Instruments.uni-trend.com

# UPO6000LP Series High-Resolution Oscilloscopes

# User Manual


The manual applies to:

UPO6004LP, UPO6008LP Series

V1.0

May 2026

# 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.

# 1. Introduction

This manual describes the safety requirements, installation, and operation of the UPO6000LP series high-resolution oscilloscopes.

# 2. 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.

| **Safety Precautions** | |
|---|---|
| **Warning** | **Please follow these guidelines to avoid possible electric shock and risk to personal safety.** |
| | 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.<br>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. |
| **Safety Statements** | |
| **Warning** | "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 performed or observed. Do not proceed to the next step until you fully understand and meet the conditions stated in the "Warning" statement. |
| **Caution** | "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 |

| | | |
|---|---|---|
| | performed or observed. Do not proceed to the next step until you fully understand and meet the conditions stated in the "Caution" statement. | |
| **Note** | "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. | |
| **Safety Sign** | | |
| | **Danger** | It indicates danger of electric shock, which may cause personal injury or death. |
| | **Warning** | It indicates that there are factors you should be cautious of to prevent personal injury or product damage. |
| | **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 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 |
| | **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. |
| | **EFUP** | 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.<br>The AC input voltage of the line reaches the rated value of this device. See the product manual for specific rated value.<br>The line voltage switch of this device matches the line voltage.<br>The line voltage of the line fuse of this device is correct.<br>It is not used to measure the main circuit. |
| **Check all terminal rated values** | | Please check all rated values and marking instructions on the product to avoid fire and impact of excessive current. Please consult the product manual for detailed rated values before connection. |
| **Use the power cord properly** | | The user 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 4 kV for contact discharge and 8 kV for air discharge. |
| **Measurement accessories** | Measurement accessories are of lower class, which are definitely not applicable to main power supply measurement, CAT II, CAT III or CAT IV circuit measurement. Probe subassemblies and accessories within the range of IEC 61010-031 and current sensor within the range of IEC 61010-2-032 can meet its requirements. |
| **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. |
| **Correctly connect and disconnect** | Do not plug or unplug the probe or test lead when it is connected to a voltage source.Only use insulated voltage probes, test leads, and adapters that are supplied with the product or specified by UNI-T as suitable for use with the product. |
| **Power fuse** | Please use power fuse of specified 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 to operators inside. Do not remove the protective cover.<br>Maintenance must be carried out by qualified personnel. |
| **Service environment** | This device should be used indoors in a clean and dry environment with ambient temperature from 0 °C to +40 °C.<br>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** | Do not use this device in a flammable and explosive environment to |

| | |
|---|---|
| **flammable and explosive environment** | 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.<br>Please ensure adequate ventilation and leave a gap of at least 15 cm 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** | Poor 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 carried out by qualified personnel. |

## 2.1. Environmental Requirements

This instrument is suitable for the following environment.

- Indoor use
- Pollution degree: Class 2
- Overvoltage category: This product should be powered from a mains supply that complies with Overvoltage Category II, which is a typical requirement for connecting equipment via power cords and plugs.
- Operating: Altitude below 4,600 meters; non-operating: Altitude below 15,000 meters.
- Unless otherwise specified, the operating temperature range is from 0 °C to +50 °C, and the storage temperature range is from -40 °C to +70 °C.
- Humidity: Below +10 °C: Any relative humidity (RH); +10 °C to +30 °C: 5% to 95% RH; +30 °C to +40 °C: 5% to 75% RH; +40 °C to +50 °C: 5% to 45% RH.

There are ventilation outlets on the rear and side panels of the instrument. Please ensure that these openings are not obstructed to allow proper airflow. To prevent excessive dust from blocking the vents, clean the instrument housing regularly. The housing is not waterproof. Disconnect the power supply before cleaning, and wipe the housing with a dry cloth or a slightly damp, soft cloth.

## 2.2. Connecting Power Supply

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

UPO6004LP Series:

| Voltage Range | Frequency | Power |
|---|---|---|
| 100 V-240 VAC (Fluctuations ±10%) | 50 Hz /60 Hz | 140 W |

UPO6008LP Series:

| Voltage Range | Frequency | Power |
|---|---|---|
| 100 V-240 VAC (Fluctuations ±10%) | 50 Hz /60 Hz | 160 W |

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 cable provides reliable protective earthing of the instrument enclosure. The oscilloscope is equipped with a three-conductor power cord that complies with international safety standards and meets the protective grounding requirements 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.

## 2.3. 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.

# 3. UPO6000LP Series Overview

UPO6000LP series high-resolution oscilloscopes includes six models.

| Model | Analog Channel | Analog Bandwidth | Gen |
|---|---|---|---|
| UPO6**104LP** | 4 | 1 GHz | ○ |
| UPO6**054LP** | 4 | 500 MHz | ○ |
| UPO6**034LP** | 4 | 350 MHz | ○ |
| UPO6**108LP** | 8 | 1 GHz | × |
| UPO6**058LP** | 8 | 500 MHz | × |
| UPO6**038LP** | 8 | 350 MHz | × |

○: Option ●: Standard ×: Not support

The UPO6000LP is a high-resolution oscilloscope that integrates four or eight analog channels and an arbitrary waveform generator (AWG). It offers a bandwidth of up to 1 GHz, a maximum real-time sampling rate of 5 GSa/s, and 12-bit ADC resolution, providing high-speed signal acquisition and deep memory performance.

With a maximum memory depth of 500 Mpts/CH, the oscilloscope supports long-duration capture and detailed analysis of high-speed signals. It enables fast and accurate measurement and analysis, meeting application requirements in fields such as communications, automotive electronics, industrial automation, aerospace, medical electronics, scientific research, and electronic design and development.

# 4. Document Overview

This user manual is designed to help users quickly understand the front panel, rear panel, user interface, and basic operations of the UPO6000LP series high-resolution oscilloscopes.

**Note**: The latest edition of the user manual can be downloaded from the UNI-T website: https://www.uni-trend.com

(1) Software Version

The software update may change or add new functions. Please visit the UNI-T website for the latest version or contact UNI-T to upgrade the software.

(2) Document Format

a. Button

A button with a character frame represents a key on the front panel. For example, Default represents the Default button.

b. Menu

Double quotation marks represent a menu or a pop-up menu. For example, Channel Setting pop-up menu on the operation interface allows you to click Vertical Scale to operate and set the vertical scale settings.

c. Operation Step

Use an arrow “>” represent the next step. For example, Storage > Save represents that in the Storage Menu, you should first click Storage and then click Save to save the waveform, setting, or image file.

d. Connector

“Square brackets + Word” represents a connector on the front or rear panel. For example, [AUX OUT].

e. Hyperlink

“Underline + Blue text” represents a hyperlink. For example, Connecting Power Supply.

# 5. Getting Started Guide



This chapter introduces the initial use of oscilloscope, the front and rear panels, operation panel, the user interface, parameter setting, and remote control.

## 5.1. General Inspection

It is recommended to inspect the instrument by following the steps below before using the UPO6000LP series oscilloscopes for the first time.

(1) Check for Damages caused by Transport

If the packaging carton or the foam plastic cushions are severely damaged, please contact the UNI-T distributor of this product immediately.

(2) Check Accessories

The details of the supplied accessories are described in the UPO6000LP Series High-Resolution Oscilloscopes accessories section in this manual. Please refer to this section for the list of accessories. If any accessories are missing or damaged, contact UNI-T or the local distributors.

(3) Machine Inspection

If the instrument appears to be damaged, not working properly, or has failed the functionality test, please contact UNI-T or the local distributors.

If the equipment is damaged during shipping, retain the packaging and notify both the carrier and UNI-T, UNI-T will arrange maintenance or replacement.

## 5.2. Before Use

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

(1) Connecting Power Supply

Refer to Connecting Power Supply, use the assembled power line or other power line that meets the local country standards to connect the oscilloscope. When the oscilloscope is connected to the power supply, the soft power switch at the lower left turns red. Press the switch to power on the oscilloscope.

(2) Boot Check

Press the soft power switch button and the indicator should change from red to green. The oscilloscope will show a boot animation and then enter the normal interface.

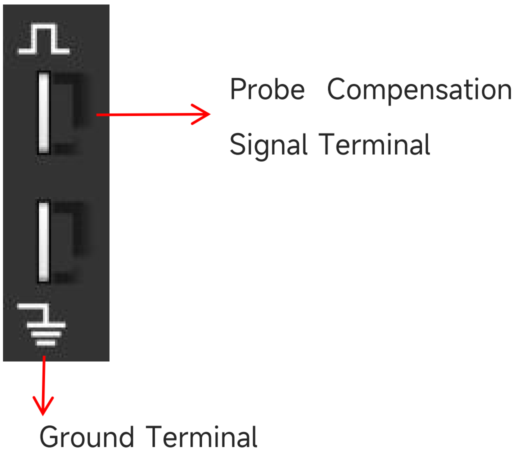
(3) Connect the monitor and the mouse

Connect the HDMI port on the rear panel of the device to an external display using an HDMI cable, and connect a mouse to the USB Host port on either the front or rear panel of the device. After the connections are made, the device can be used as a standalone oscilloscope. The HDMI port supports a resolution of 1920 × 1080.

(4) Connecting Probe

This oscilloscope is supplied with one probe for compensation. Connect the probe BNC connector to the C1 input BNC connector on the oscilloscope. Connect the probe tip to the probe compensation signal output and connect the probe ground alligator clip to the corresponding ground terminal.

The probe compensation signal output provides a waveform with an amplitude of approximately 3 Vpp and a default frequency of 1 kHz.




Probe Compensating Signal Terminal and Ground Terminal

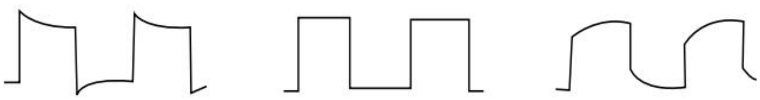
(5) Function Check

Click AUTO icon on the toolbar. A square wave (amplitude 3 Vpp, frequency 1 kHz) should appear on the screen. Repeat Step 4 to check all channels.

(6) Probe Compensation

When the probe is connected to any input channel for the first time, probe compensation should be adjusted to match the probe and the input channel. An uncompensated probe may lead to measurement errors or mistakes. Please follow the following steps to adjust the probe compensation.

- Set the probe attenuation ratio to 10x in the probe menu, and set the probe switch to 10x. Connect the oscilloscope probe to C1. If a probe hook tip is used, ensure that it makes stable contact. Connect the probe tip to the "Probe Compensation Signal Terminal" on the oscilloscope, and connect the ground alligator clip to the corresponding ground terminal. Open C1 on the oscilloscope and Click AUTO icon on the toolbar.
- View the displayed waveform, as shown in the following figure.



