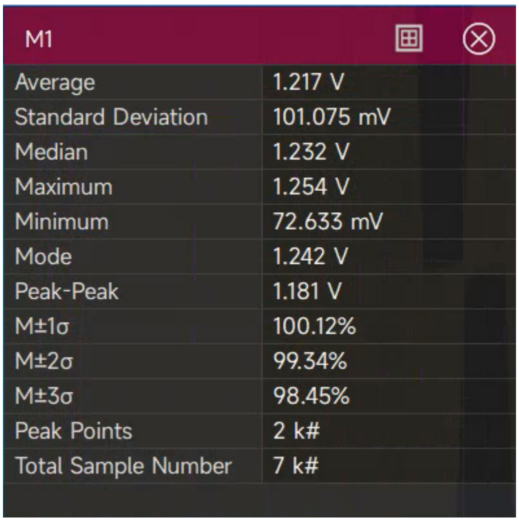
Note: The scales can be set to automatic, allowing the oscilloscope to adjust the vertical and horizontal scales of the histogram automatically.

Set the vertical and horizontal positions

Click the  and  icons to adjust the vertical and horizontal positions of histogram, with the unit division.

Histogram parameter table

The histogram parameter table can be opened from the settings menu. The parameters include average, standard deviation, median, maximum, minimum, mode, peak-to-peak, μ±1σ, μ±2σ, μ±3σ, peak count, and total sample.



M1	
Average	1.217 V
Standard Deviation	101.075 mV
Median	1.232 V
Maximum	1.254 V
Minimum	72.633 mV
Mode	1.242 V
Peak-Peak	1.181 V
M±1σ	100.12%
M±2σ	99.34%
M±3σ	98.45%
Peak Points	2 k#
Total Sample Number	7 k#

Set the histogram label and customized unit

(1) Histogram label

In setting menu, click the "Label" below the setting box to open the numeric keypad. Use the numeric keypad or an external physical keyboard to name the label. The label's color will be consistent with the color set in the histogram settings menu.

(2) Customized unit

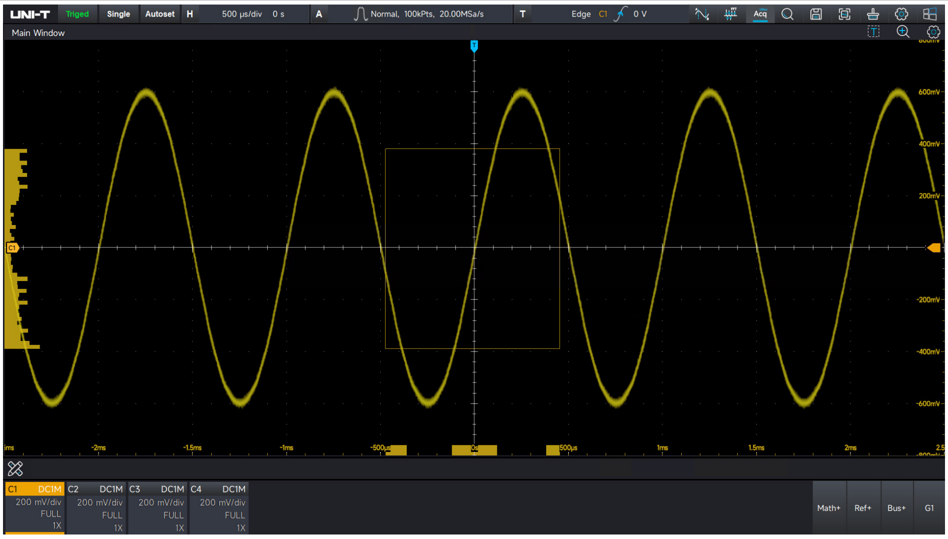
In setting menu, click and check the white box before "Custom Unit". Then, click "Custom Unit" below the setting box again to open the numerical keyboard, and set the customized unit for the histogram.

17.2 Zone Histogram

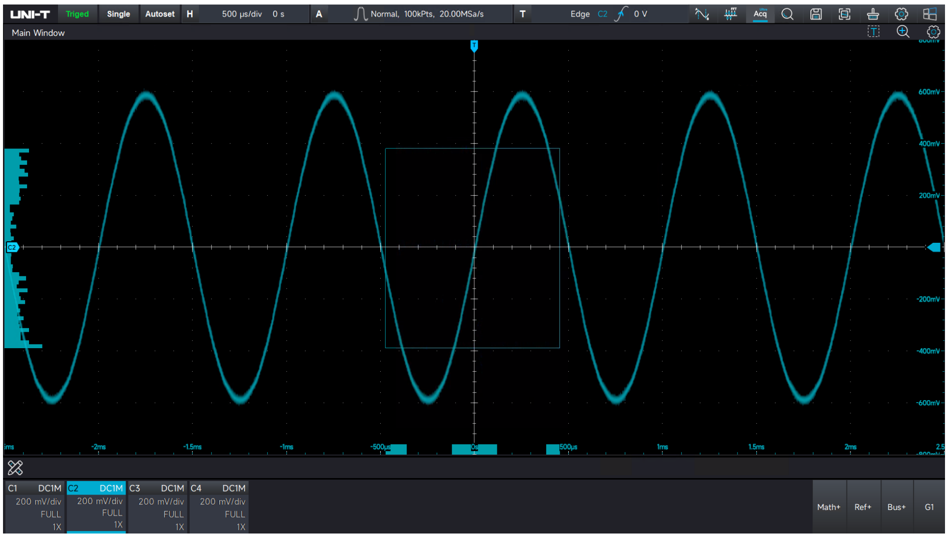
Zone histograms calculate probability distributions in the vertical and horizontal directions, which can quickly identify potential signal anomalies.

Click the icon  in the top right, a rectangular will appear. Select an edge of the rectangle to change its size. When the histogram is open, the histogram cylinder will display in both the horizontal and vertical directions. The larger the rectangle's range, the wider the range of statistics. The color of each channel's zone histogram is consistent with the channel color.

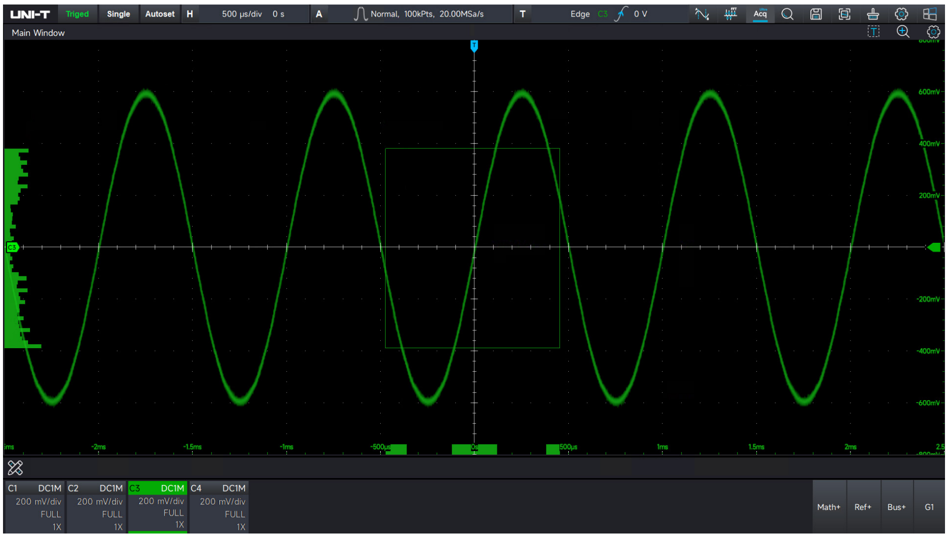
C1 will be displayed in yellow.



C2 will be displayed in in blue.



C3 will be displayed in green.



C4 will be displayed in purple.


Triged
Single
Autoset
H
500 µs/div 0 s
A
Normal, 100kPts, 20.00MSa/s
T
Edge
0 V
Acq
Main Window
C1 DC1M
200 mV/div
FULL
1X
C2 DC1M
200 mV/div
FULL
1X
C3 DC1M
200 mV/div
FULL
1X
C4 DC1M
200 mV/div
FULL
1X
Math+
Ref+
Bus+
G1

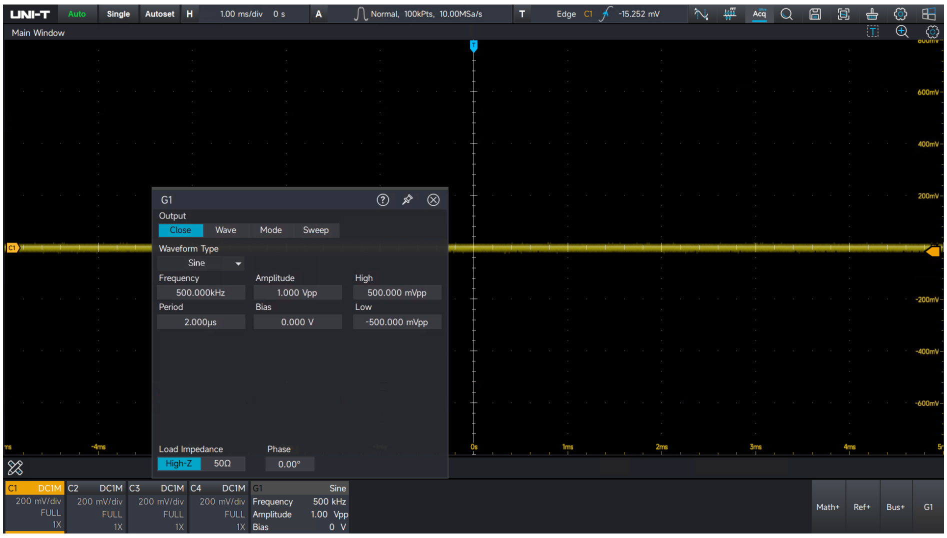
18. Function/Arbitrary Waveform Generator (Option)

UPO7000L series has a built-in function/arbitrary waveform generator, with a maximum output frequency of up to 60MHz, using direct digital synthesis technology to produce accurate and stable waveform output.

18.1 Turn on/off Function/Arbitrary Waveform Generator (AWG)

ON: Click the icon "G1" (AWG) G1 at the botton right to turn on the AWG channel and enter the signal outpit setting menu.

OFF: Use the mouse to click on the Gen channel at the bottom to turn off the Gen.



18.2 Output Continuous Wave Signal

(1) Turn on continuous wave signal

In the signal output setting menu, click to switch to continuous wave mode.

(2) Set waveform parameters

The continuous wave can be set to sine wave, square wave, pulse wave, slope wave, noise, DC, Sinc, exponential rising, exponential falling, Lorentz, haversine, Gaussian, ECG (electrocardiograph), and arbitrary wave. For the arbitrary wave, the user can select an arbitrary wave file to save the waveform output.

The following table shows all wave types and their parameter.

Sine wave	Frequency, cycle, amplitude, high-low level, offset
Square wave	Frequency, cycle, amplitude, high-low level, offset
Pulse wave	Frequency, cycle, amplitude, high-low level, offset, duty ratio, rising time, falling time
Slope wave	Frequency, cycle, amplitude, high-low level, offset, symmetry
Noise	Amplitude, offset, high-low level
DC	Offset
Sinc	Frequency, cycle, amplitude, high-low level, offset
Exponential rising	Frequency, cycle, amplitude, high-low level, offset
Exponential falling	Frequency, cycle, amplitude, high-low level, offset
Lorentz	Frequency, cycle, amplitude, high-low level, offset
Haversine	Frequency, cycle, amplitude, high-low level, offset
Gaussian	Frequency, cycle, amplitude, high-low level, offset
ECG	Frequency, cycle, amplitude, high-low level, offset
Arbitrary wave	Frequency, cycle, amplitude, high-low level, offset, arbitrary wave selection, output point by point

■ Output frequency

In the signal setting menu, click the "Frequency" text field to open the numeric keypad or use the external physical keyboard to configure the frequency and unit. Different waveforms can be configured with different frequency ranges. For specific frequency ranges, please refer to *UPO7000L Series Digital Phosphor Oscilloscopes-Data Sheet*.

■ Cycle

In the signal setting menu, click the "Cycle" text field to open the numeric keypad or use the external physical keyboard to configure the cycle and unit. For specific frequency ranges, please refer to *UPO7000L Series Digital Phosphor Oscilloscopes-Data Sheet*.

Note: The relationship between frequency and cycle is reciprocal. Changing the frequency automatically updates the cycle, and changing the cycle automatically updates the frequency.

■ Amplitude

In the signal setting menu, click the "Amplitude" text field to open the numeric keypad or use the external physical keyboard to configure the amplitude and unit.