Excessive Compensation Correct Compensation Insufficient Compensation

Probe Compensation Calibration

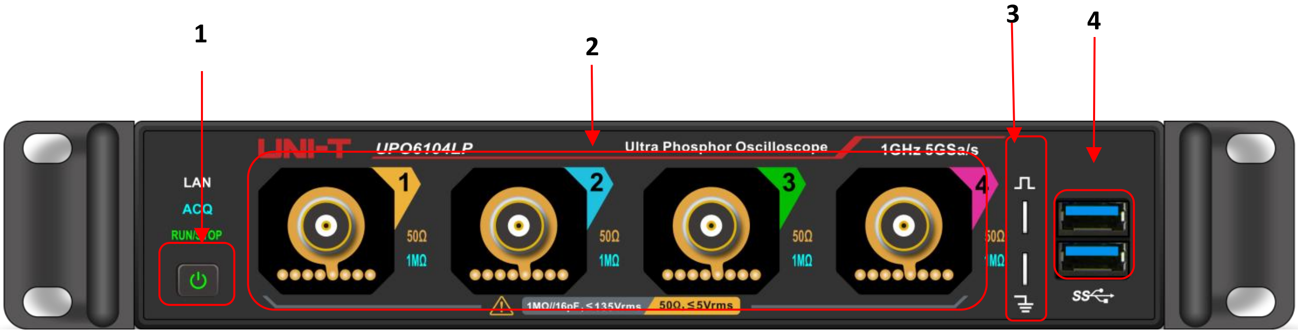
- If the displayed waveform resembles the above Insufficient Compensation or Excessive Compensation, use a non-metallic screwdriver to adjust the probe's variable capacitance until the display matches the Correct Compensation waveform.

**Note**: The probe type is UT-P07A and UT-P08A. When connected to the oscilloscope, the probe ratio will be automatically identified as X10.

**Warning: To avoid electric shock when using the probe to measure high voltage, please ensure that the probe insulation is in good condition and avoid physical contact with any**

**metallic part of the probe.**

## 5.3. Front Panel




Front Panel

| No. | Description |
|---|---|
| 1 | Soft Power Switch |
| 2 | Analog Channel Input ① |
| 3 | Probe Compensation Signal Terminal and Ground Terminal ② |
| 4 | USB HOST |

① Analog channel input terminal: Connect the oscilloscope probe or BNC cable to these BNC connectors to input a signal into the oscilloscope.

② Probe compensation signal terminal and ground terminal: Connect the probe BNC connector to the C1 input BNC connector on the oscilloscope. Connect the probe tip to the probe compensation signal output, and connect the probe ground alligator clip to the corresponding ground terminal. This setup outputs the oscilloscope's internal compensation signal. For more details, refer to *Connecting Probe* in Before Use - Connecting the Probe.

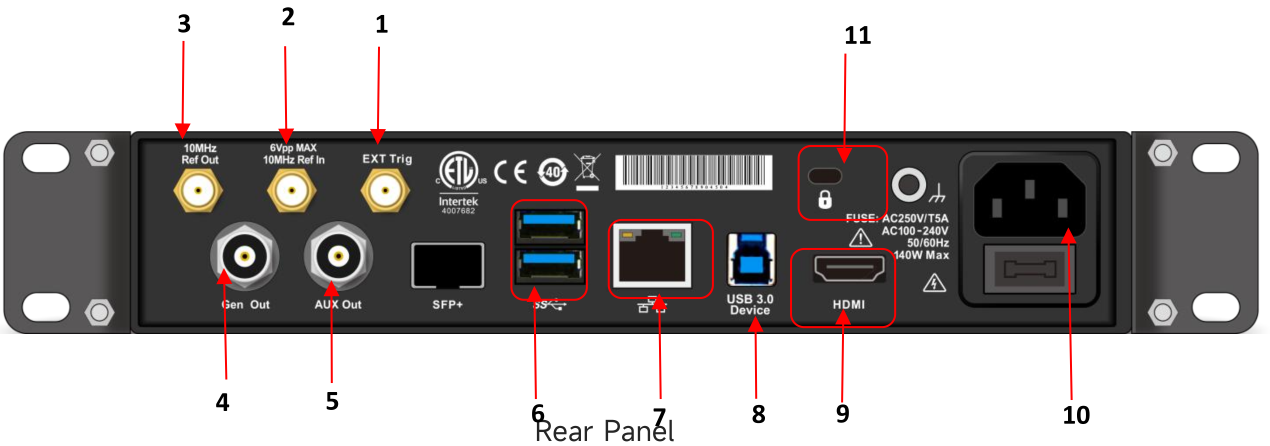
③ Indicator

| Indicator | Red | Green | White | Blue | Yellow | None |
|---|---|---|---|---|---|---|
| **Power** | Powered on but not enabled | Enabled | | | | |
| **RunStop** | Stop | Run | | | | |
| **Lan** | | | Internet connected normally | | | Internet connection error |
| **Acq** | | | | Solid: Acquiring Blinking: Waiting for acquisition | | |

| | | | | | | |
|---|---|---|---|---|---|---|
| **Impedance** | | | | 1MΩ | 50Ω | Channel disabled |

**Note**: The output of the signal generator is equipped with overvoltage protection. If the input voltage exceeds ±9 V, the overvoltage protection will be activated.

## 5.4. Rear Panel




Rear Panel

1. EXT Trig: External trigger input terminal.
2. 10MHz Ref In: Inputs an external 10 MHz reference clock signal.
3. 10MHz Ref Out: Outputs a 10 MHz reference clock signal generated by internal crystal oscillator of the instrument.
4. Gen Out: Outputs signals such as continuous wave (CW), amplitude modulation (AM), frequency modulation (FM), amplitude shift keying (ASK), frequency shift keying (FSK), and frequency sweep via a BNC connector.
   Note: This function is not available on 8-channel models.
5. AUX OUT: Output terminal for trigger output, Pass/Fail output, and DVM output
6. USB HOST: Supports USB flash drive.
7. LAN: Connects to the local area network for remote control.
8. USB Device: Provides communication between the oscilloscope and a PC.
9. HDMI: Supports to connect an external displayer with HDMI port.
10. AC power input: Use the assembled power cable to connect the oscilloscope to AC power. For more details, refer to the Connecting Power Supply.
11. Safety lock: Lock the oscilloscope at fixed position (sold separately).

**Warning**

During measurement, the ground terminal of instrument must be properly grounded.

# 5.5. Operation Panel

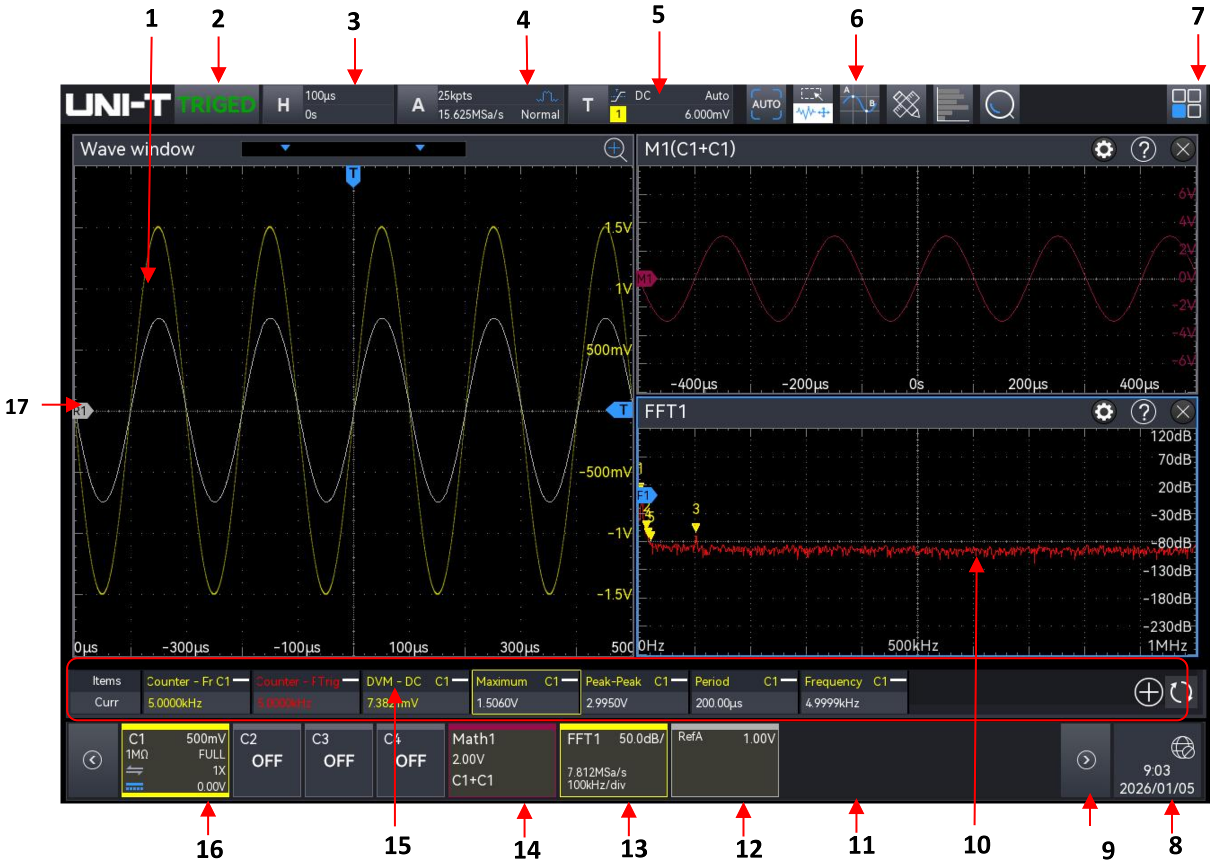
(1) Home Menu

Press the Home icon in the upper-right corner to open the Home quick menu, including: Measure, Cursor, DVM (digital voltmeter), Counter (frequency counter), Math, Ref (reference), Gen (signal generator), Bode (Bode plot), Search, Decode, Navigation, Record, Mark, FFT, Pwr Analy (power analysis), Pass/Fail, Display, Utility, Help, Area Draw, Auto, Default, Single, and Power off.

The Home quick menu provides quick access to the corresponding function menus.


# 5.6. User Interface

The UPO6000LP series can be configured as a standalone oscilloscope by connecting an external display to the rear-panel HDMI port and a mouse to a USB host port on either the front or rear panel.




1. Display: C1-C4 waveform measurement window, Ref waveform, and Math waveform.
2. Trigger state icon: TRIGED, AUTO, READY, STOP, ROLL, and SCAN.
3. Horizontal base label: Displays the current horizontal time base, click to enter the horizontal setting menu.
4. Sampling rate and memory depth label: Displays the current sampling rate and memory depth, click to enter the horizontal setting menu.
5. Trigger info label: Displays the trigger information, including the trigger type, source, level, and mode. Click label to open the Trigger Setting window and adjust the trigger parameters.
6. Function toolbar: Displays the currently added functions in toolbar. Touch an icon to enter the corresponding function menu.
7. Home menu: Open the function guide menu, click each function key to enter the corresponding menu.
8. Notification: Displays the USB, LAN connection icon, and time. Click this area to open the setting menu. For more details, refer to the Notification Setting.
   - USB: Displays a USB icon when the instrument detects that a USB device is connected.
   - LAN: Displays a LAN icon when the LAN connection is successfully established.
   - Time: Displays the current time and allows you to set the system time.
9. Volts/div signal bar: When the volts/div has multiple info window at the bottom of the screen,

press the arrow keys ,  to move to left or right and reveal the hidden box.

10. Multi-window display area: If multiple functions are enabled simultaneously, such as XY, Math, and FFT, multiple function windows can be displayed at the same time.
11. Ref label: Displays the status of Ref1-Ref4 and the vertical scale setting. Up to four Ref labels can be displayed.
12. FFT label: Displays the status of FFT1-FFT4 switch, vertical scale, sampling rate, and the frequency of each div. Up to 4 FFT labels can be displayed.
13. Math label: Displays the status of M1-M4 switch, vertical scale, and operation type. Up to 4 Math labels can be displayed.
14. Measured result display window: Displays counter results, DVM measurements, parameter measurements, and statistical results. This window can be enabled or disabled.
15. Channel label: Displays the status of C1-C4, vertical scale, impedance, bandwidth limitation, inverse phase, channel coupling, probe attenuation ratio, and vertical bias.
16. Analog channel label: Displays the C1-C4 icons. Each channel icon matches the color of its corresponding waveform.

## 5.7. Parameter Setting

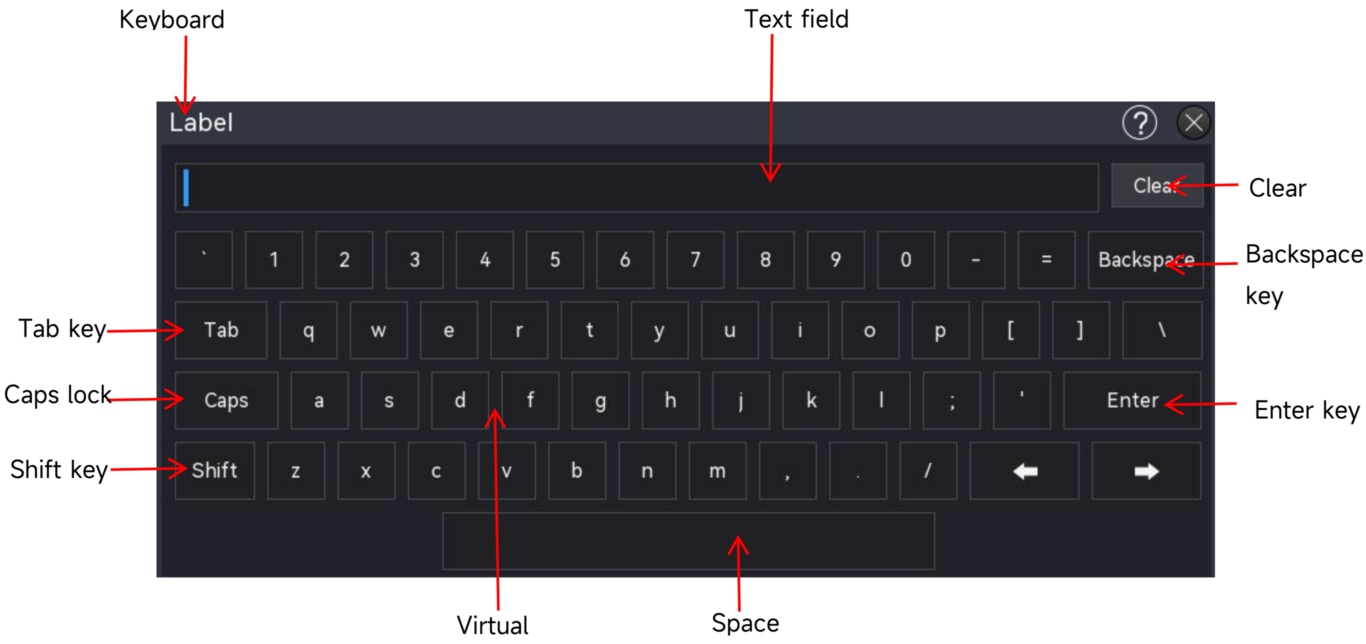
The UPO6000LP series supports parameter configuration using the Virtual Keyboard. The setup procedure is as follows.

(1) Virtual Keyboard

Once the parameter or input field has been selected with the mouse, double-click to open the virtual keyboard and enter a parameter value, label name, or file name.

1. Enter character string

When naming a file or folder, use the character keyboard to enter a character string.




a. Input field

Enter text using letters, numbers, and special characters. The maximum length is 255 characters.

b. Clear key

Press the Clear key to delete all contents in the input field.

c. Caps key

Press the Caps key to switch between upper and lower case.

d. Tab key

Press the Tab key to enter 2 spaces at a time.

e. Shift key

Press the Shift key to switch among number, special character, upper and lower case.

f. Arrow keys (left, right)

If part of the content needs to be changed, press the ←, → key to move the cursor to left or right and then to edit the content.

g. Space key

Press the Space key to enter one space in the input field.

h. Backspace key

Press the Backspace key to delete a single character. This key is used to delete a character when the input field contains a large amount of content.

i. Enter key

Once the content has been entered, press the Enter key to confirm the setting and close the virtual keyboard.

(2) Numeric Keypad

When setting or editing a parameter, use the numeric keypad to enter the numeric value.

1. Click number or unit to enter.


After entering all the values and selecting the desired units, the numeric keypad will automatically close, completing the parameter setting. Additionally, the user can manually close the numeric keypad by clicking the confirm key. In this case, the unit defaults to the currently set unit. On the numeric keypad, the following operations are also available:

a. Delete the entered parameter value.

b. Set the parameter to the maximum or minimum value (sometimes specifically the maximum or minimum value for the current state).

c. Set the parameter to the default value.

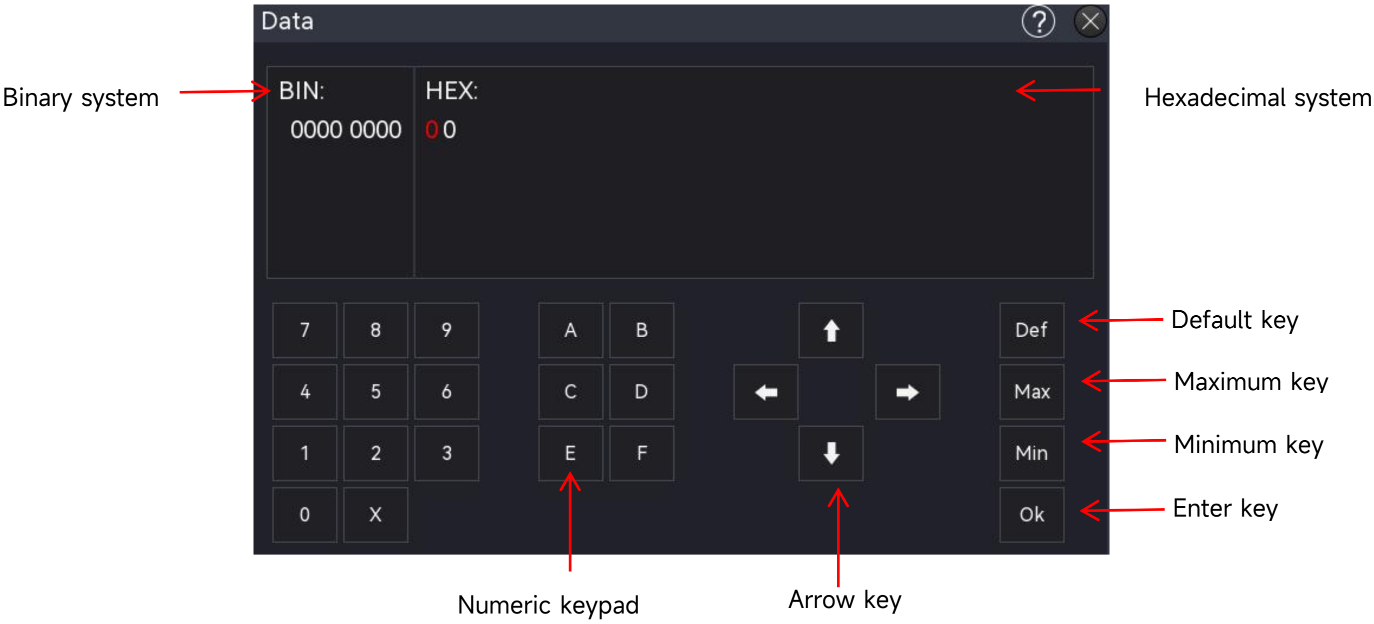
d. Clear the parameter input field.

e. Move the cursor to modify the parameter value.

2. Enter binary, hexadecimal system value

During the decoding triggering, use the numeric keypad to enter binary or hexadecimal values for data and address settings.

Enter Method: Tap to select the number or input field to be edited, and then use the numeric keypad to enter the desired numeric or letter values.




3. After entering all the values and pressing the Ok button, the numeric keypad will automatically close, completing the parameter setting. Additionally, on the numeric keypad, the following operations are also available:
   a. Move the cursor to modify the parameter value.
   b. Set the parameter to the maximum or minimum value (sometimes specifically for the current state).
   c. Set the parameter to the default value.
   d. Clear the parameter input field.

## 5.8. Remote Control

UPO6000LP series high-resolution oscilloscopes can communicate with a PC via USB and LAN ports for remote control. Remote control is implemented using SCPI (Standard Commands for Programmable Instruments).

UPO6000LP series has three methods for remote control.

(1) Custom Programming

The user can perform the programming control on the oscilloscope through SCPI (Standard Commands for Programmable Instruments). For detailed descriptions on command and programming, please refer to *UPO6000LP Series High-Resolution Oscilloscopes-Programming Manual*.

(2) PC Software Control (Instrument Manager)

The user can use a PC software to remotely control the oscilloscope. The instrument manager can display the oscilloscope screen in real time and control the operation with the mouse. It is recommended to use the PC software provided by UNI-T. It can be downloaded from UNI-T official website (https://www.uni-trend.com).

Operating steps:

- Set up the communication between the instrument and a PC.
- Open the instrument manager software and search the instrument source.
- Right-click to open the oscilloscope, operate the instrument manager to remotely control the oscilloscope. For more details, refer to *Instrument Manager-User Manual*. This device supports remote control through communication with a computer via USB and LAN interfaces. Remote control is implemented using the SCPI command set, allowing users to operate and configure the device remotely.

(3) Web Control

Once the network is connected, users can access a web page via the IP address. After logging in with the username and password, users can control the device remotely. The Web Control feature displays the instrument screen interface in real-time. It supports web access from PCs, smartphones, and iPads, and allows both internal and external remote control of the device. For more details on logging into Web Control, refer to the Web Access section.

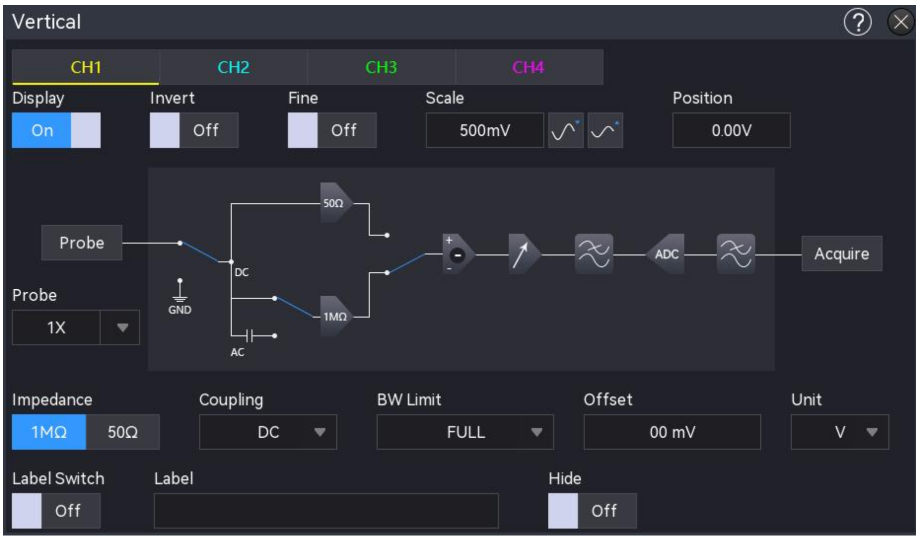
# 6. Vertical System



UPO6000LP provides an independent vertical control system for each channel. The vertical system setup method is the same for all channels. This chapter introduces the vertical channel settings using C1 (CH1) as an example.