Amplitude range：20mV to 6V

- High/low level

  In the signal setting menu, click the "High/Low" text field to open the numeric keypad or use the external physical keyboard to configure the high/low level and unit.

Note: The range of the high-low level can be set between -3V and 3V.

Amplitude = |high level| + |low level|. When adjusting the high or low level, the amplitude will change accordingly.

- Offset

  In the signal setting menu, click the "Offset" text field to open the numeric keypad or use the external physical keyboard to configure the offset and unit.

  Offset range：-3V to 3V

- Duty ratio for pulse wave

  In the signal setting menu, click the "Duty Ratio" text field to open the numeric keypad or use the external physical keyboard to configure the duty ratio.

  Duty ratio range: 0.01% to 99.99%

Note: The duty ratio is only available for pulse waves.

- Rising/falling time for pulse wave

  In the signal setting menu, click the "Rising/Falling Time" text field to open the numeric keypad or use the external physical keyboard to configure the rising/falling time.

  Rising/falling time range: 5ns to 2s.

Note: Rising/falling time is only available for pulse waves.

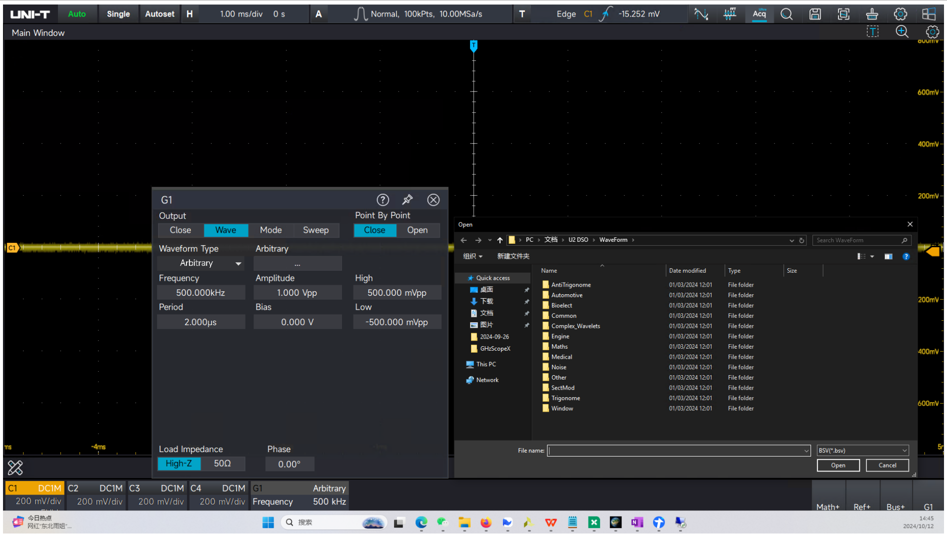
- Symmetry for slope wave

  In the signal setting menu, click the "Symmetry" text field to open the numeric keypad or use the external physical keyboard to configure the symmetry.

  Symmetry range: 0.01% to 99.99%

- Arbitrary wave

  In the signal setting menu, click the "Arbitrary Wave" text field to select an arbitrary file from the saved arbitrary file folder.



After loading the arbitrary wave, the output frequency and amplitude of the arbitrary wave can be set.

Output point-by-point:

In this mode, the signal generator automatically calculates the frequency of the output signal based on the waveform length and sampling rate. The signal generator outputs the waveform point by point at the calculated frequency. This mode prevents the loss of important waveform points. The default setting is "NO."

When the mode is set to "NO," the software automatically interpolates or extracts points to generate an arbitrary waveform with a fixed length and frequency as specified in the parameter list.

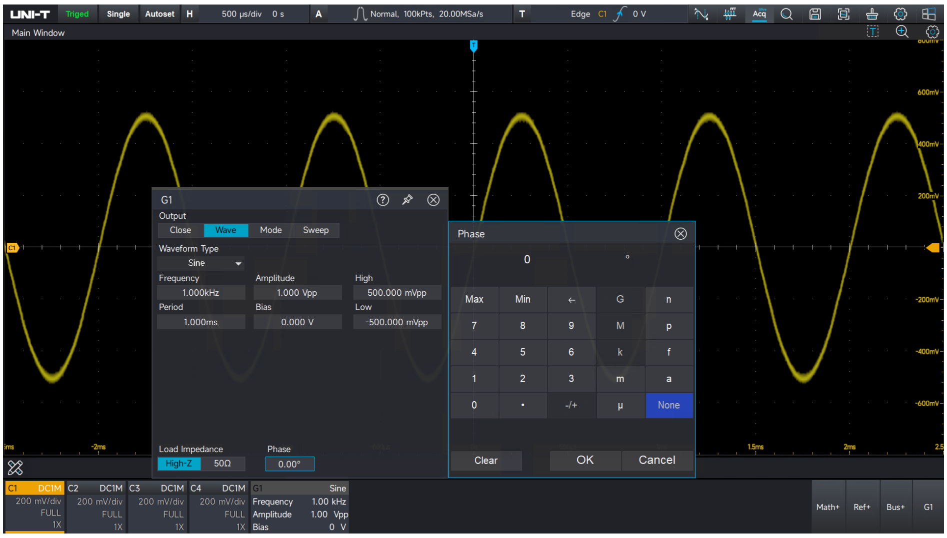
■ Impedance

The load impedance can be switched between "High-Z" and "50Ω." In the "High-Z" state, the output amplitude of the signal generator is twice that of the "50Ω" state. To avoid amplitude measurement errors caused by impedance mismatch between the signal generator and other test devices, please refer to the following formula to adjust the impedance.

$$V_{BNC} = \boldsymbol{VHighz} * \frac{\boldsymbol{R}\,(\text{external})}{50\Omega + \boldsymbol{R}\,(\text{external})}$$

■ Start phase

In the signal setting menu, click the "Phase" text field to open the numeric keypad or use the external physical keyboard to configure the start phase.



18.3 Output Modulation Signal

- Output AM signal

In AM mode, the modulated waveform consists of a carrier wave and a modulation wave. The amplitude of the carrier wave varies with the amplitude of the modulation wave. The modulation modes of the two channels are independent of each other.

This section uses a carrier wave (a sine wave of 10kHz, 1Vpp) and a modulation wave (a sine wave with a modulation frequency of 1kHz and a modulation depth of 100%) as an example.

(1) In the signal setting menu, click to switch to modulation wave.

(2) Select wave type

The modulating source comes from an internal source. The carrier wave can be set to a sine wave, square wave, pulse wave, slope wave, or arbitrary wave. For waveform parameter settings, please refer to the section "Output Continous Wave Signal". The waveform is set to a sine wave with a frequency of 10kHz and an amplitude of 1V.

(3) Select the modulation type to AM.

(4) Select modulation wave

The modulation wave can be set to a sine wave, square wave, pulse wave, slope wave, or arbitrary wave. The waveform is set to a sine wave.

(5) Set modulation frequency

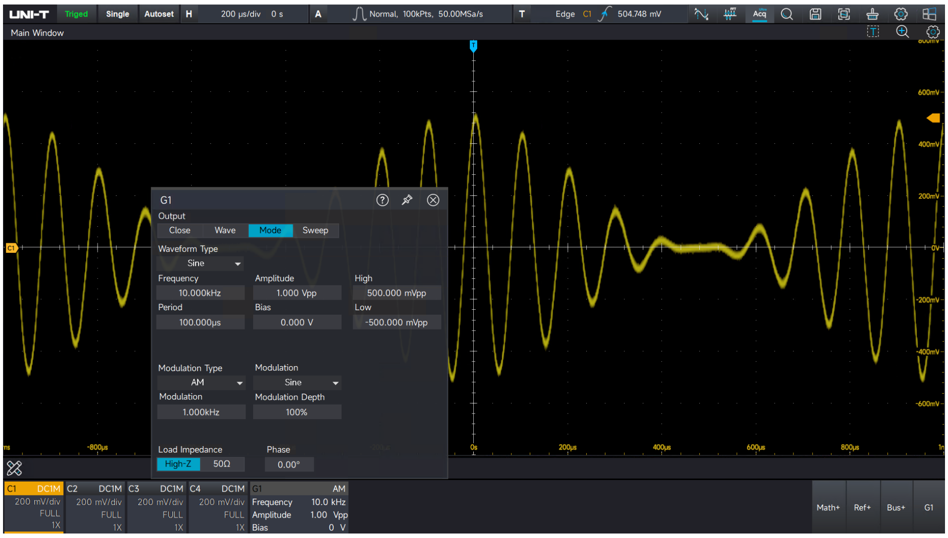
Modulation frequency range: 2mHz to 200kHz, set it to 1kHz.

(6) Set modulation depth

Modulation depth range: 0% to 120%, set it to 100%.

The modulation depth indicates the variation in amplitude and is expressed as a percentage.

The AM modulation depth range is 0% to 120%, with a default value of 100%. When the modulation depth is set to 0%, the output is a constant amplitude (half of the set carrier amplitude). When the modulation depth is set to 100%, the output amplitude changes according to the modulation wave. If the modulation depth exceeds 100%, the output amplitude of the oscilloscope will not exceed 3V (with a 50Ω load).



■ Output FM signal

In FM mode, the modulated waveform consists of a carrier wave and a modulation wave. The frequency of the carrier wave varies with the amplitude of the modulation wave.

This section uses a carrier wave (a sine wave of 10kHz, 100mV) and a modulation wave (a square wave with a modulation frequency of 2kHz and frequency offset of 5kHz) as an example.

(1) In the signal setting menu, click to switch to modulation wave.

(2) Select wave type

The carrier wave can be set to a sine wave, square wave, pulse wave, slope wave, or arbitrary wave. For waveform parameter settings, please refer to the section "Output Continous Wave Signal". The waveform is set to a sine wave with a frequency of 10kHz and an amplitude of 1V.

(3) Select the modulation type to FM.

(4) Select the modulation wave

The modulation wave can be set to a sine wave, square wave, pulse wave, slope wave, or arbitrary wave. The waveform is set to a sine wave.

(5) Set modulation frequency

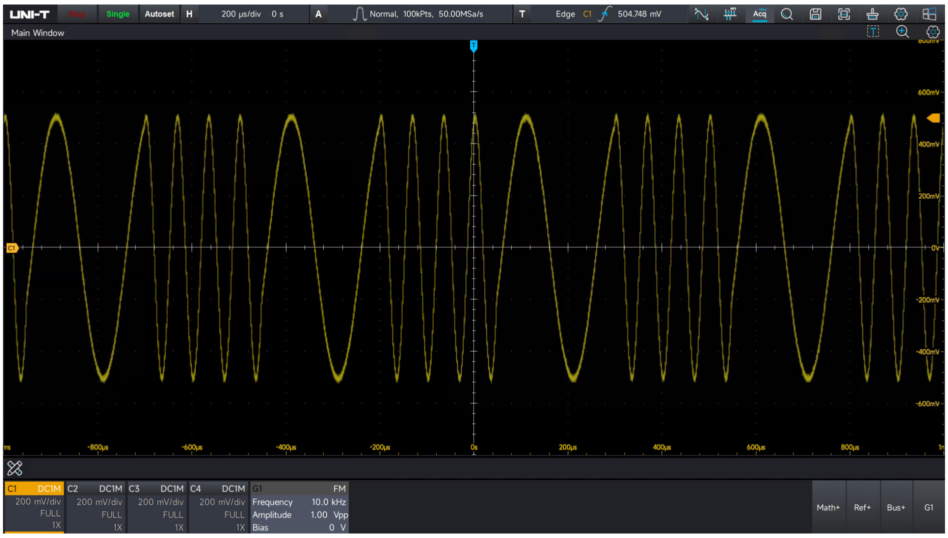
Modulation frequency range: 2mHz to 200kHz, set it to 2kHz.

(6) Set frequency offset

The frequency offset indicates the deviation of the frequency of the FM-modulated waveform from the carrier frequency.

FM frequency offset range: 0 ≤carrier frequency±modulation frequency ≤ system bandwidth.

The frequency offset is set to 5kHz.



■ Output PM signal

In PM mode, the modulated waveform consists of a carrier wave and a modulation wave. The phase of the carrier wave varies with the amplitude of the modulation wave.

This section uses a carrier wave (a sine wave of 500kHz, 1Vpp) and a modulation wave (a square wave with a modulation frequency of 50 kHz and phase offset of 180°) as an example.

(1) In the signal setting menu, click to switch to modulation wave.

(2) Select wave type

The carrier wave can be set to a sine wave, square wave, pulse wave, slope wave, or arbitrary wave. For waveform parameter settings, please refer to the section "Output Continous Wave Signal". The waveform is set to a sine wave with a frequency of 500kHz and an amplitude of 1V.

(3) Select the modulation type to PM.