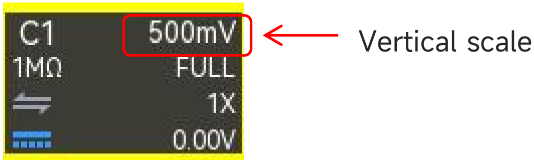
Access the Channel Setting menu using the following method.

- When the channel is active, tap the Channel label to enter the Channel Setting menu.



## 6.1. Vertical Scale

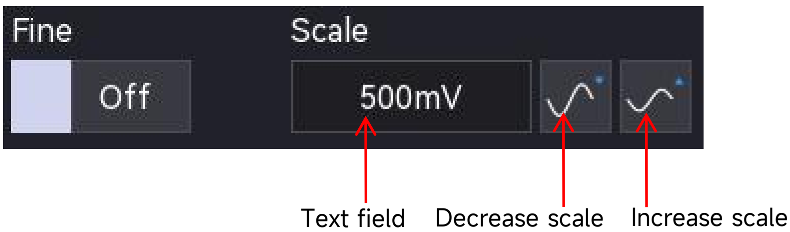
The vertical scale represents the voltage value per division in the vertical direction, typically expressed as V/div. When adjusting the vertical scale, the waveform's amplitude will increase or decrease, and the scale displayed in the channel label at the bottom of the screen will update in real-time (as shown in the figure below).




The vertical scale range depends on the currently selected probe and input impedance. The default probe attenuation ratio is 1X. When the input impedance is set to 1 MΩ, the vertical scale range is from 500 μV/div to 10 V/div. When the input impedance is set to 50 Ω, the vertical scale range is from 500 μV/div to 1 V/div.

When C1 is active, adjust the vertical scale using the following steps.

- Move the mouse pointer to the channel label at the bottom of the screen, and scroll the mouse wheel to change the vertical scale.
- Open the Channel Setting menu, select C1, and Double-click Vertical Scale menu to open the numeric keypad and enter the desired value. Alternatively, click decrease , increase icons of the vertical scale label to adjust the vertical scale.




In the Channel Setting menu, the adjustment method can be set to Coarse tuning or Fine tuning. ON indicates coarse tuning, and OFF indicates fine tuning. The default is coarse tuning.

- Coarse tuning: Adjusts the vertical scale within the allowable range using the 1-2-5 step sequence.
- Fine tuning: Adjusts the vertical scale in 1% increments of the current vertical scale range. For example, if the current scale is 500 mV, each fine-tuning step is 500 mV × 1%. If the current scale is 1 V, each step is 1 V × 1%.

## 6.2. Open/Activate/Close Analog Channel

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

(1) Open the analog channel

- Tap the channel label at the bottom of the screen to turn on C1.
- In Channel Setting menu, select C1 and set it to ON to turn on C1, or set it to OFF to turn off C1.

(2) Close the analog channel

- When C1 is opened and in the activated state, double-click channel label at the bottom of the screen to turn off C1.
- When C1 is opened but not in the activated state, first activate C1, then then Double-click channel label at the bottom of the screen to turn it off.
- Open the Channel Setting menu and select C1, set it to OFF to turn off C1.

(3) Activate the analog channel

When multiple channels are open simultaneously, only the activated channel can be adjusted (a channel can only be activated in the open state). The vertical scale, vertical position, and channel settings of the activated channel can be adjusted.

- Tap the channel label at the bottom of the screen to activate C1.
- Open the Channel Setting menu and select C1 to activate it.


Activated State


Open but not Activated


Off State

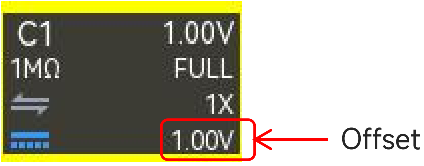
## 6.3. Offset

During measurements with an oscilloscope, small offsets in a channel's zero voltage may occur due to component temperature drift or external environmental interference, which can affect vertical parameter measurements. The UPO6000LP Series allows users to apply a zero offset voltage to calibrate the channel's zero point, thereby improving measurement accuracy.

Adjusting the vertical offset in the channel label at the bottom of the screen updates the display in real time (as shown in the following figure). The range of the vertical offset is related to the current input impedance, probe attenuation ratio and the vertical scale.

| Bandwidth | Vertical Scale | Offset Range |
|---|---|---|
| 1 GHz | 500 μV/div to 100 mV/div | ± 2 V (50 Ω or 1 MΩ) |
| | 102 mV/div to 1 V/div | ± 5 V (50 Ω) |
| | 102 mV/div to 1 V/div | ± 20 V (1 MΩ) |
| | 1.02 V/div to 10 V/div | ± 200 V (1 MΩ) |
| 500 MHz<br>350 MHz | 500 μV/div to 50 mV/div | ± 2 V (50 Ω or 1 MΩ) |
| | 51 mV/div to 1 V/div | ± 5 V (50 Ω) |

| | | |
|---|---|---|
| | 51 mV/div to 1 V/div | ± 25 V (1 MΩ) |
| | 1.02 V/div to 10 V/div | ± 250 V (1 MΩ) |




When C1 is activated, the vertical offset can be set by the following steps.

- Open the Channel Setting menu, select the C1 label, and Double-click Offset input field to open the numeric keypad and directly enter the offset value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

## 6.4. Position

Vertical position refers to the vertical display location of the channel waveform relative to the center of the screen. Adjusting the vertical position moves the waveform up or down on the display without changing the measured voltage values.


When C1 is active, adjust the vertical position using the following steps.

- Open the Vertical menu, double-click value input field to open the numeric keypad and directly enter the position value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

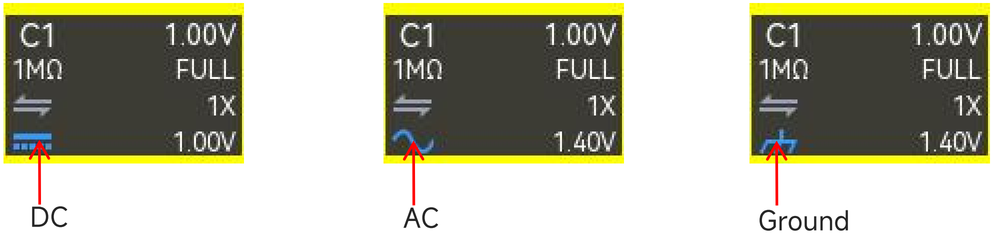
## 6.5. Channel Coupling

The channel coupling is used to filter out unwanted signals. For example, the measured signal is a signal containing DC offset.

Click channel label at the bottom of the screen to open the Channel Setting menu, and then click Coupling to select the coupling mode.

- DC: When the coupling mode is set to DC, the measured signal containing DC component and AC component can all be passed through.
- AC: When the coupling mode is set to AC, the measured signal containing DC component will be blocked.
- Ground: When the coupling mode is set to Ground, the measured signal containing DC component and AC component will all be blocked.

Once the channel coupling is set, the channel coupling mode will display in the channel label at the bottom of the screen, as shown in the following figure.




## 6.6. Bandwidth Limit

The bandwidth limit is used to decrease the noise in the waveform. It is mainly used to reduce high-frequency noise in a signal when observing low-frequency signals. For example, the measured signal is a pulse signal containing high-frequency oscillation.

Click channel label at the bottom of the screen to open the Channel Setting menu, and then click Bandwidth Limit to select a bandwidth limit value. When the bandwidth limit is enabled, the value of bandwidth limitation will be displayed in the channel label at the bottom of the screen, as shown in the following figure.


The UPO6000LP series can be set with a bandwidth limit of 20 MHz or FULL.

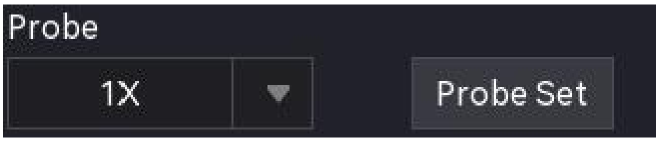
- 20 MHz: When the measured signal contains a high-frequency that is greater than 20 MHz, it will be attenuated.
- FULL: The measured signal containing the high frequency can be passed through.

## 6.7. Probe Attenuation Ratio

(1) Probe

To match the probe's attenuation coefficient, set the probe attenuation ratio in the channel operation menu. For example, with a probe attenuation coefficient of 10:1, the probe attenuates the measured signal by a factor of 10 before it reaches the oscilloscope. Therefore, the probe coefficient in the oscilloscope's channel menu should be set to ×10, meaning the incoming signal will be multiplied by 10 to ensure that the oscilloscope reads the correct voltage.

Click the channel tab at the bottom of the screen to open the “Channel Settings” menu. Then, click the “Probe attenuation ratio” drop-down menu to select the appropriate probe attenuation ratio.




Probe attenuation ratio: 0.001X, 0.01X, 0.1X, 1X, 10X, 100X, 1000X, and a custom value.

When the channel unit is set to A, it indicates a current probe, and the probe attenuation ratio can be adjusted to 5 mV/A, 10 mV/A, 50 mV/A, 100 mV/A, 200 mV/A, 5000 mV/A, 1 V/A, or a custom value.

(2) Probe Setting

When the oscilloscope detects a probe with a probe attenuation ratio detection pin (different resistors represent different attenuation ratios), it automatically recognizes the probe's attenuation coefficient, sets the probe attenuation ratio to the corresponding value, and displays information such as the manufacturer, probe model, serial number, and probe magnification in the probe setting menu.

Click Probe to enter the Probe Setting menu. In this menu, the user can configure the probe indicator, probe key function, and view the corresponding manufacturer, probe model, serial number, and probe attenuation ratio.

a. Probe Indicator: Click Probe Indicator to turn it ON or OFF.

b. Probe Key Function: Click Probe Key drop-down menu to set Run/Stop, Single, or Screenshot.

- Run/Stop: Press the active probe key to switch between Run and Stop modes.
- Single: Press the active probe key to switch the oscilloscope trigger mode to Single.
- Screenshot: Press the active probe key to automatically capture a screenshot.

## 6.8. Inverse Phase

Click channel label at the bottom of the screen to open the Channel Setting menu, and then click Inverse Phase to sey it on or off. When the inverse phase is enabled, an icon  in the channel label will be illuminated.

Inverse phase: ON


When the inverse phase is disabled, the waveform is displayed normally. When the inverse phase is enabled, the waveform voltage will be reversed, and the results of math operation and waveform measurement will also be changed, as shown in the following figure.