(4) Select the modulation wave

The modulation wave can be set to a sine wave, square wave, pulse wave, slope wave, or arbitrary wave. The waveform is set to a sine wave.

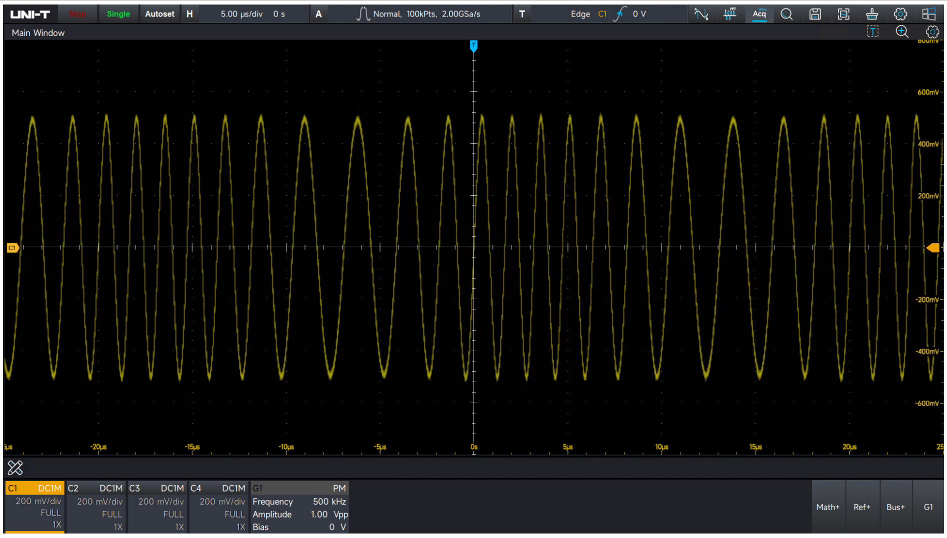
(5) Set modulation frequency

Modulation frequency range: 2mHz to 200kHz, set it to 50kHz.

(6) Set phase offset

The phase offset indicates the deviation of the phase of the PM-modulated waveform from the carrier frequency.

PM phase offset range: 0° to 360°, with the default setting at 180°.



18.4 Output Sweep Frequency Signal

In sweep frequency mode, the instrument changes the output frequency within the specified sweep time, with the frequency sweeping from the start frequency to the stop frequency using either a linear or logarithmic method.

The sine wave, square wave, slope wave, and arbitrary wave can generate a sweep frequency (except for DC).

This section uses a square wave with an amplitude of 1Vpp and a duty ratio of 50% as the sweep-frequency wave (with linear sweep, start frequency of 1kHz, stop frequency of 50kHz, and sweep time of 2ms) as an example.

(1) In the signal setting menu, click to switch to sweep frequency.

(2) Select wave type

The modulation wave can be set to a sine wave, square wave, pulse wave, slope wave, or arbitrary wave. The waveform is set to a sine wave. For waveform parameter settings, please refer to the section "Output Continous Wave Signal". The waveform is set to a square wave with an amplitude of 1V and a duty ratio of 50%.

(3) Select sweep frequency type

Linear: Changing the output frequency using the linear method, which is expressed as “Hz/second.”

Logarithmic: Changing the output frequency using the logarithmic method, which is expressed as “octave/second” or “10 times/second.”

(4) Set sweep frequency time

Use the numeric keypad or an external numerical keyboard to set the sweep frequency time, with a range of 1ms to 500s. The sweep frequency time is set to 2ms.

(5) Set trigger source

Internal: The oscilloscope's built-in clock is used as the trigger source to start the sweep frequency.

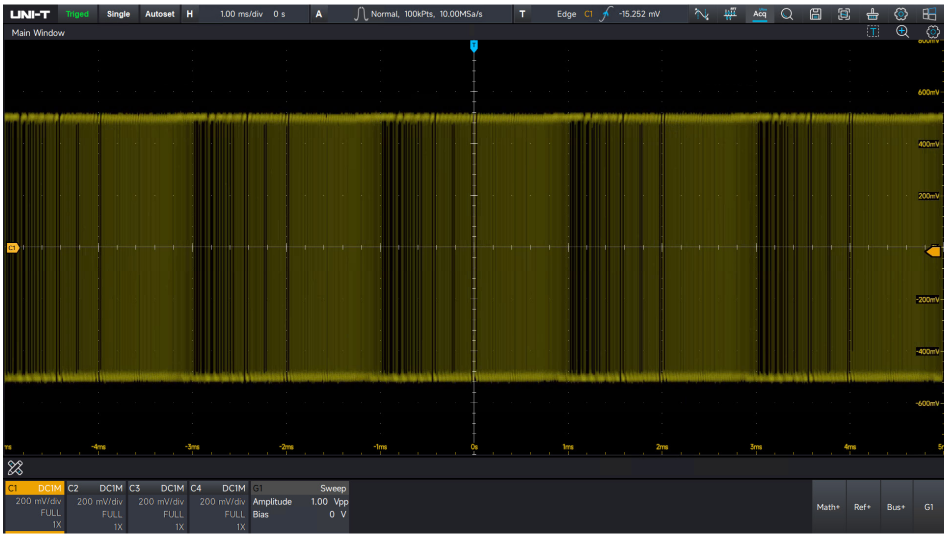
(6) Set start frequency and stop frequency

The start frequency and stop frequency define the upper and lower limits of the sweep frequency. The arbitrary waveform generator scans the output from the specified start frequency to the stop frequency and then back to the start frequency. The start frequency is set to 1kHz, and the stop frequency is set to 50kHz.

■ Start frequency < Stop frequency, the arbitrary waveform generator sweeps from the low frequency to the high frequency.

■ Start frequency > Stop frequency, the arbitrary waveform generator sweeps from the high frequency to the low frequency.

■ Start frequency = Stop frequency, the arbitrary waveform generator outputs a fixed frequency.



19. Search and Navigation

The search function helps users quickly find the events of interest and mark them. Then, with the help of event navigation, users can quickly locate the marked signals. Search conditions for the waveform search include edge and pulse width.

The navigation function allows users to quickly view and locate waveforms. To enter the search function, click the icon in the top right corner of the screen.

Search type: Edge type and pulse width.

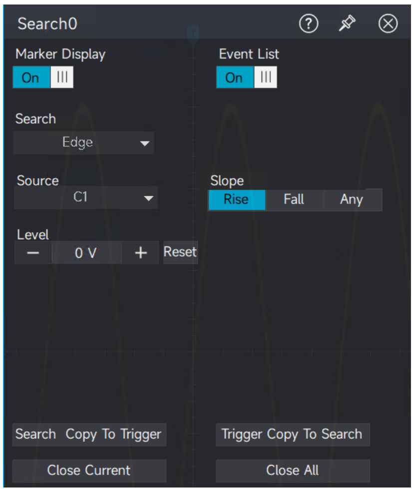
The edge search is used as an example to introduce the search function.

Search source: C1-C4.

Search edge: Rising edge, falling edge, or arbitrary edge

Search level:

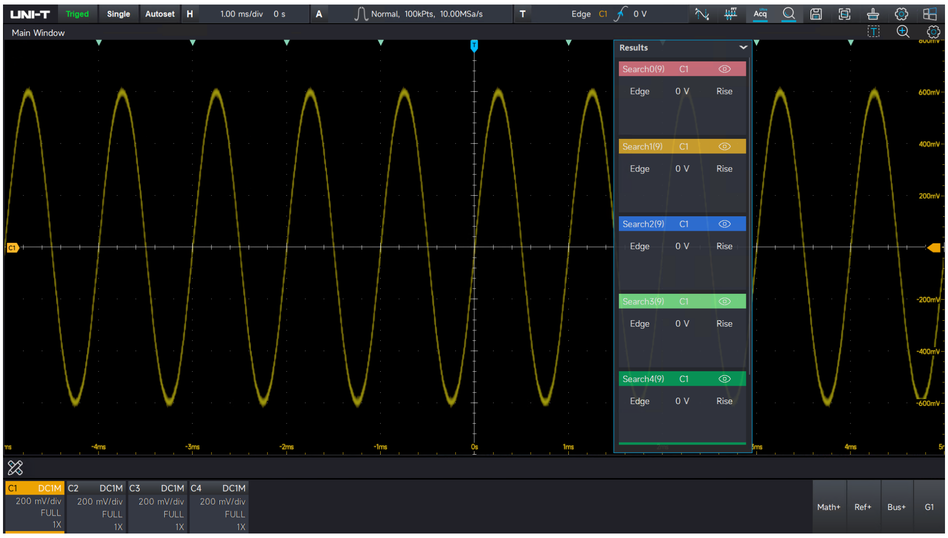
Use the numeric keypad or click the − and + icons to adjust the level value.



Copy search to trigger: Copy the search condition to the trigger and use it for triggering.

Copy trigger to search: Copy the trigger condition to the search and use it for searching.

Add various search items: Click the search icon in the top right corner of the screen to add different search items. UPO7000L supports up to 10 items.

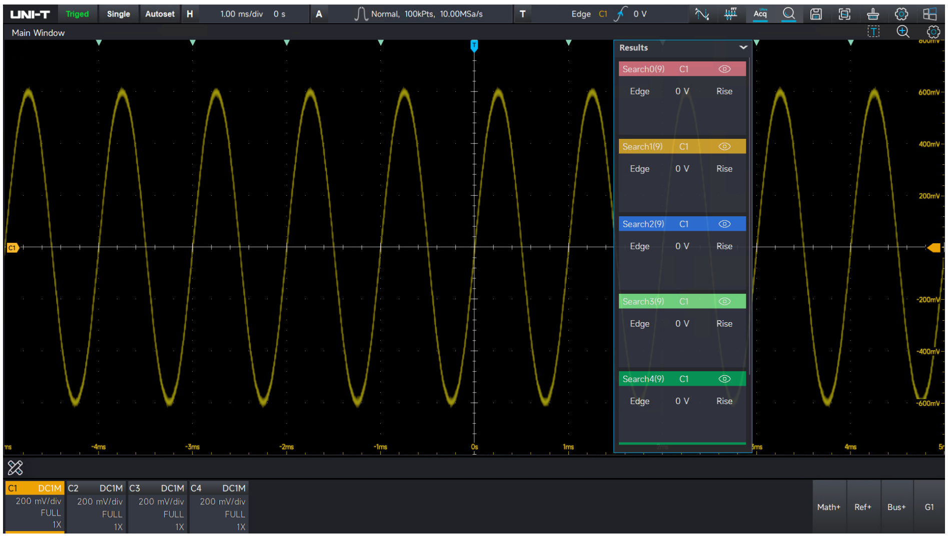


Turn off search:

Click the search label to disable the current search item, or click "Off" in the search menu to disable the current search item or all search items.

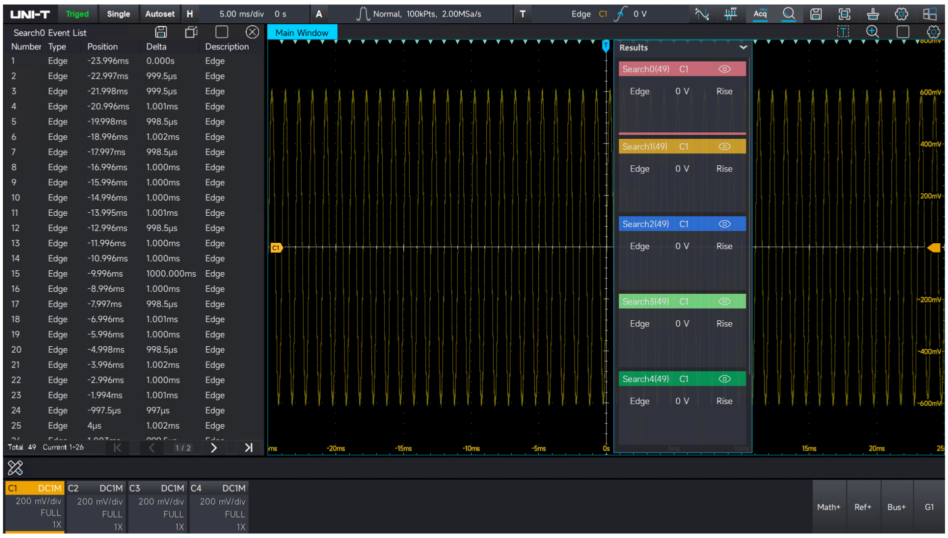
Search Marker

Different search items are marked with inverted triangle icons in various colors. Click the eye icon in the top right corner of a search item to toggle the visibility of its marker.