Inverse Phase: OFF

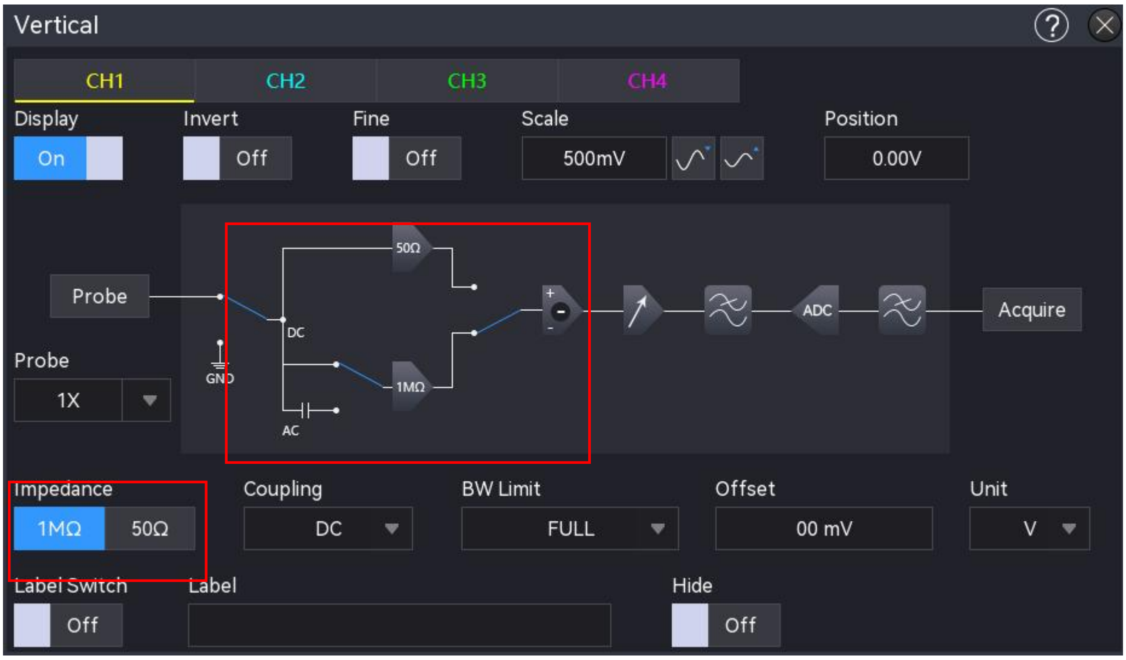
Inverse Phase: ON

## 6.9. Impedance

To reduce the electric load cause by the interaction of the oscilloscope and the measured circuit, this oscilloscope provides two input impedance modes: 1 MΩ (default) and 50 Ω. In the Channel Setting menu, click Impedance key to select the input impedance to 1 MΩ or 50 Ω.

- 1 MΩ: The input impedance of oscilloscope is extremely high at this time, so the current flowing into the oscilloscope from the measured circuit can be ignored.
- 50 Ω: Match the oscilloscope to a device with an output impedance of 50 Ω.

  The circuit diagram in the Channel Setting menu will change with the input impedance, as shown in the following figure.

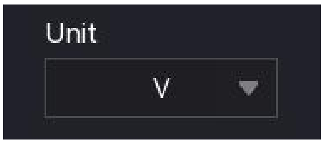



- The input impedance setting affects the range of the channel vertical scale and vertical offset.

**Note**: After the oscilloscope automatically recognizes the probe, the input impedance is also automatically set to the appropriate mode, and manual setting is not required.

## 6.10. Unit

Click channel label at the bottom of the screen to open the Channel Setting menu, and then Click Unit to set the unit to V, A, W, or U. The default unit is V. When using the current probe, the unit switches to A. Once the unit is set, the unit in the channel label and the measurement unit will also be changed.




## 6.11. Channel Fold

Click Channel Fold to set the channel waveform state to hide (ON) or display (OFF).
ON: Hides the channel waveform.
OFF: Displays the channel waveform normally.

## 6.12. Label

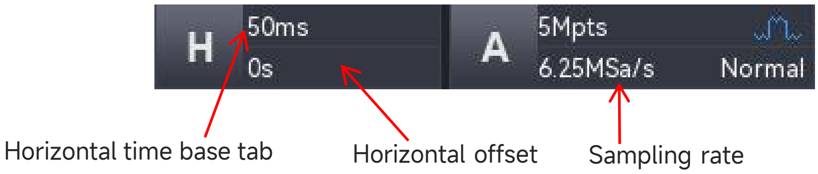
The instrument uses channel number to identify the channel by default. Users can assign a custom name to each channel according to their preferences, for example, CH1.
Click channel label at the bottom of the screen to open the Channel Setting menu, and then click Label to select display (ON) or not display (OFF) the channel label. The channel label can also be set by Double-click input field to open the virtual keyboard and directly enter the character string. For details on the use of the virtual keyboard, refer to the section of 5.7 Parameter Setting.

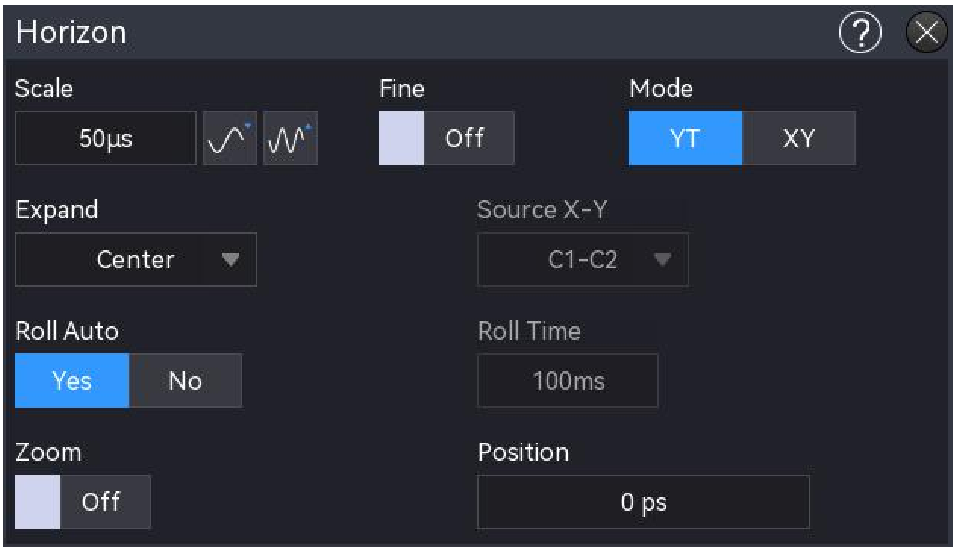
# 7. Horizontal System



Access the Horizontal Control System menu using the following method.

- Tap the H (Horizontal Scale) label at the top of the screen to enter Horizontal menu.




## 7.1. Horizontal Scale

The horizontal scale is also known as the horizontal time base, i.e., the time value represented by each division in the horizontal direction of the screen, usually expressed as s/div. The range of horizontal scales as shown in the following table. When adjusting the horizontal time base, it changes in step of 1-2-5, i.e. 500 ps/div, 2 ns/div, 5 ns/div…500 s/div, and 1 ks/div.

| Bandwidth | Horizontal Scale |
|---|---|
| **1 GHz** | 500 ps/div to 1 ks/div |
| **500 MHz** | 500 ps/div to 1 ks/div |
| **350 MHz** | 1 ns/div to 1 ks/div |

When the horizontal time base is changed, the waveforms of all channels are horizontally expanded or compressed relative to the currently selected horizontal extension reference (see Horizontal Extension), and the horizontal time base displayed in the upper-left corner updates in real time, as shown in the following figure.


Set the horizontal time base using the following steps.

- Move the mouse pointer to the H label at the top of the screen, and scroll the mouse wheel to change the horizontal time base.
- Click Horizontal Scale label to open the Horizontal Scale menu, and Double-click horizontal scale input field to open the numeric keypad and directly enter the desired value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. Alternatively, click decrease , increase icons of the horizontal scale label to adjust the horizontal scale.


UPO6000LP series supports horizontal adjustment.

- Coarse tuning: Click decrease , increase icons on the right side of the horizontal scale label to adjust the horizontal scale. The horizontal time base for all channel waveforms will be adjusted within the allowable range using the 1-2-5 step sequence.
- Fine tuning: Click decrease , increase icons on the right side of the horizontal scale label to adjust the horizontal scale. The horizontal time base further adjusts all channel waveforms within the allowable range.

## 7.2. Horizontal Extension

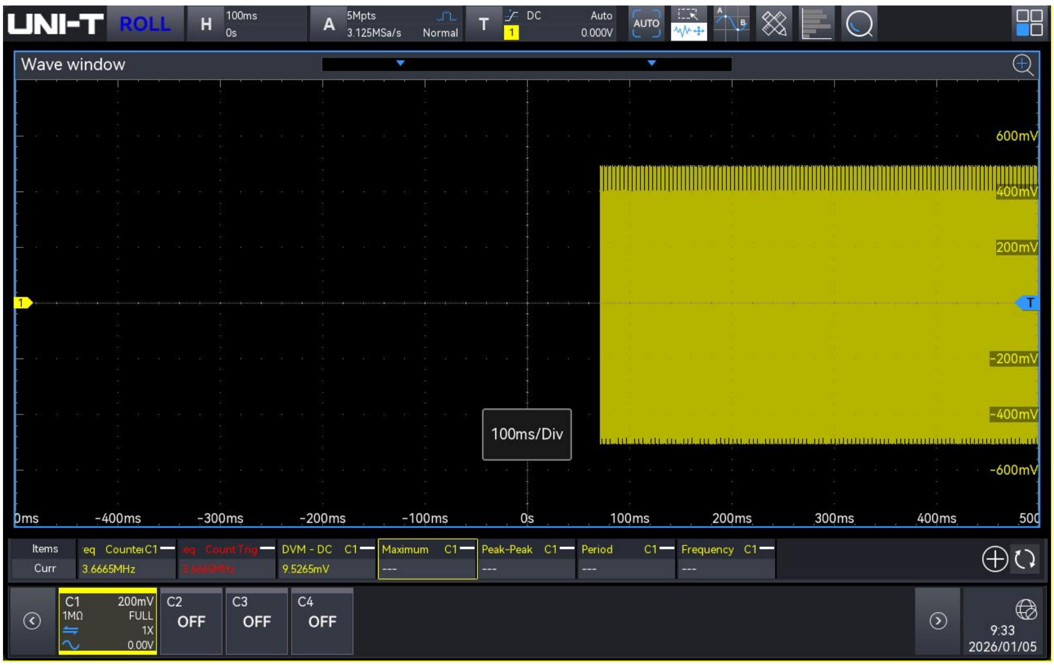
The horizontal extension refers to the reference position for horizontal extension and horizontal compression when adjusting the horizontal time base. In the Horizontal menu, select Horizontal Extension to set the reference position to Center, Left, Right, or Trigger Point. The default setting is Center.

- Center: When adjusting the horizontal time base, the waveform is horizontally expanded or compressed around the center of the screen.
- Left: When adjusting the horizontal time base, the waveform is horizontally expanded or compressed around the left edge of the screen.
- Right: When adjusting the horizontal time base, the waveform is horizontally expanded or compressed around the right edge of the screen.
- Trigger point: When adjusting the horizontal time base, the waveform is horizontally expanded or compressed around the trigger point.

## 7.3. Auto Roll Mode

Click Auto Roll Mode menu, set whether to enter SCAN or ROLL when the current time base is slower than the fastest roll time base. It toggles between Yes and No.

- Yes: When the time base is slower than the fastest roll time base, the oscilloscope will enter the ROLL mode. In this mode, the oscilloscope continuously draws the voltage-time tendency of the waveform on the screen. The waveform is refreshed from right to left and the latest waveform is drawn on the far right.




- No: When the time base is slower than the fastest roll time base, the oscilloscope will not enter the ROLL mode and will be in SCAN state. In the SCAN state, the oscilloscope enters the slow sweep mode. When using the slow sweep mode to observe the low-frequency signal, it is recommended that the channel coupling is set to DC. In this mode, the waveform starts from the trigger position, refreshing from left to right, with the latest waveform drawn on the far left.


## 7.4. Fastest Roll Time Base

The fastest scrolling/roll time base is a fixed setting used to determine when the oscilloscope enters ROLL or SCAN time base mode. This parameter is for indication only and cannot be adjusted by the user.