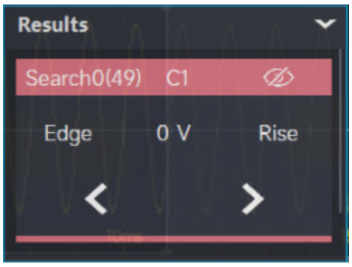
Search Event Table

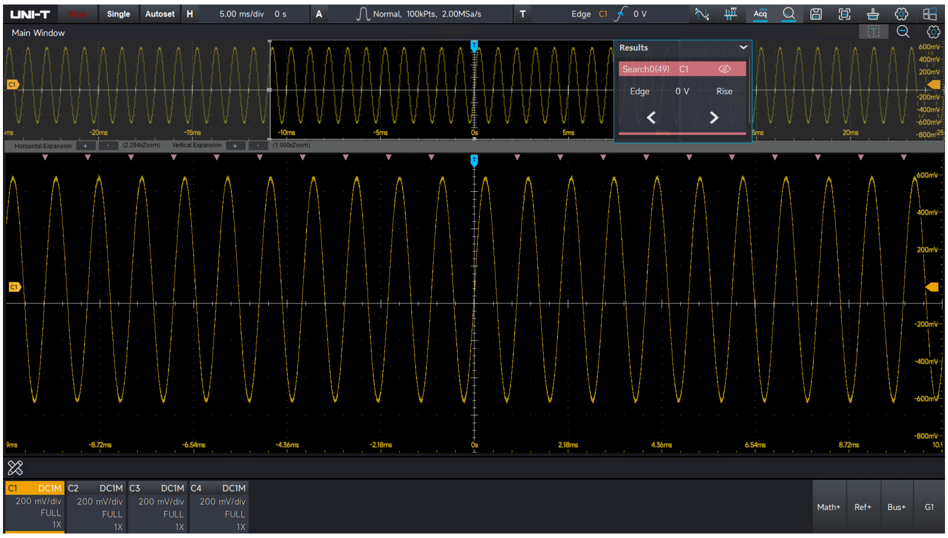
The search event table lists all waveform details currently displayed on the screen. Additionally, the number of searches is shown in the top left corner of the search item. The event list includes the event number, search type, location (with timestamp), interpolation, and description. Clicking on different search items will switch to the corresponding event list.



Sequency Navigation

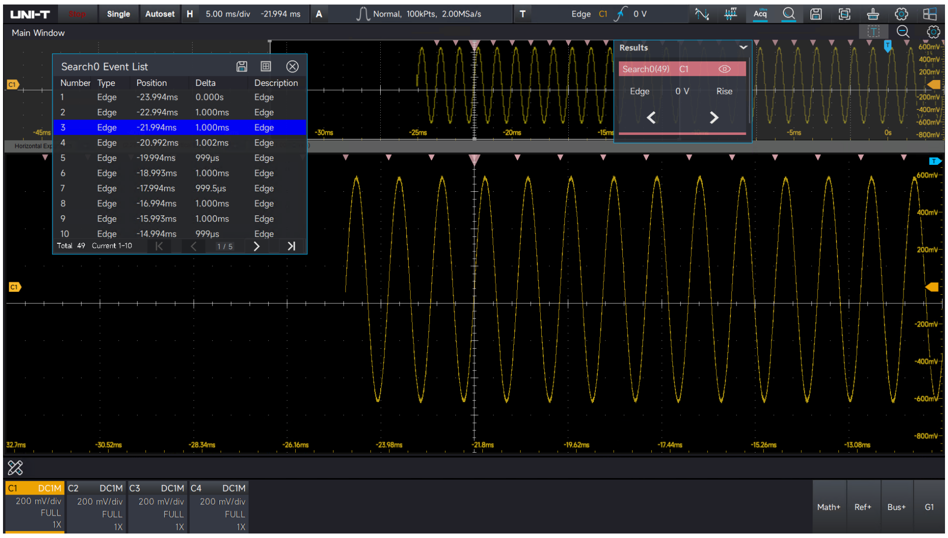
In the stop state, the user can navigate through the searched waveforms. Click the "Previous" and "Next" icons at the bottom of the search item for sequential navigation. During navigation, the oscilloscope will automatically expand the window to display the waveforms in maximized view.





Event Navigation

In the stop state, the user can click on an event in the search event list to navigate to the searched waveforms. During navigation, the oscilloscope will automatically expand the window to display the waveforms in maximized view.



20. Window Display

- Marker Display
- Persistence
- Grid Type
- Waveform Type
- Brightness Control

The marker display, persistence, grid type, waveform type, and brightness of the waveform can be configured in the main window.

Note: In other independent windows, persistence cannot be set, and UltraAcq® mode cannot be enabled.

20.1 Marker Display

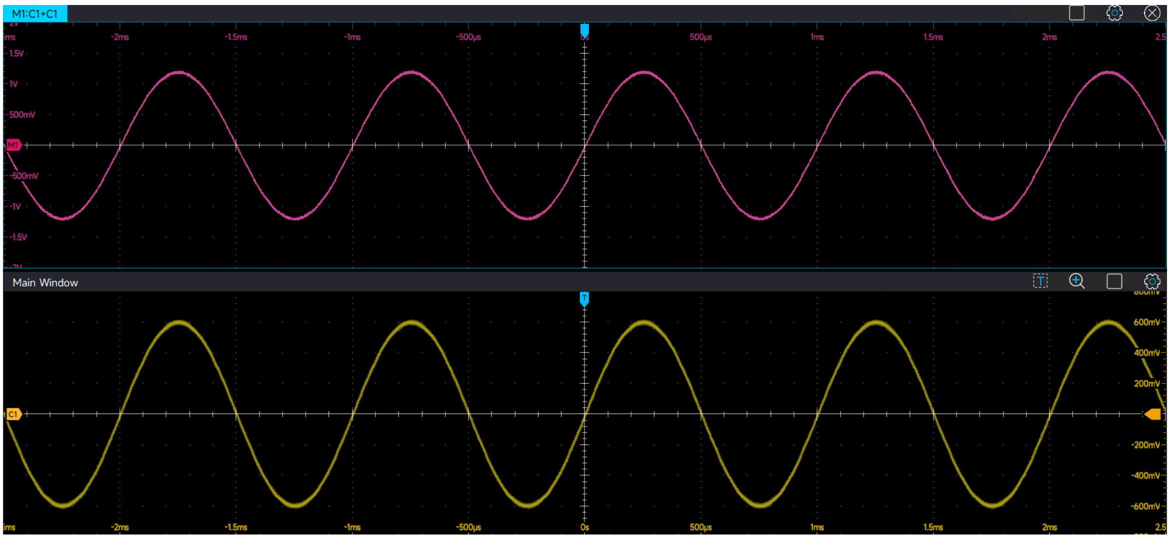
(1) Enable marker display

In the window display menu, enable the marker display and toggle the vertical marker display on/off.

(2) Adjust marker position

Vertical marker position: Displayed on the left or right side.

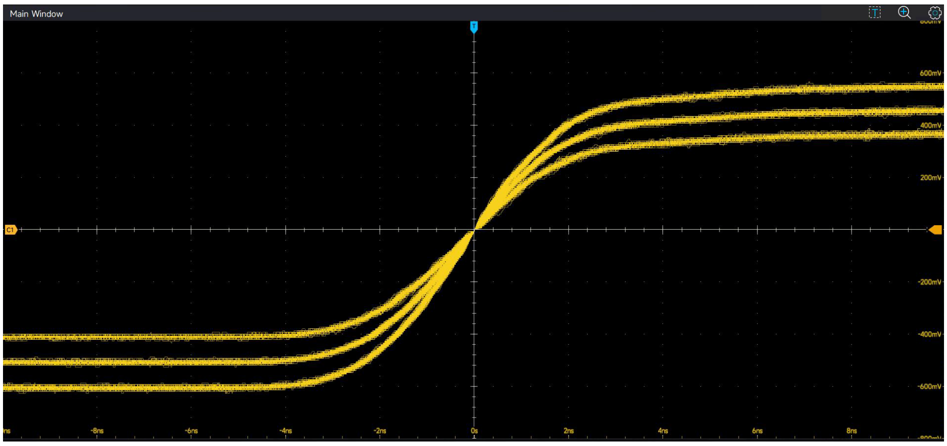
Horizontal marker position: Displayed on the top or bottom.



20.2 Persistence

Persistence display visualizes waveform changes at a high refresh rate, facilitating the preliminary analysis of abnormal waveform behavior. UPO7000L series offers two persistence modes: automatic and Infinite.

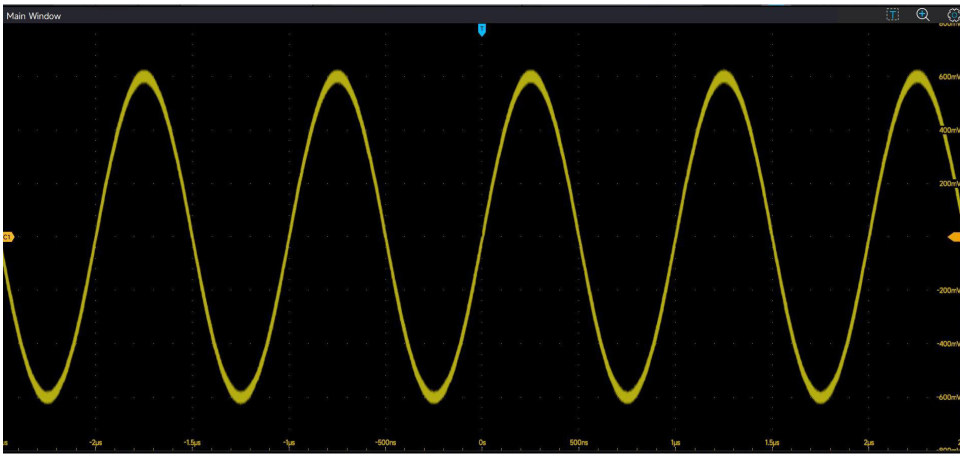
- Auto: The oscilloscope automatically adjusts the waveform persistence.
- Infinite: When a new waveform is collected, the oscilloscope does not clear the previous waveform. The new waveform will have higher brightness, while the older waveform will have slightly lower brightness. Infinite persistence is useful for quickly analyzing noise and jitter ranges and effectively capturing probability events in high refresh scenarios.



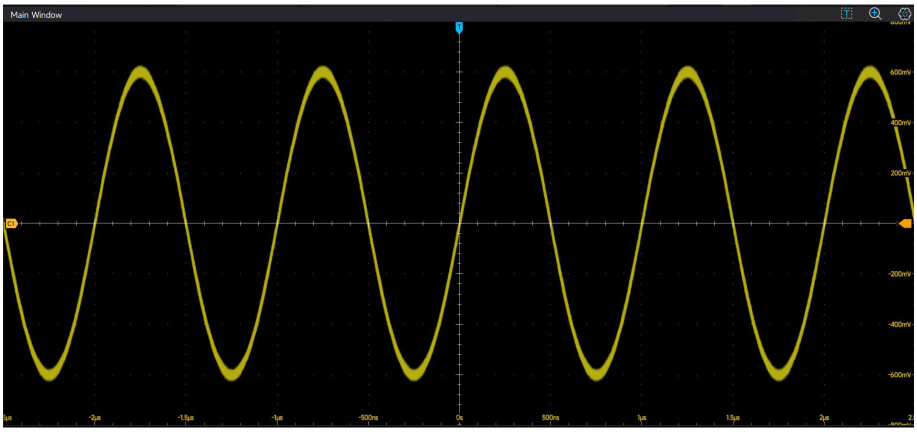
20.3 Grid Type

Waveform grid type: Simple, full, or none.

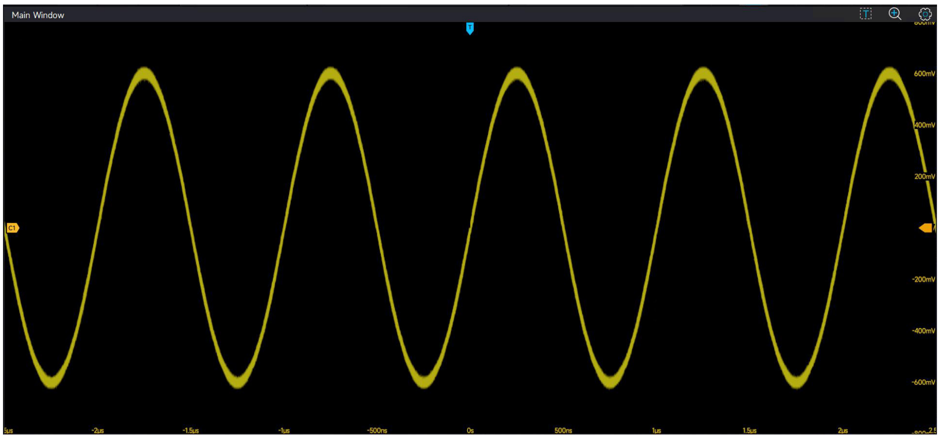
Simple type:



Full type:



none:

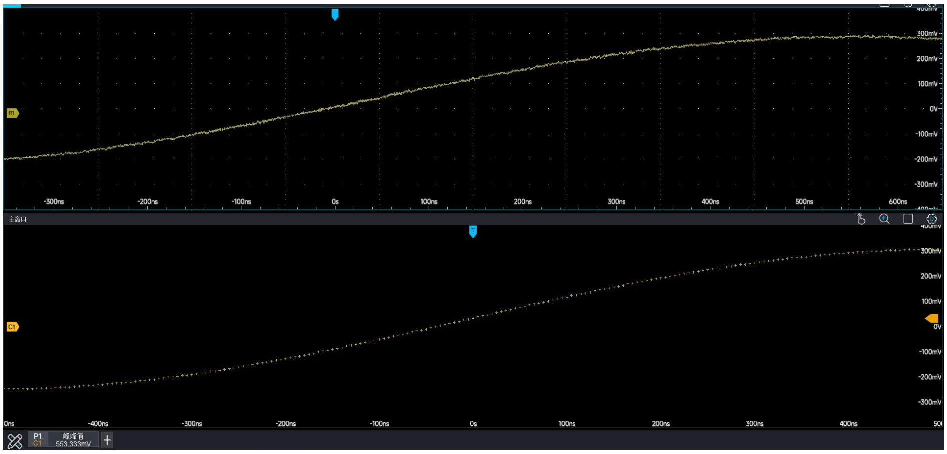


20.4 Waveform Type

In the window display menu, the user can select the waveform type as either vector or dot.

Vector: This mode typically provides the most realistic waveforms, making it easier to observe sharp edges, such as sharp edges in a square wave.

Dot: This mode displays the sampling points directly.



20.5 Brightness Control

Adjust waveform brightness:

The waveform brightness can be adjusted by clicking and dragging the brightness bar left or right with the mouse. The progress bar displays the current value, ranging from 1 to 100.

Adjust grid brightness:

The grid brightness can be adjusted by clicking and dragging the brightness bar left or right with the mouse. The progress bar displays the current value, ranging from 1 to 100.

21. Save and Load

- Save and Load Waveform
- Save Screenshot
- Save and Read Setting
- External Storage and Load

The storage function allows the user to save the oscilloscope's waveform and screen image to internal memory or an external USB flash drive. UPO7000L provides four USB Host ports for connecting external storage devices. Users can reload saved settings and waveforms as needed by clicking the storage icon in the top right corner of the screen to access the storage menu.

21.1 Save and Load Waveform

Save Waveform:

(1) Storage formats

UPO7000L series supports 7 waveform formats: Binary (.bin), Text (.txt), Matlab (.mat), Excel (.xlsx), CSV (.csv), TSV (.tsv), and DAT (.dat).

The Text (.txt) format includes 5 text encoding options:II, GB2312, UTF8, UTF32, and Unicode.

(2) Save waveform

UPO7000L can save waveforms from four analog channels (C1–C4).

(3) Filename

Click the filename field to open the soft keyboard for entering the desired filename.

Suffix of date: The file will be saved with the current system date automatically appended to the filename, for example, Uni-t00120231010163554902.bin.

(4) Storage location

Click the icon to select the storage location. If a filename already exists, click "Save", a prompt will appear: "File already exists, do you want to overwrite it?" The user can choose to overwrite the file or enter a new filename.

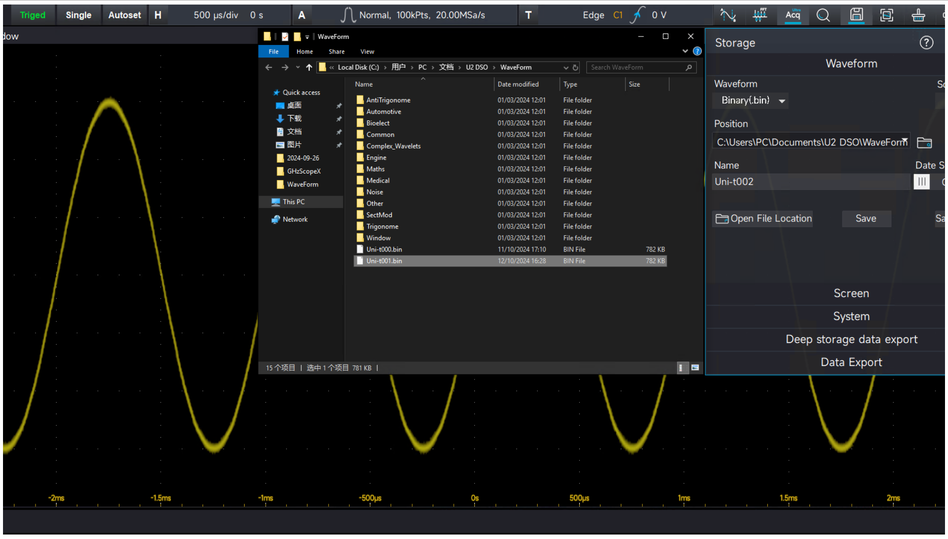
(5) File location

The oscilloscope will navigate to the directory where the image is stored.

Load waveform:

UPO7000L currently supports loading waveform in .bin and .csv formats.

Click "Save & Load" to simultaneously save the waveform and load it back.



21.2 Save Screenshot

In the save settings menu, click “Screenshot” to access the screen image storage page.

(1) Screenshot area

UPO7000L supports two screenshot area options: screen and grid.

(2) Image color

UPO7000L provides three image processing: standard, black and white, and inverse color.

Description of Save Screenshot

Function	Setting	Description
Color	Standard	The oscilloscope screenshot is stored using the same colors displayed on the interface.
	Inverse	The oscilloscope converts dark backgrounds to light-colored backgrounds when storing screenshots. This feature is designed to save ink during printing.
	Black and white	The oscilloscope can convert color screenshots into grayscale images for storage.
Area	Screen	The oscilloscope screenshot can be stored with the full screen information.
	Gird	The oscilloscope screenshot can be stored with only the grid information.

(3) Storage formats

UPO7000L series supports 5 waveform formats: .bmp, .tiff, .gif, .png, and .jpeg.

(4) Storage location

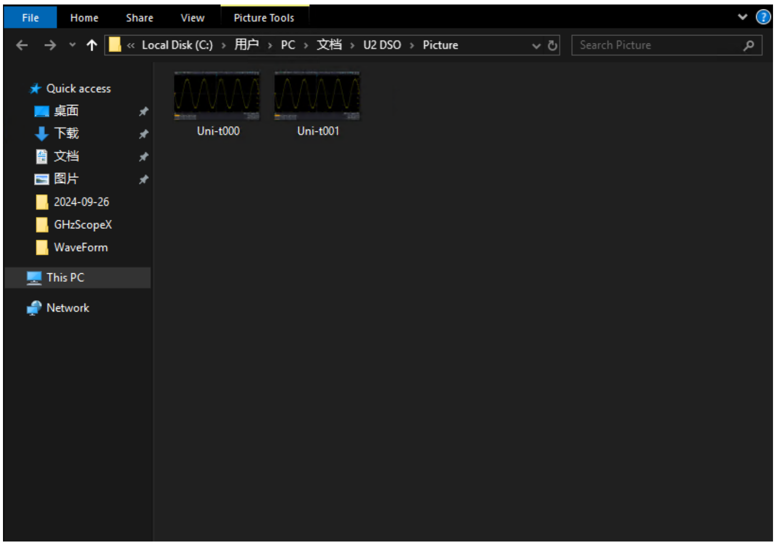
Click the icon to select the storage location. If a filename already exists, click "Save", a prompt will appear: "File already exists, do you want to overwrite it?" The user can choose to overwrite the file or enter a new filename.

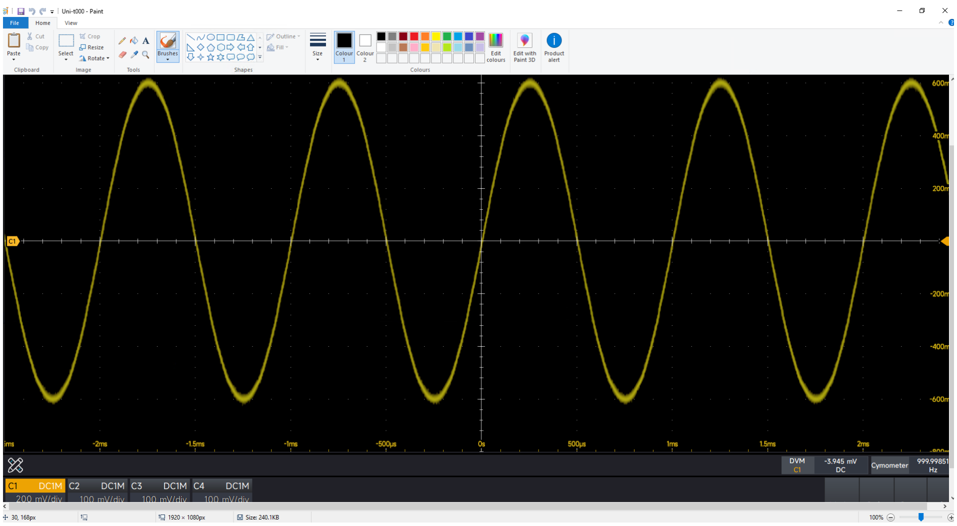
(5) Save & Jump

Click "Save & Jump" to save the image and automatically navigate to the image storage directory.

(6) File location

The oscilloscope will navigate to the directory where the image is stored.





Note: UPO7000L series supports a quick save feature. Click the icon, the oscilloscope will automatically save the image to the default save path.

21.3 Save and Read

In the save settings menu, clicking on “System” settings to open the system settings menu. The oscilloscope saves its setup file in the .set format. This allows users to easily recall the saved setup file in future sessions, making it convenient to quickly restore the oscilloscope to the previous configuration (e.g., for QC testing).

(1) Storage location

Click the icon to select the storage location. If a filename already exists, click "Save", a prompt will appear: "File already exists, do you want to overwrite it?" The user can choose to overwrite the file or enter a new filename.

(2) Save & Jump

Click "Save & Jump" to save the image and automatically navigate to the image storage directory.

(3) File location

The oscilloscope will navigate to the directory where the image is stored.

(4) Read

The oscilloscope loads the saved setup file to restore the state to its previous configuration.

Note: UPO7000L series can read the setup file from the local directory and can also save and read from an external storage device.

21.4 External Storage and Load

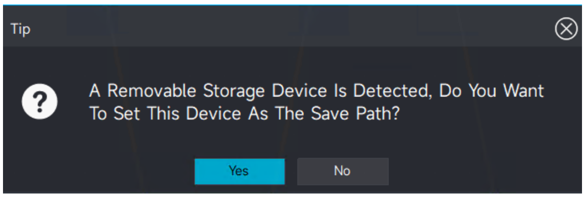
UPO7000L series supports saving waveforms, screenshots, and setup files to external storage devices via USB, and it also supports reloading waveforms and settings from USB.

(1) Recognizer removable storage device (using USB as an example)

When a USB drive is inserted, the oscilloscope will detect it and display a prompt: "A removable storage device is detected, do you want to set this device as the saved path?"

If "No" is selected, the oscilloscope will use the local storage as the default save path.

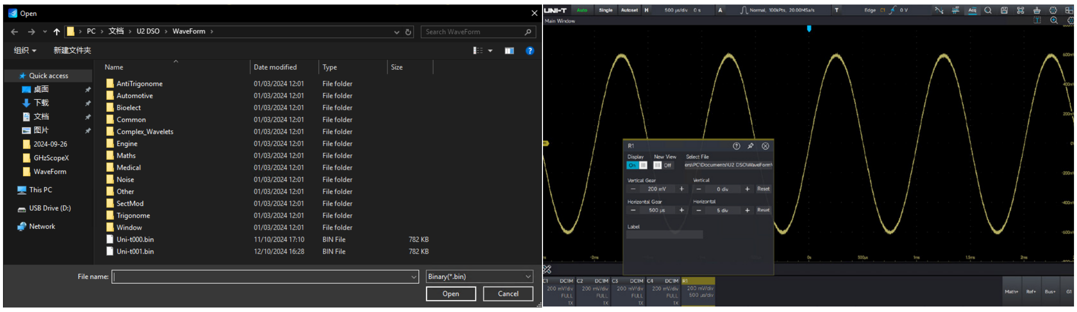
If "Yes" is selected, the oscilloscope will save the waveform, screenshot, and setup file to the USB file directory.