## 7.5. Horizontal Position

Tap the Horizontal Position input field to change its value. Set the horizontal center as the zero point, the waveform will move to the left if the horizontal position is greater than 0; the waveform will move to the right if the horizontal position is less than 0.

Set the horizontal position using the following steps.

- Double-click horizontal position input field to open the numeric keypad and directly enter the desired value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.
- Use the mouse to select the waveform, then drag it left or right to adjust the waveform's horizontal position.
- Use the mouse to select the waveform tag, then drag the tag left or right to adjust the waveform's horizontal position.

# 7.6. Window Expansion

The window expansion function is used to horizontally zoom in on a waveform for detailed observation, helping users analyze signal details more effectively. Follow the steps below to enable the zoom window.


- Tap the H (Horizontal Scale) label at the top of the screen to open the Horizontal menu, then Click Zoom (Time base extension) to turn it on or off.

  ON: Enables the time base extension.

  OFF: Disables the time base extension.

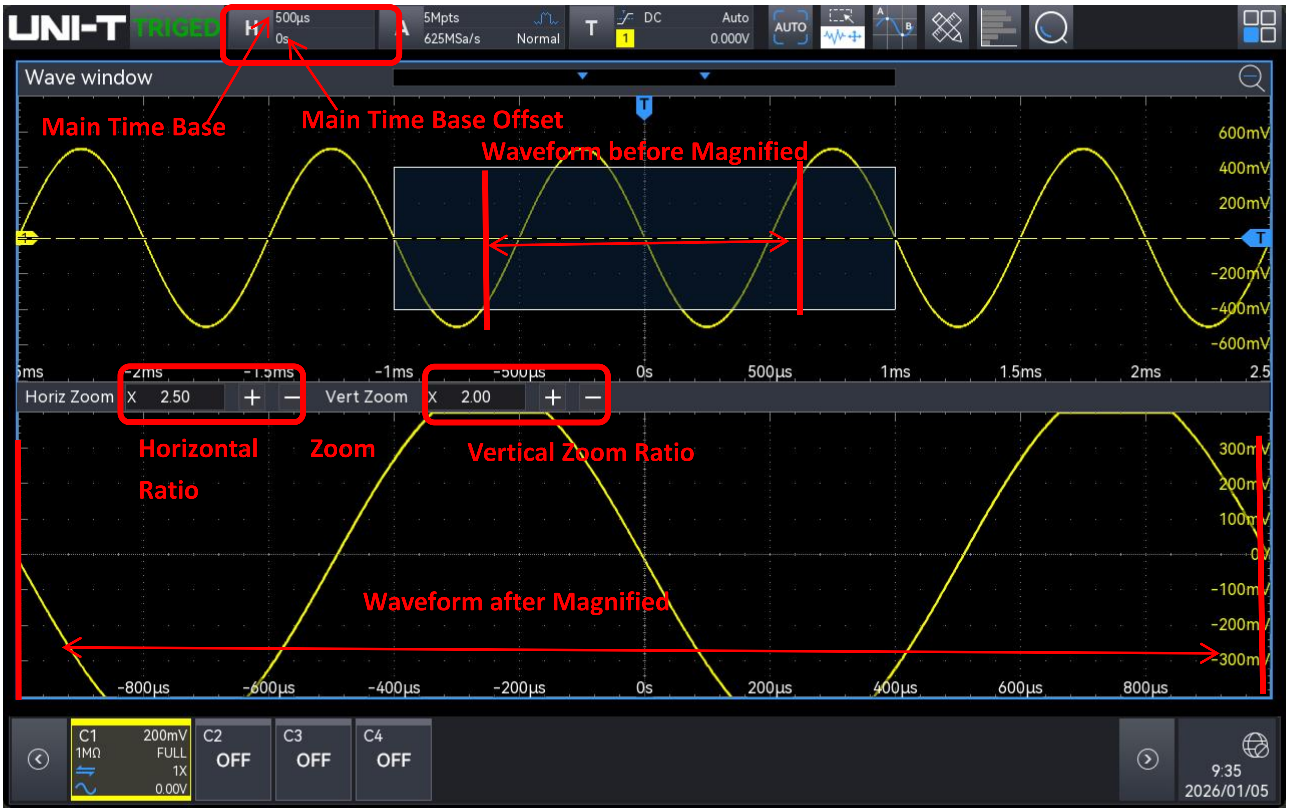
- Tap the icon in upper-right corner of the screen to quickly open the time base extension, tap the icon to exit the time base extension.

(1) Magnify Waveform

When the window expansion is enabled, the waveform displayed in the main window can be magnified horizontally and vertically.

Adjust the magnification using the following methods.

- Double-click Horizontal Zoom or Vertical Zoom input field on the screen to open the numeric keypad and directly enter the value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.
- Tap + or - next to the Horizontal Zoom and Vertical Zoom input field on the screen to increase or decrease the value.
- Drag the four borders of the zoom area in the main window to define the waveform region to be magnified.




(2) Waveform before Magnified

The waveform to be magnified is displayed as a shaded region in the upper part of the screen, can be adjusted using the horizontal and vertical zoom settings.

(3) Waveform after Magnified

The magnified waveform is displayed in the lower part of the screen. Window expansion provides higher resolution relative to the main time base, allowing waveform details to be observed more clearly.

**Note**: Window expansion is available only when the horizontal time base is faster than the fastest roll mode. If window expansion is enabled in Roll mode, the main time base is automatically set to 50 ms/div by default.

## 7.7. XY Mode

The waveform displayed in XY mode is also known as a Lissajous curve. XY mode supports cursor measurements, allowing for quick measurement of the phase difference between two signals.

(1) Time Base Format

a. YT: Displays the voltage value on the time base (horizontal scale).

b. XY: Displays a Lissajous curve, it can easily measure the phase difference between two signals with same frequency.

(2) Display: When XY mode is enabled, the channel waveform and XY curve are displayed in a split

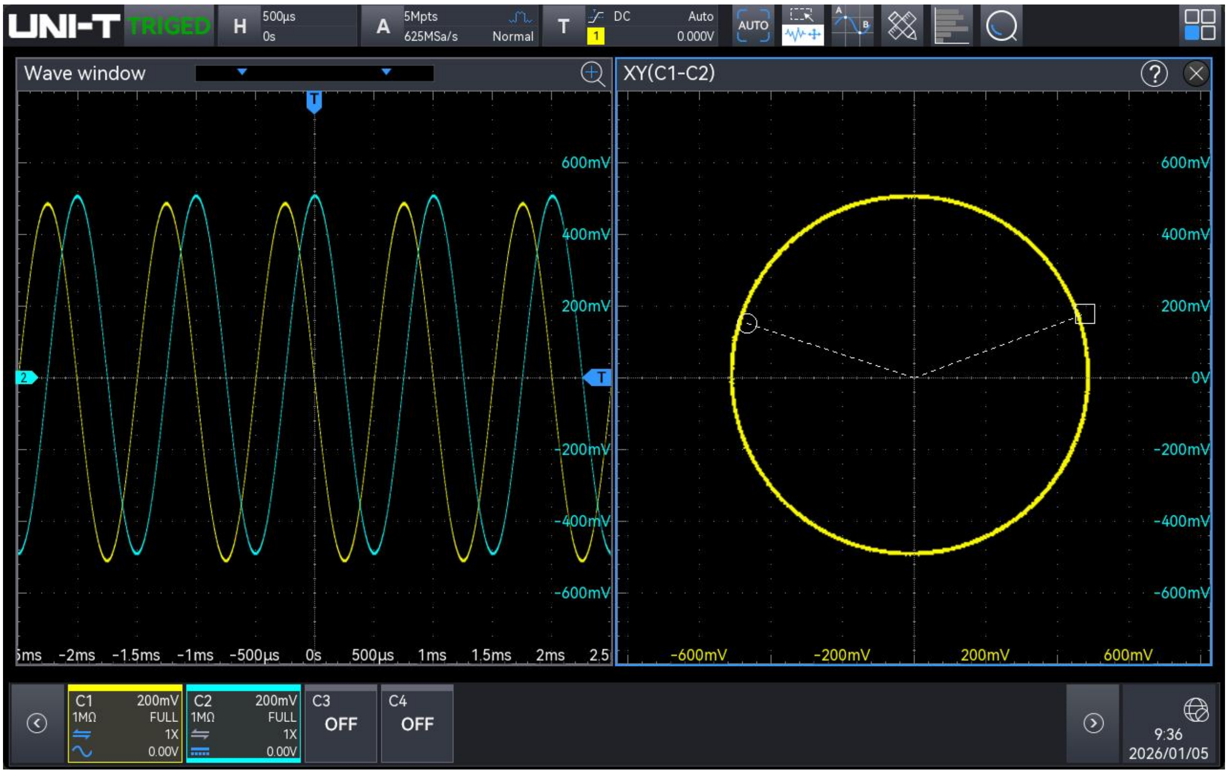
screen by default.

(3) Source X - Y:

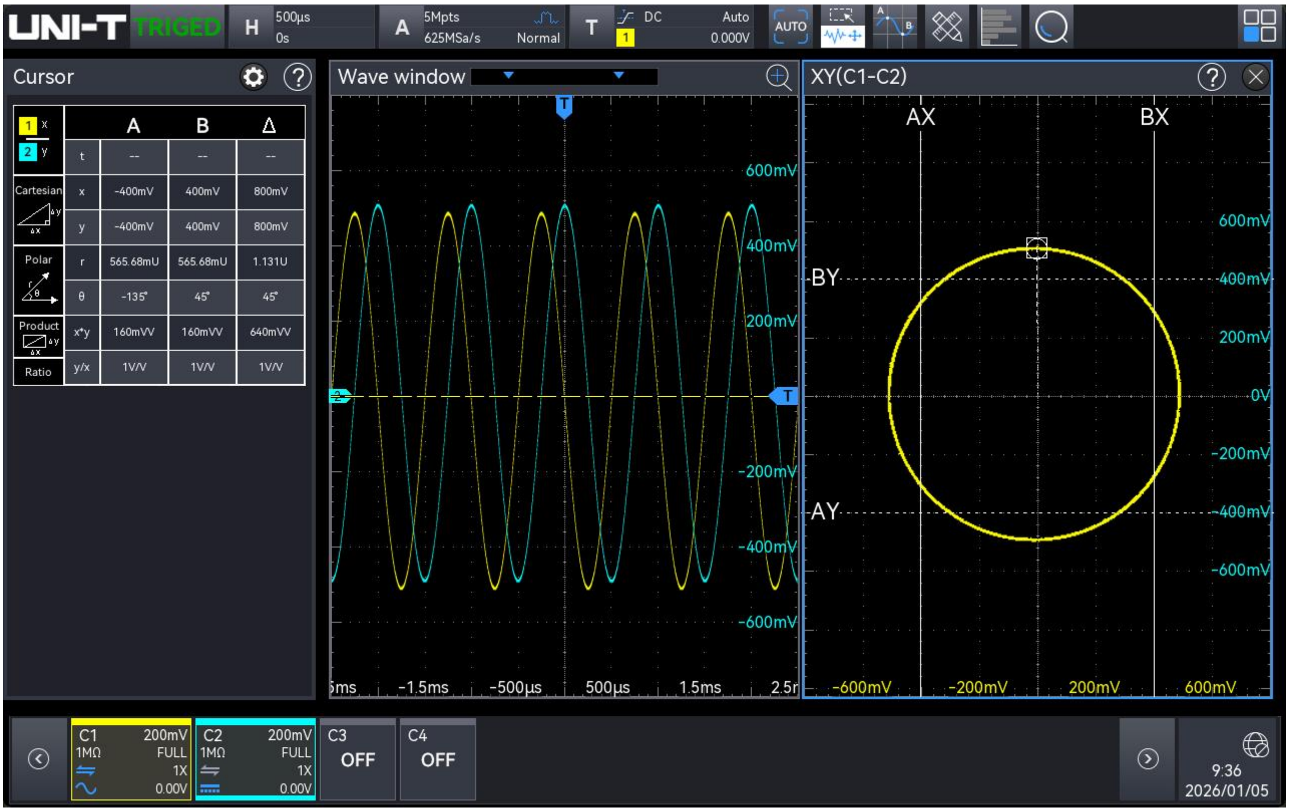
Set the waveform to generate a Lissajous curve, which can be selected from C1-C2, C1-C3, C1-C4, C2-C3, C2-C4, or C3-C4.