Note: The oscilloscope cannot create files directly on its operating page. To save waveforms, screenshots, and setup files to an external storage device, please create and name a folder on the removable storage device in advance.

(2) Loading reference waveform from external storage

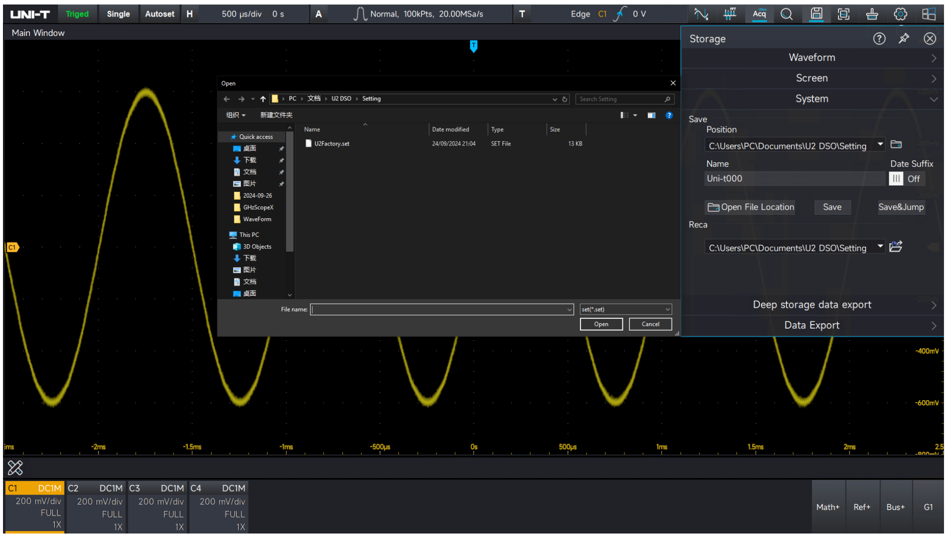
Click "Ref +" at the bottom right corner to open the file selection window. Select the saved .bin file and click "Confirm" to reload the reference waveform.



Note: The default local storage disk for the oscilloscope is the C drive. UPO7000L supports up to four removable storage devices.

(3) Loading settings from external storage

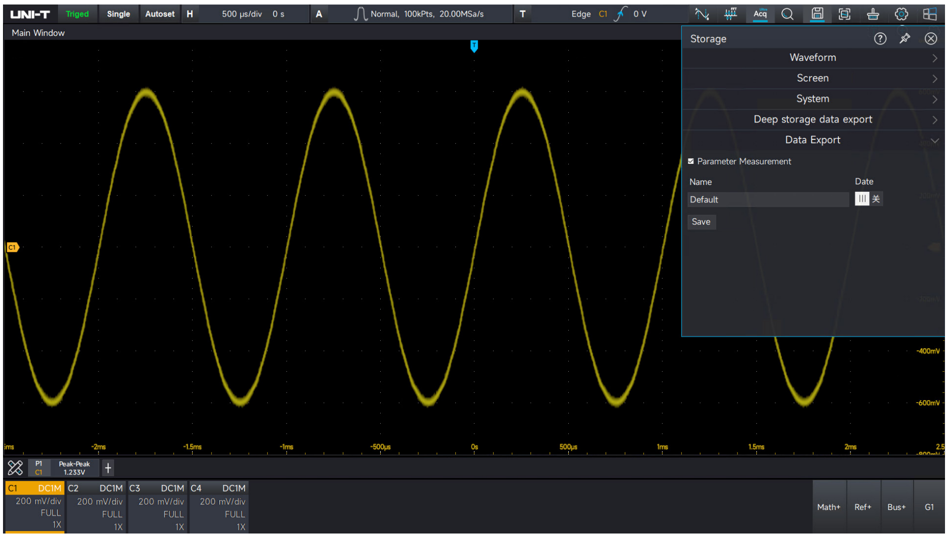
In the save settings menu, click System Setting>> Reading File Storage Path, select the saved .set file, and click “Confirm” to load the saved settings.



21.5 Export Data

UPO7000L supports exporting statistical data of measurement parameters, including the current value, maximum value, minimum value, average value, standard deviation, and data count. The data can be saved in two formats: CSV and TXT.

Additionally, the measurement data displayed in a list, such as power analysis parameters or jitter analysis parameters, can also be exported. To export the data, simply click the "Export Data" button on the interface.



22. System Setting

- Display Setting
- Automatic and Calibration Settings
- Communication
- Auxiliary Output
- Function Module
- Others

Access the system setting:

Click the icon in the top right corner or enter the start menu to click the system icon Settings to enter the system menu.

22.1 Display Setting

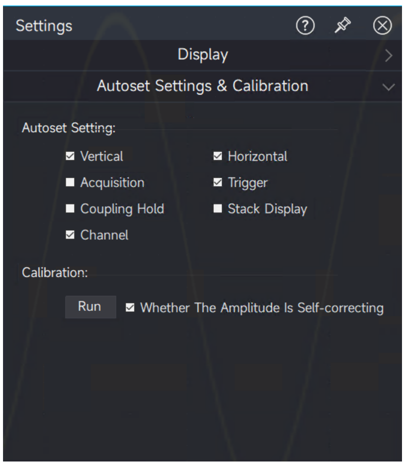
The screen contrast ratio can be adjusted by dragging the screen brightness slider. The adjustable range is from 50% to 100%, with the default brightness set at 70%.

22.2 Automatic and Calibration Settings

Automatic setting

UPO7000L features a fast Autoset mode, which automatically adjusts the vertical settings, horizontal settings, acquisition settings, trigger settings, coupling hold, channel settings, and stacked display based on the input signals. This ensures that the waveforms are displayed stably on the screen.

Users can check (☑) or uncheck (☐) these settings to customize which parameters the oscilloscope will adjust during the fast Autoset process.



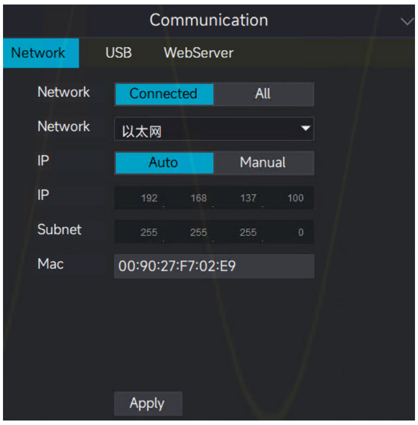
Automatic calibration

The calibration function ensures that the oscilloscope operates in optimal conditions for the most accurate measurements. It is recommended to perform this function whenever the ambient temperature changes by 5°C or more. Before performing auto-calibration, make sure the oscilloscope has been powered on and running for at least 20 minutes to ensure proper stabilization.

22.3 Communication

(1) Network

Before using the LAN bus, connect the oscilloscope to the network via a network cable. The oscilloscope's network port is located on the rear panel. Users can check the current network settings and configure network parameters in the system settings page.



Network selection

- IP address

  The format of the IP address is nnn.nnn.nnn.nnn. The first "nnn" can be set from 0 to 255 (except 127), with a valid range of 0 to 223, and the remaining three "nnn" values can range from 0 to 255.

- Subnet mask

  The subnet mask format is nnn.nnn.nnn.nnn, where each "nnn" value ranges from 0 to 255.

- Gateway

  In static IP mode, the gateway can be set. The format of the gateway is nnn.nnn.nnn.nnn, where the first "nnn" can be set from 0 to 223, and the remaining three "nnn" values can range from 0 to 255.

- DNS (Domain Name System)

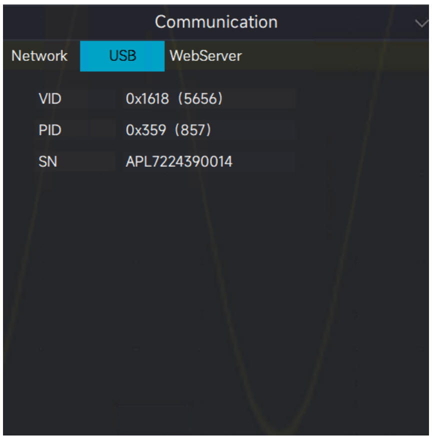
  In static IP mode, the DNS can be set. The format of the DNS is nnn.nnn.nnn.nnn. The first "nnn" can be set from 0 to 223, and the remaining three "nnn" values range from 0 to 255. Typically, users do not need to set DNS for the network.

- MAC address

  Each oscilloscope has a unique MAC address. When assigning an IP address to the instrument, the device is always identified by its MAC address. After configuring the network information, click "IP" to change the IP address, or click "DHCP" to automatically obtain an IP address.

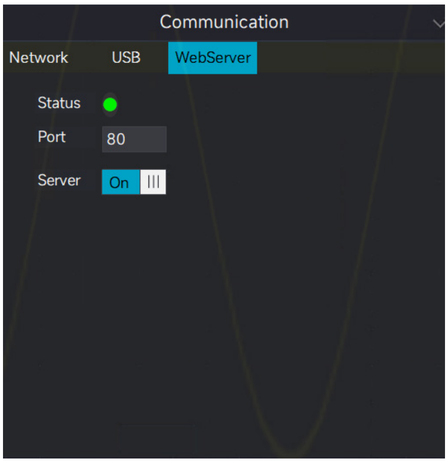
(2) USB

The oscilloscope can display the manufacturer's ID, product ID, serial number, and the currently used VISA address. It can connect directly to a PC via the USB DEVICE port on the rear panel for communication, without the need for additional configuration parameters.



(3) WebServer

The WebServer section displays the current network switch state. The default network port is 80.



For remote control via WebServer, please refer to the "WebServer Remote Control"section.

22.4 Auxiliary Output

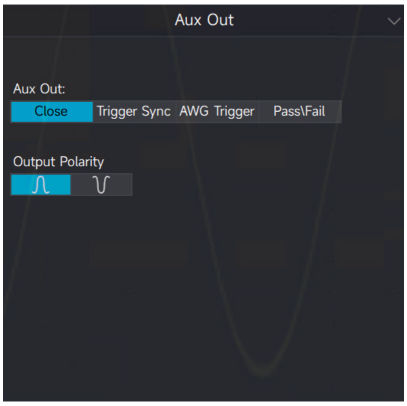
UPO7000L offers a wide variety of interfaces and is equipped with a signal auxiliary output function. These functions can be configured through the system settings interface.

Auxiliary Output Signal:

Triggering synchronization: The oscilloscope uses the period of the trigger signal as the pulse edge output, with the signal being sent through the Aux Out interface on the rear panel.

AWG trigger output: The sweep time of the function/arbitrary waveform generator in sweep mode is used as the pulse output, and the signal is sent through the Aux Out interface on the rear panel.

Pass/fail test: The pass/fail result is output as a pulse signal, which is also sent through the Aux Out interface on the rear panel. For more details, refers to the "Pass/Fail Test"section.



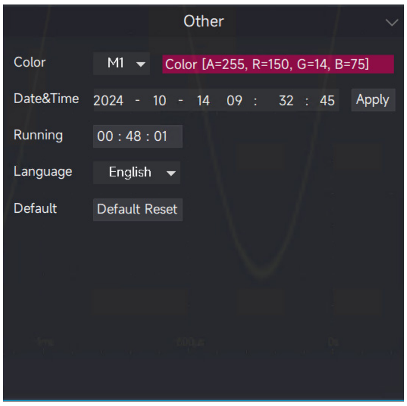
22.5 Function Module

UPO7000L can use the internal functions by selecting and enabling the functional components of the device itself. The optional components include:

- BUS: Bus decoding function
- Ref: Reference waveform
- Segement: Sequential mode
- Lissajous: Lissajous curve
- Jitter: Jitter analysis and eye diagram
- PowerAnalysis: Advanced power analysis
- Math: Mathematical operation
- P/F: Pass/fail test
- Search: Search and navigation
- Measure: Parameter measurement
- AWG: Function/arbitrary waveform generator

22.6 Others

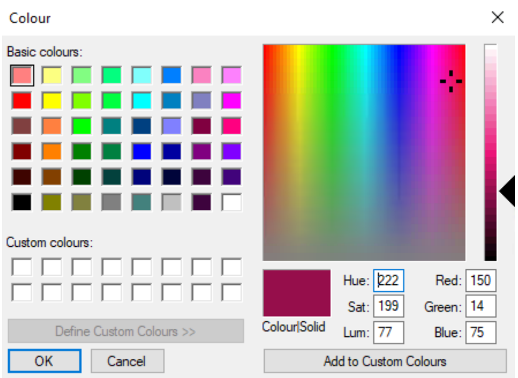
Other settings: modifying the channel's color, setting the time format, changing the language, restoring factory settings, and enabling touch lock.



Modifying the channel's color

(1) Select the channel whose color needs to be changed: M1–M8, R1-R4, B1, G1.

(2) Click "Color" on the right side to open the color selection window. Change the channel's color using the color panel. Users can select a standard color or a user-defined color.



Time format: Year: Month: Date: Hour: Minute: Second.

Language: UPO7000L supports simplified Chinese and English.

Default settings:

Click "Restore Factory Settings" to reset the instrument to its factory settings. The interface will prompt the user with "Do you want to restore factory settings?" Select "Yes" to confirm the restoration.

The factory settings for UPO7000L series oscilloscopes are shown in the following table.

Module	Function	Factory Setting
Time base	Horizontal scale	1μs/div

	Delay	0 (Horizontal midpoint)
Acquisition setting	Acquisition mode	Normal
	Average time	8
	Envelope time	8
	Envelope line	Top
	10MHz clock input	Internal
	Memory depth	Auto
	Interpolation	Sine wave
	Fast acquisition	OFF
	Sequence mode	OFF
	Sequence mode - display mode	Single frame
	Sequence mode - start frame	1
	Sequence mode - reference frame	1
	Sequence mode - current frame	1
	Sequence mode - display type	None
Trigger setting	Trigger type	Edge
	Trigger mode	Auto
	Source	C1
	Level	0
	Coupling	DC
	Edge	Rising edge

	Edge - EXT - input impedance	1MΩ
	Pulse width - source	C1
	Pulse width - level	0
	Pulse width - polarity	Positive
	Pulse width - condition	>
	Pulse width – low limit	3.2ns
	Pulse width – upper limit	3.6ns
	Video - source	C1
	Video - source	PAL
	Video - polarity	Positive
	Video - synchronization	Even field
	Video – line number	1
	Video - level	0
	Slope - source	C1
	Slope	Rising edge
	Slope - condition	>
	Slope – low level	0 V
	Slope - high level	0.1V
	Slope - lower time limit	3.2ns
	Slope - upper time limit	3.6ns
	Runt - source	C1
	Runt - polarity	Positive

	Runt - condition	>
	Runt - low level	0V
	Runt - high level	0.1V
	Runt - lower time limit	3.2ns
	Delay - source 1 edge	Rising edge
	Delay - source 2 edge	Rising edge
	Delay - source 1	C1
	Delay - source 2	C2
	Delay - condition	>
	Delay - lower time limit	3.2ns
	Delay - upper time limit	3.6ns
	Delay - source 1 level	0V
	Delay - source 2 level	0V
	Timeout - source	C1
	Timeout – edge type	Rising edge
	Timeout - level	0V
	Timeout - duration	3.2ns
	Duration - condition	>
	Duration - lower time limit	3.2ns
	Duration - upper time limit	3.6ns
	Setup & Hold – clock source	C1
	Setup & Hold - data source	C2

	Setup & Hold – clock edge	Rising edge
	Setup & Hold - clock level	0V
	Setup & Hold - code pattern	H
	Setup & Hold - data level	0V
	Setup & Hold - condition	Setup
	Setup & Hold - setup time	3.2ns
	Setup & Hold - hold time	3.2ns
	Nth edge - source	C1
	Nth edge - edge type	Rising edge
	Nth edge - idle time	3.2ns
	Nth edge - edge count	1
	Nth edge - level	0V
	Trigger hold time	6.4ns
Cursor setting	Display	OFF
	Source	C1
	Data display	Suspension
	Synchronous movement	OFF
	Type	Horizontal
	Horizontal A position	-3.2div
	Horizontal B position	3.2div
	Vertical A position	1div
	Vertical B position	9div

	Horizontal unit	div
	Vertical unit	s
Print	Direction	Horizontal
	Area	Full
	Color	Standard
Save setting	Save - waveform - waveform format	bin
	Save – waveform - source	C1
	Save - waveform - position	C:\Users\Administrator\ Documents\U2 DSO\WaveForm
	Save - waveform - name	Uni-t000
	Save - waveform - date suffix	OFF
	Save - screenshot - area	Screen
	Save – screenshot - color	Standard
	Save - screenshot - type	.png
	Save - screenshot - position	C:\Users\Administrator\ Documents\U2 DSO\Picture
	Save - screenshot - name	Uni-t000
	Save - screenshot - date suffix	OFF
	Save - system setting - save - position	C:\Users\Administrator\ Documents\U2 DSO\Setting

	Save - system setting - save - name	Uni-t000
	Save - system setting - save - date suffix	OFF
	Save - system setting – read - position	C:\Users\Administrator\ Documents\U2 DSO\Setting
System setting	System setting - display - screen brightness	100
	System setting - display – screen contrast ratio	100
	System setting - automatic and calibration setting	Horizontal, vertical, acquisition, trigger, and channel settings
	System setting - auxiliary input	OFF
	System setting - auxiliary output - output polarity	Rising edge
Pass/fail test	Test	OFF
	Source	C1
	Type	Limit test
	Vertical tolerance	200m/div
	Horizontal tolerance	100m/div
	Reference source	C1
	Total number of violations	1

	Total waveform	100
	Total time	1s
	Test failure operation	Save, alarm
	Test template	ON
	Standard	ANSI T1.102
	Lock template	OFF
Jitter analysis	Jitter analysis	OFF
	Source	C1
	Signal type	Data signal
	Comparison threshold	50%
	Delay	30%
	Data mode length	127bits
	Baud rate	(Nan bps) ---
	Clcol recovery	Constant clock
	PLL mode	Type I
	Cut-off frequency	(Nan bps) ---
	Cut-off factor	1667
	Figure	Tendency chart
	Spectrum figure – judgment threshold	50%
	Number of histogram columns	Num250
	Natural frequency	(Nan bps) ---

	Damping factor	0.707
Digital voltmeter	Display	OFF
	Source	C1
	Auto range	OFF
	Mode	DC
Frequency meter	Display	OFF
Parameter snapshot	Display	OFF
	Source	C1
Parameter measurement	Display	OFF
	Measurement threshold	Screen
	Measurement statistics	OFF
Analog channel setting	Vertical - display	C2, C3, C4, OFF
	Vertical - reverse phase	OFF
	Vertical - coupling	DC1M
	Vertical - vertical scale	200mV/div
	Vertical - bandwidth limit	FULL
	Vertical - position	0 (Vertical midpoint)
	Vertical - unit	V
	Vertical – offset voltage	0V
	Vertical - label	“ “
	Input - probe unit	V

	Input – probe button	Headlight
	Input – probe attenuation factor	1X
	User-defined attenuation factor	1X
Ref setting	Display	OFF
Bus	Display	OFF
AWG	Function	OFF
	Output	OFF
Mathematical operation	Display	OFF
	Operation type	Basic operation
	Math - basic operation - source1	C1
	Math - basic operation - source2	C2
	Math - basic operation - operation	+
	Math - basic operation – vertical scale	Latest scale (default)
	Math - basic operation – vertical position	0div
	Math - basic operation - label	""
	Math - basic operation – user-defined unit	V
Main window setting	Marker	ON
	Persistence	OFF

	Vertical marker position	Right
	Horizontal marker position	Down
	Waveform type	Vector
	Gird type	Full
	Gird brightness	10
	Waveform brightness	65

23. Remote Control

- User-defined Programming
- PC Software Control
- Web Server

UPO7000L series oscilloscopes support a variety of remote control methods. This chapter provides detailed instructions on how to use NI-MAX software to remotely control the oscilloscope through various interfaces.

Note: Before connecting the communication cable, ensure the instrument is turned off to prevent damage to its communication interface.

23.1 User-defined Programming

The user can control the oscilloscope programmatically using SCPI (Standard Commands for Programmable Instruments). For detailed descriptions of commands and programming, please refer to the *UPO7000L Series Digital Phosphor Oscilloscopes-Programming Manual*.

23.2 PC Software Control

The user can send commands to remotely control the oscilloscope via PC software. UPO7000L series oscilloscope require NI-VISA connection.

Operating steps:

(1) Set up the communication between the instrument and the PC.

(2) Open the NI-MAX software and search for the instrument.

(3) Open the remote control panel and send the command.

The oscilloscope can communicate with a PC via the following interfaces.

LAN Control

(1) Connect the device

Connect the oscilloscope to the LAN using a network cable.

(2) Configure network parameters

Refer to "Communication" section to configure the network parameters for the oscilloscope.

(3) Check the device

Open NI-MAX, click "Devices and Interfaces", and view the VISA name corresponding to the network communication address in the oscilloscope's settings.

(4) Remote control

Right-click the device name and select "Open VISA Test Panel" to access the remote command control panel, where the user can send commands and read data.

(5) Load the LXI Web page

Access the LXI web page by entering the oscilloscope's IP address into a web browser. The page provides important information about the instrument, including the model, manufacturer, MAC address, and IP address.

USB Control

(1) Connect the device

Connect the oscilloscope to the LAN using a USB-type-B cable.

(2) Check the device

Open NI-MAX, click "Devices and Interfaces", and view the VISA name corresponding to the USB communication address in the oscilloscope's settings.

(3) Remote control

Right-click the device name and select "Open VISA Test Panel" to access the remote command control panel, where the user can send commands and read data.

23.3 Web Server

Web Server displays the current network switch state. The default network port is set to **80**.

PC Access

The computer and the oscilloscope must be connected to the same LAN and capable of ping each other. The user can view the oscilloscope's local IP address by clicking the setting icon to view, and then can view the oscilloscope's local IP address by IP: 80.

Example

PC IP: 192.168.137.101

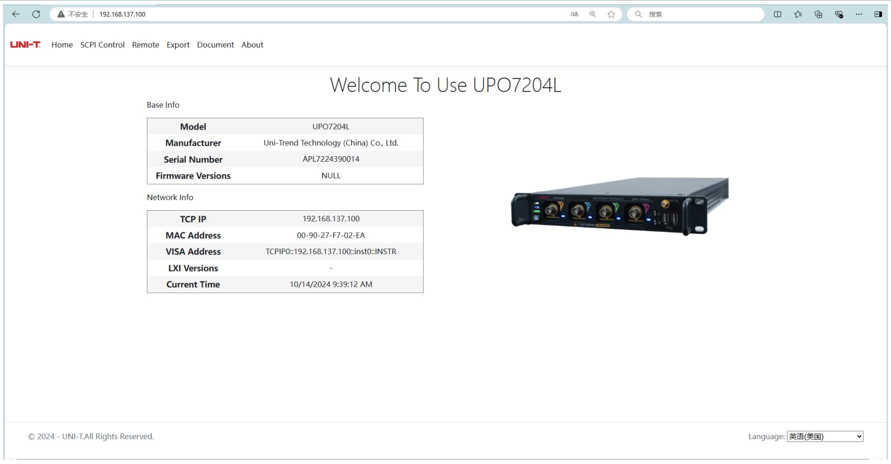
Oscilloscope IP: 192.168.137.100

Gateway: 192.168.137.1

To access the oscilloscope, enter 192.168.137.222:80 in the browser. The following features are available:

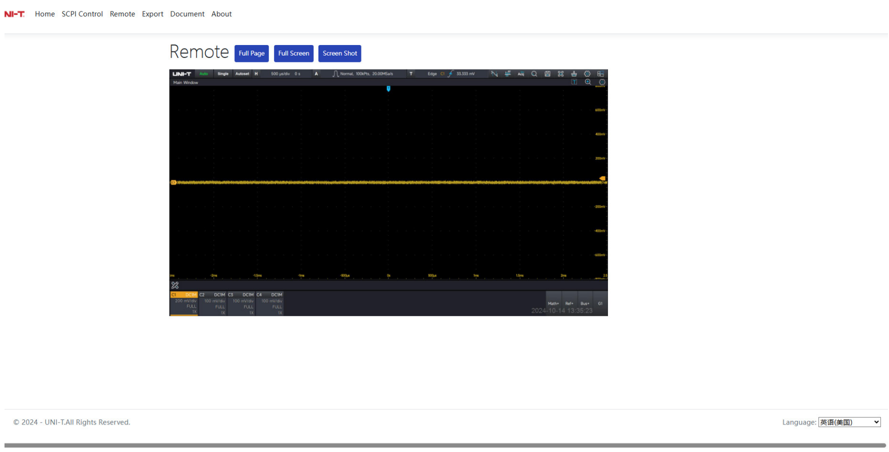
- Device information and remote control: View and control the oscilloscope remotely.
- SCPI control: Send and execute SCPI commands.
- Export data file: Export waveforms and files.

Home page: Displays product model, manufacturer, serial number, hardware version, network information, and communication details.