If X-Y is set to C1-C2, the C1 signal is applied to the horizontal axis (X), and the C2 signal is applied to the vertical axis (Y).

The waveform in XY mode is shown in the following figure.




In this state, set the menu to display in split screen, and press the cursor icon , as shown in the following figure.




When XY mode is enabled, the cursor supports both time and voltage measurements. For time measurements, the cursor appears in the waveform window and can be moved only within that window. For voltage measurement, the cursor is displayed in the XY window. Refer to the Cursor Measurement section for details on time and voltage cursor usage.

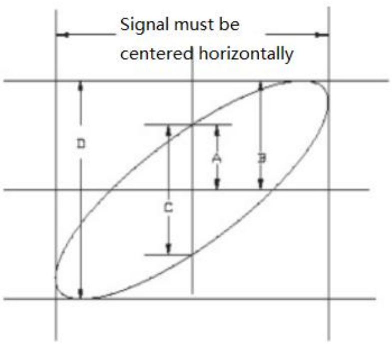
Cursor ①: Time, rectangular coordinates, polar coordinates, product, and ratio.

Cursor ②: Time, rectangular coordinate, polar coordinates, product, and proportion.

△: Delta (numerical difference between two cursors).

### Application of XY mode

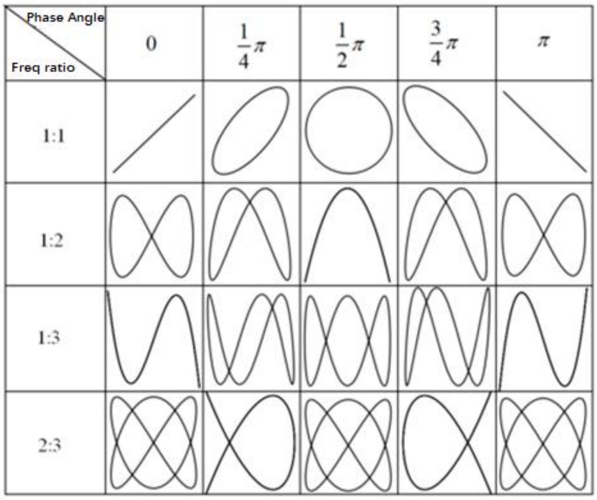
The phase difference between the two signals with the same frequency can be easily observed through Lissajous curve. The following figure explains a schematic diagram for observing the phase difference.




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 $\theta = \pm \arcsin(A/B)$ or $\theta = \pm$

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 - 3 π/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.



# 8. Triggering System

- SENT Triggering
- ARINC429 Triggering
- Zone Triggering

Triggering refers to setting a condition that must be met for the oscilloscope to capture and display a waveform. When the waveform meets the specified trigger condition, the oscilloscope captures the waveform and its adjacent part, displaying it on the screen. While the oscilloscope continuously captures waveforms during operation, only those that meet stable trigger conditions will be displayed.

The trigger ensures that each time base sweep or acquisition starts from a defined condition, synchronizing each scan with the acquisition. This synchronization ensures that the acquired waveforms overlap and provide a stable display.

Trigger settings determine when the oscilloscope acquires and displays data based on the characteristics of the input signal. For example, setting the trigger to activate on the rising edge of the analog channel 1 input signal enables the user to capture the desired waveform more efficiently. Understanding the signal under test is essential for quickly acquiring the desired waveform.

The UPO6000LP series offers several advanced trigger types, including various serial bus triggers. This chapter describes each trigger type in detail.

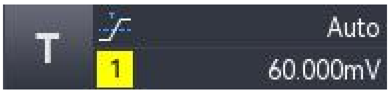
Advanced protocol triggered decoding supports the models and standards listed in the table below:

| Option Name | Description | Models | Standard/Option |
|---|---|---|---|
| Computer serial bus triggering and analysis | RS - 232/422/485/UART | UPO6000LP Series | Standard |
| Embedded serial bus triggering and analysis | $I^2C$, SPI | UPO6000LP Series | Standard |
| Automobile serial bus triggering and analysis | CAN, LIN | UPO6000LP Series | Option |
| Automobile serial bus triggering and analysis | CAN-FD | UPO6000LP Series | Option |
| Automobile serial bus triggering and analysis | FlexRay | UPO6000LP Series | Option |
| Automobile sensor bus triggering and analysis | SENT | UPO6000LP Series | Option |
| Audio serial bus triggering and analysis | Audio, LJ, RJ, TDM | UPO6000LP Series | Option |

| | | | |
|---|---|---|---|
| Aerospace serial bus triggering and analysis | MIL - STD - 1553， ARINC 429 | UPO6000LP Series | Option |
| Wireless communication trigger and analysis | Manchester | UPO6000LP Series | Option |

Access the Trigger menu using the following method.

- Tap the T (Trigger) label at the top to enter the Trigger Setting menu, as shown in the following figure.




## 8.1. Noun Explanation of Triggering System

(1) Trigger Source

A trigger source is the signal used to generate a trigger. The trigger can be derived from various sources, such as analog channels (C1–C4), an external trigger (EXT), or the AC line (mains electricity).

a. Analog channel: Select any of the analog input channels (C1–C4) on the front panel of the oscilloscope as the trigger signal.

b. External trigger: Select the EXT Trig input signal on the rear panel of the oscilloscope as the trigger signal. For example, connect an external clock signal to EXT Trig and set it as the trigger source. The EXT trigger level can be adjusted within the range of -5 V to +5 V.

c. AC Line: This trigger source is used to observe signals related to mains electricity, such as the relationship between lighting equipment and power supply equipment, to achieve stable synchronization.

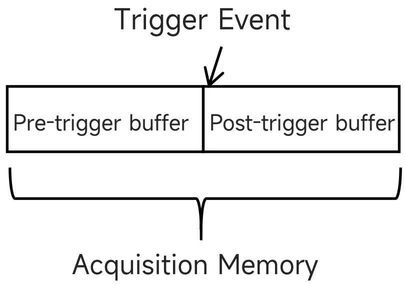
Tap the Trigger label at the top of the screen to open the Trigger menu, then click Source drop-down menu to select the trigger source.

**Note**: For an 8-channel oscilloscope, stable triggering is supported only when multiple trigger sources are selected simultaneously from either C1–C4 or C5–C8.

(2) Trigger Mode

The trigger mode determines how the oscilloscope behaves when a trigger condition occurs. The following briefly describes the trigger acquisition process using the pre-trigger buffer and the post-trigger buffer.

When the oscilloscope is operating, the pre-trigger buffer is filled first, continuously storing data while the system searches for a trigger event. The data is transferred to the pre-trigger buffer in a FIFO (First In, First Out) manner. Once a trigger event is detected, the pre-trigger buffer contains data captured before the trigger. The oscilloscope then fills the post-trigger buffer with the data captured after the trigger and displays the sampled waveform.




This oscilloscope provides three trigger modes: Auto, normal, and single. Set the trigger mode using the following steps.

■ Tap the Trigger label at the top of the screen to open the trigger menu and select the trigger mode.

a. Auto: In Auto mode, when no trigger signal is detected, the oscilloscope will automatically collect and display data. Once a trigger signal is generated, the oscilloscope automatically switches to sweep mode, ensuring signal synchronization. The auto mode is suitable for the following condition.

■ Checking DC signal or a signal with unknown electric feature.

**Note**: In Auto mode, the oscilloscope can operate at 100 ms/div or slower when no trigger signal is detected in ROLL mode.

b. Normal: The oscilloscope collects waveforms only when the trigger condition is met. If no trigger signal is detected, the oscilloscope will stop collecting data and enter a wait state. When the trigger condition is met, the oscilloscope refreshes the waveform on the screen; otherwise, it retains the last triggered waveform.

The normal mode is suitable for the following conditions.

■ Only collect the specified event appointed by the trigger setting.

■ A rare trigger event. The normal mode can prevent the oscilloscope from automatically triggering, so that the waveform can be displayed stably.

c. Single: In the single trigger mode, press the Single key on the front panel clears the waveform on the screen, and the oscilloscope enters a wait state. When a trigger is detected, the oscilloscope samples and displays the waveform, then enters the STOP state.

The single mode is suitable for the following conditions.

■ Capture a single event by accident or non-periodic signal, such as rising or falling waveforms.

■ A rare trigger event.

(3) Trigger Coupling

Trigger coupling determines which part of the signal will be transmitted to the trigger circuit. This setting is available only when the edge trigger is selected, and the trigger source is an analogue channel.

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Coupling drop-down menu to select the trigger coupling mode (Default: DC).

a. DC: Passes all DC and AC components of the signal.

b. AC: Blocks the DC component of the signal.

c. HF reject: Attenuates high-frequency components above 40 kHz.

d. LF reject: Attenuates low-frequency components below 40 kHz.

(4) Trigger Level

Trigger level is used to set the edge position of trigger point, and the trigger level is related to the trigger source.

- When the trigger source is C1-C4, double-click Trigger Level input field to open the numeric keyboard and set the trigger level. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. During this process, a trigger level line (the color is consistent with the channel color) and a trigger icon will appear on the screen, this line will move up and down according to the change of trigger level. After the adjustment stops, the dotted trigger level line disappears automatically after approximately 2 seconds. The current trigger level is displayed in the trigger label on the screen.

  For runt-amplitude trigger, ramp trigger and over-amplitude trigger, high level and low level must be set. If the trigger window is opened, tap on High or Low to open the virtual keyboard and set the trigger level. Two trigger level icons are displayed on the right.
- When the trigger source is AC Line, it has no trigger level.
- When the trigger source is EXT, double-click Trigger Level input field to open the virtual keyboard and set the trigger level. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The current trigger level is displayed in the trigger label on the screen. For this trigger source, only the trigger level change is displayed, but no trigger level line.

(5) Trigger Holdoff

Trigger holdoff is used to stabilize the display of complex or overlapping waveforms, such as pulse trains with multiple edges or events. It specifies the amount of time the oscilloscope waits before restarting the trigger circuit after a trigger condition is met. During the trigger holdoff, even if the trigger condition is met, the oscilloscope will not trigger until the end of holdoff time. For example, a set of pulse trains that require triggering on the first pulse of the train, then the holdoff time can be set to the width of the pulse train.


Tap the Trigger label at the top of the screen to open the Trigger menu, then Click Trigger Holdoff input field to open the numeric keypad and set the trigger holdoff time (until the waveform is stably triggered, the default is 80 ns). For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(6) Noise Rejection

Noise rejection is used to attenuate the high-frequency noise in a signal, to reduce the error trigger probability of the oscilloscope. Tap the Trigger label at the top of the screen to open the trigger menu, then Click Trigger Holdoff to turn it ON or OFF.