WebServer Home Page

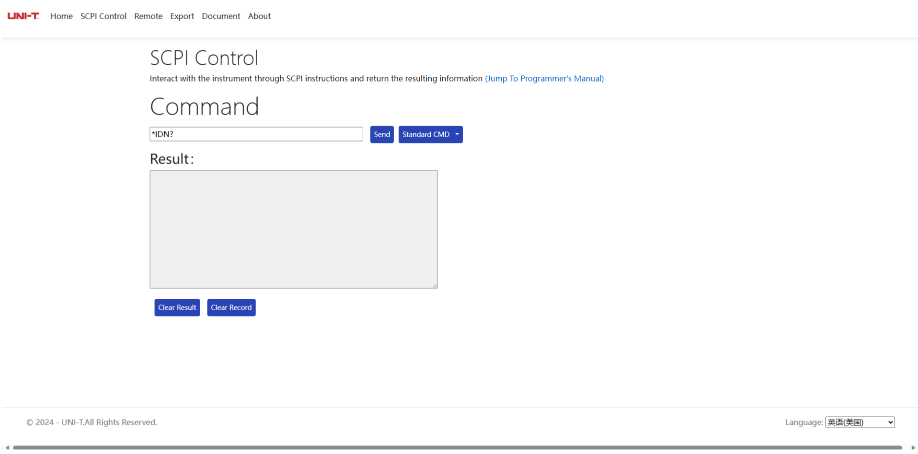
Remote control: The instrument's response can be controlled via the webpage, consistent with its real instrument operation. It supports full display functionality and screenshot capture.



Remote Control

SCPI

The oscilloscope can be operated by sending SCPI (Standard Commands for Programmable Instruments) commands. For detailed information on SCPI, please refer to the *UPO7000L series digital phosphor oscilloscopes-Programming Manual*.



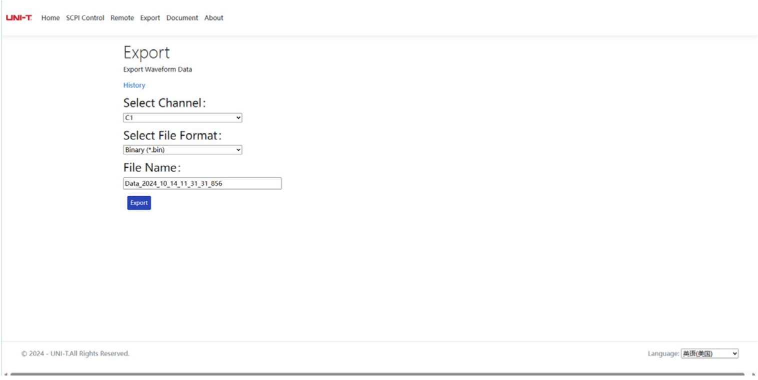
Export file

The waveform file can be exported in various formats through the webpage.

(1) Select channel: C1-C4.

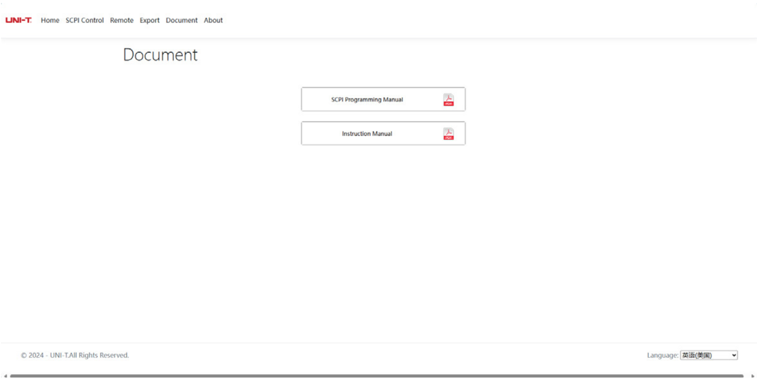
(2) Select file format: Binary (.bin), Text (.txt), Matlab (.mat), Excel (.xlsx), CSV (.csv), TSV (.tsv).

(3) Rename export file.



Document

UPO7000L series digital phosphor oscilloscopes-programming manual and user manual are embedded in the WebServer. Users can click on the corresponding manual to view the instrument operation guide directly.

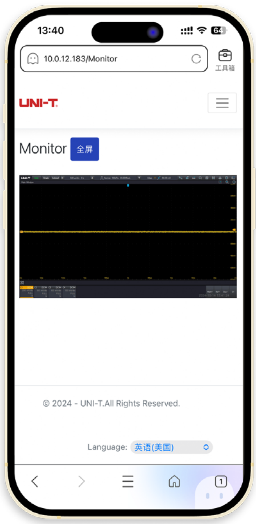


Cellphone Access

The cellphone and the oscilloscope must be connected to the same LAN (usually under the same WLAN band). The user can view the oscilloscope's local IP address on the setting menu and access the oscilloscope via a web browser by entering its IP address followed by IP: 80.

The functionality on the cellphone is identical to that on the computer, with differences only in the layout.

Web Server Home Page



Remote Control

24. Troubleshooting

This section provides a list of potential faults and troubleshooting methods that may occur while using the oscilloscope. If you encounter any of these issues, please follow the corresponding steps to resolve them. If the problem persists, contact UNI-T and provide the equipment information for your device.

(1) If the oscilloscope remains on a black screen without displaying anything after pressing the soft power button.

- Check if the power plug is properly connected and the power supply is normal.
- Check whether the power switch of the oscilloscope is turned on. Once the switch is on, the power soft switch button on the front panel should display a red light. After pressing the start soft switch, the soft power switch button will turn blue, and the oscilloscope will emit a startup sound.
- If a sound is heard, it indicates that the oscilloscope has booted up normally.
- If the product still does not work properly, contact the UNI-T Service Center for assistance.

(2) After signal acquisition, the waveform of the signal does not appear on the screen.

- Check whether probe and DUT are connected properly.
- Check whether the signal connecting line is connected to analog channel.
- Check whether the analog input terminal of the signal matches the channel that is currently selected on the oscilloscope.
- Connect the probe tip to the probe compensation signal connector on the front panel of the oscilloscope and verify whether the probe is functioning properly.
- Check if the device under test is generating a signal. The user can connect the signal-generating channel to the problematic channel to help diagnose the issue.
- Click Autoset to allow the oscilloscope to automatically reacquire the signal.

(3) The measured voltage amplitude value is 10 times larger or 10 times smaller than the actual value.

- Check whether the probe attenuation setting on the oscilloscope matches the attenuation factor of the probe being used.

(4) There is a waveform display, but it is unstable.

- Check the trigger settings in the trigger menu to ensure they match the actual signal input channel.
- Check the trigger type: general signals should typically use the “Edge” trigger. The waveform will display stably only if the trigger mode is set correctly.

- Try changing the trigger coupling to HF rejection or LF rejection to filter out high-frequency or low-frequency noise that may be interfering with the trigger.

(5) Waveform refresh is very slow.

- Check whether the acquisition method is set to "Average" and if the average times are large.
- To accelerate the refresh speed, the user can reduce the number of averaging times or select other acquisition methods.

25. Appendix

25.1 Appendix A Accessory and Option

Order Information	
Product Model	
UPO7204L	Bandwidth of 2GHz, maximum sampling rate of 10GSa/s (single channel: sampling rate of 10GSa/s; dual channel: sampling rate of 5GSa/s; four channels: sampling rate of 2.5GSa/s), four-channel oscilloscope
UPO7104L	Bandwidth of 1GHz, maximum sampling rate of 10GSa/s (single channel: sampling rate of 10GSa/s; dual channel: sampling rate of 5GSa/s; four channels: sampling rate of 2.5GSa/s), four-channel oscilloscope
Accessory	
UT-D14	USB2.0 data cable x 1
UT-P07	Passive high resistance probe 500MHz x 4 set
UT-L45	BNC-BNC x 2
--	National standard power cable x 1
--	Calibration certificate
Option	
UPO7000L-RM	Rack mount kit
Bandwidth Upgrade	
UPO7000L-BW-10T20	UPO7000L series upgrades from 1GHz to 2GHz
Option	
UPO7000L-AWG	Arbitrary waveform generator 60MHz
UPO7000L-JITTER	Jitter and eye-diagram
UPO7000L-PWR	Power analysis
UPO7000L-CANFD	Automobile serial bus triggering and analysis (CAN-FD)

UPO7000L-FLEX	Automobile serial bus triggering and analysis (FlexRay)
UPO7000L-SENT	Automobile sensor serial bus triggering and analysis (SENT)
UPO7000L-AUDIO	Audio serial bus triggering and analysis (I^2S, LJ, RJ, TDM)
UPO7000L-AREO	Aerospace serial bus triggering and analysis (MIL-STD-1553, ARINC 429)
UPO7000L-BND	Upgrade kit (JITTER, PWR, CANFD, FLEX, SENT, AUDIO, AERO)
Probe	
UT-P07A	Passive high resistance probe (1X:8MHz; 10X:500MHz)
UT-PA2000	Active single-ended probe (2GHz)
UT-P20	Passive high voltage probe (100MHz; probe factor 100:1, 1.5kVrms)
UT-V23	Passive high voltage probe (100MHz; 2kVpp)
UT-P21	Passive high voltage probe (50MHz; maximum operating voltage DC 15kVrms)
UT-P40	Current probe (100kHz; 0.4A-60A)
UT-P41	Current probe (100kHz; 0.4A-100 A)
UT-P42	Current probe (150kHz; 0.4A-200A)
UT-P43	Current probe (25MHz; maximum measuring current 20A)
UT-P44	Current probe (50MHz; maximum measuring current 40A)
UT-P4030D	Current probe (100MHz; maximum measuring current 30A)
UT-P4150	Current probe (12MHz; maximum measuring current 150A)
UT-P4500	Current probe (5MHz; maximum measuring current 500A)
UT-4100A	Current probe (600kHz; maximum measuring current 100A)
UT-4100B	Current probe (2MHz; maximum measuring current 100A)
UT-P30	High voltage differential probe (100MHz; ±800Vpp)
UT-P31	High voltage differential probe (100MHz; ±1.5kVpp)
UT-P32	High voltage differential probe (50MHz; ±3kVpp)
UT-P33	High voltage differential probe (120MHz; ±14kVpp)

UT-P35	High voltage differential probe (50MHz; 1.3kV)
UT-P36	High voltage differential probe (50MHz; 5.6kV)

Note: Please order all hosts, accessories, and options from your local UNI-T distributor.

25.2 Appendix B Maintenance and Cleaning

(1) General Maintenance

Keep the probe and its accessories away from the direct sunlight.

Caution: Avoid contact with sprays, liquids, or solvents to prevent probe damage.

(2) Cleaning

Check the probe frequently according to the operating condition. Follow these steps to clean the external surface of the probe:

Use a soft cloth to remove dust from the probe.

Disconnect the power supply and clean the probe with mild detergent or water.

Do not use abrasive or chemical cleaners, as they may damage the probe.

Warning: Please confirm that the instrument is completely dry before use, to avoid electrical shorts or even personal injury caused by moisture.

25.3 Appendix C Limited Warranty and Liability

UNI-T guarantees that the Instrument product is free from any defect in material and workmanship within three years from the purchase date. This warranty does not apply to damages caused by accident, negligence, misuse, modification, contamination, or improper handling. If you need a warranty service within the warranty period, please contact your seller directly. UNI-T will not be responsible for any special, indirect, incidental, or subsequent damage or loss caused by using this device. For the probes and accessories, the warranty period is one year. Visit instrument.uni-trend.com for full warranty information.

Learn more at: www.uni-trend.com

Register your product to confirm your ownership. You will also get product notifications, update alerts, exclusive offers and all the latest information you need to know.

Appendix D Statement and Contact Us

UNI-T is the licensed trademark of UNI-TREND TECHNOLOGY (CHINA) CO., Ltd.

UNI-T products are protected under patent laws in China and internationally, covering both granted and pending patents. Licensed software products are the properties of UNI-Trend and its subsidiaries or suppliers, all rights reserved. This manual contains information that replaces all earlier published versions. The product information in this document subject to update without notice. For more information on UNI-T Test & Measure Instrument products, applications, or service, please contact UNI-T instrument for support, the support center is available on www.uni-trend.com ->instruments.uni-trend.com

https://instruments.uni-trend.com/ContactForm/

Headquarter

UNI-TREND TECHNOLOGY (CHINA) CO., Ltd.
Address: No.6, Industrial North 1st Road, Songshan Lake Park, Dongguan City, Guangdong Province, China
Tel: (86-769) 8572 3888

Europe

UNI-TREND TECHNOLOGY EU GmbH
Address: Affinger Str. 12
86167 Augsburg Germany
Tel: +49 (0)821 8879980

North America

UNI-TREND TECHNOLOGY US INC.
Address: 3171 Mercer Ave STE 104, Bellingham, WA 98225
Tel: +1-888-668-8648