(7) Trigger Sensitivity

When noise is present in the signal, check the trigger sensitivity and adjust it as needed. Adjusting the trigger sensitivity changes the probability of noise triggering the system. The adjustable range is from 0% to 100%.

(8) 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 mode, press the Force key to collect the signal baseline and confirm that the acquisition is functioning normally.

(9) Pre-trigger/Delay Trigger

The sampled data acquired before trigger event/post-trigger event.

Trigger position is usually set at the horizontal center of the screen. The user can observe 5 grids of pre-trigger and delay information. The user can move the waveform horizontally to view more pre-trigger information. By observing the pre-trigger data, the waveform before generated can be observed. For example, capturing the glitch at the start of the circuit, observing and analyzing the pre-trigger data to find out the cause of the glitch.

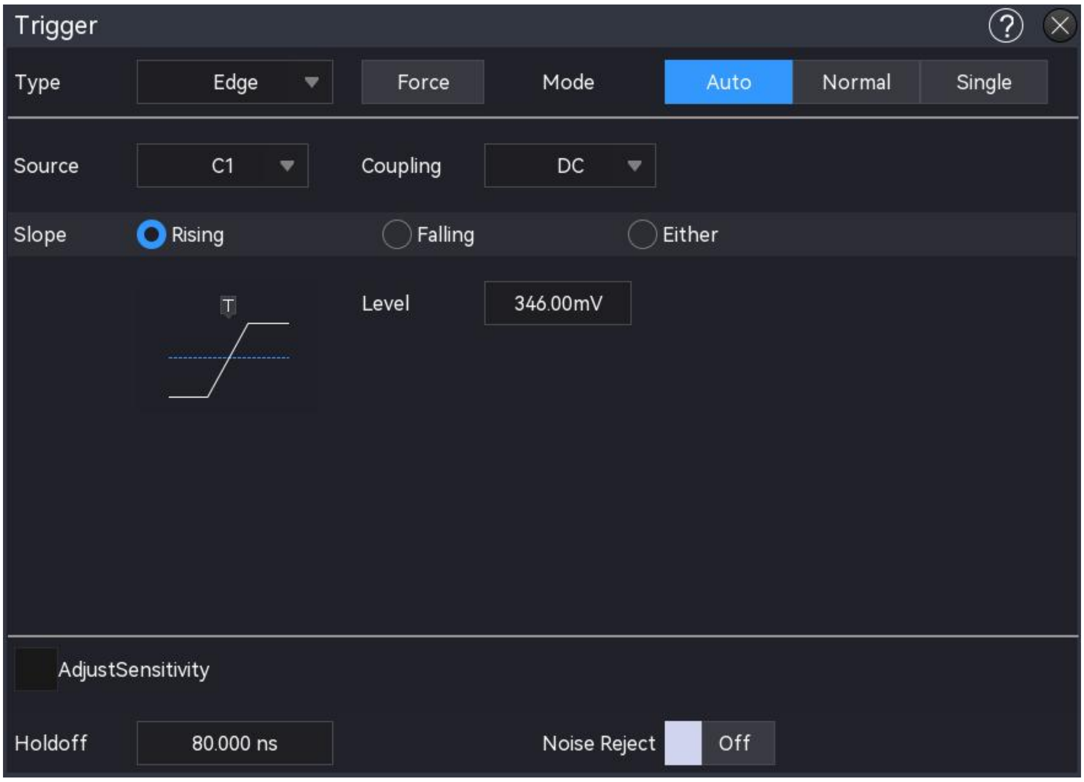
## 8.2. Edge Triggering

The edge trigger is identified by specifying the edge type (rising edge, falling edge, or rising/falling edge) and trigger level. The edge trigger menu sets the trigger source, trigger coupling, trigger mode, edge type, and trigger level. A stable waveform is generated when the trigger conditions are met.

Tap the Trigger label at the top of the screen to open the trigger menu to configure trigger settings.

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Edge as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4, AC Line, or EXT. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Trigger Coupling

Click Trigger Coupling to select DC, AC, LF reject, or HF reject. For more details on *Trigger Coupling,* refer to the section Noun Explanation of Triggering System.

(5) Edge Type

Select a signal and specify the edge on which to trigger. The current edge type is displayed in the trigger label at the top of the screen.

a. Rising edge: Triggers on the rising edge of the input signal.

b. Falling edge: Triggers on the falling edge of the input signal.

c. Either (Rising/Falling) edge: Triggers on both the rising and falling edges of the input signal.

(6) Trigger Level

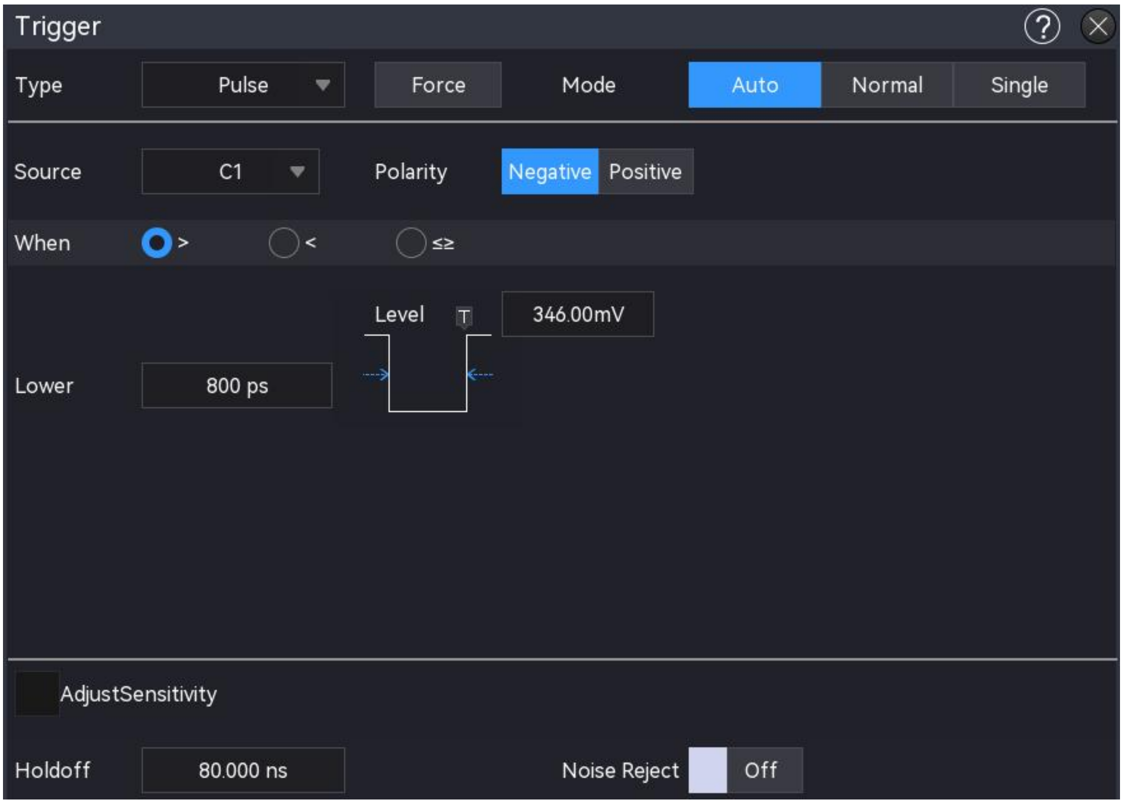
Tap Level, then use the numeric keypad to adjust the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

## 8.3. Pulse Width Triggering

Pulse width triggering sets the oscilloscope to trigger on the specified width when a positive or negative pulse meets the judgment conditions. The pulse width triggering menu sets the source, trigger condition, the upper/lower limit, polarity (positive/negative pulse width trigger), trigger type, trigger mode, and trigger level.

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Pulse Width as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4, AC Line, or EXT. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Trigger Condition

a. >: When the pulse width of the trigger signal (positive or negative pulse width) is greater than the set pulse width, the lower limit can be set.

b. <: When the pulse width of the trigger signal (positive or negative pulse width) is less than the set pulse width, the upper limit can be set.

c. ⩽ ⩾: When the pulse width of the trigger signal (positive or negative pulse width) is the same as the set pulse width or the pulse width of the trigger signal is triggered within the set range, the upper and lower limit can be set.

(5) Upper/Lower Limit

The set pulse width is compared to the pulse width of the trigger signal. The oscilloscope will be generated when the trigger condition is met. The adjustable range is from 800 ps to 4 s.

- When the trigger condition is set to > or <, click input field of the lower limit or upper limit to open the numeric keypad to set the lower or upper limit value.
- When the trigger condition is set to ⩽ ⩾, click lower or upper limit input field to open the numeric keypad to set the lower or upper limit value.

(6) Trigger Level

Tap Level, then use the numeric keypad to adjust the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

## 8.4. Video Triggering

The video signal includes the image and the time sequence information. It has multiple standards and formats. UPO6000LP can be Triggers on the field or line of the standard video signal, i.e. NTSC (National Television Standards Committee), PAL (Phase Alternating Line), and SECAM (Sequential Couleur A Memoire).

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Video as the trigger type.


(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Video Format

Click Video Format drop-down menu to select the desired format.

a. PAL: The frame frequency is 25 frames per second, the TV scan line is 625 lines, the odd field is in the front, and the even field is in the rear.

b. NTSC: The field frequency is 60 fields per second, and the frame frequency is 30 frames per second. The TV scan line is 525 lines. The even field is in the front, and the odd field is in the rear.

c. SECAM: The frame frequency is 25 frames per second, the TV scan line is 625 lines, interlaced scanning.

| **Video Format** | **Frame Frequency (Frame)** | **Sweep Type** | **TV Scan Line (Line)** |
|---|---|---|---|
| NTSC | 30 | Interlaced scanning | 525 |
| PAL/SECAM | 25 | Interlaced scanning | 625 |
| 525 p/60 | 60 | Progressive scanning | 525 |
| 625 p/50 | 50 | Progressive scanning | 625 |
| 720 p/24 | 24 | Progressive scanning | 750 |
| 720 p/25 | 25 | Progressive scanning | 750 |
| 720 p/30 | 30 | Progressive scanning | 750 |
| 720 p/50 Hz | 50 | Progressive scanning | 750 |
| 720 p/60 Hz | 60 | Progressive scanning | 750 |
| 1080 p/24 Hz | 24 | Progressive scanning | 1125 |

| | | | |
|---|---|---|---|
| 1080 p/25 Hz | 25 | Progressive scanning | 1125 |
| 1080 p/30 Hz | 30 | Progressive scanning | 1125 |
| 1080 i/25 Hz | 25 | Progressive scanning | 1125 |
| 1080 i/30 Hz | 30 | Progressive scanning | 1125 |
| 1080 Psf/24 | 24 | Progressive scanning | 1125 |

(5) Sync (Synchronization)

Synchronization can be set to even field, odd field, full field, all lines, or specified lines.

a. Even field: Triggers on the rising edge of the first sawtooth pulse in the even field. This option is available for NTSC, PAL/SECAM, 1080i/25, 1080i/30, and 1080Psf/24 video formats.

b. Odd field: Triggers on the rising edge of the first sawtooth pulse in the odd field. This option is available for NTSC, PAL/SECAM, 1080i/25, 1080i/30, and 1080Psf/24 video formats.

c. Full field: Triggers on the rising edge of the first pulse in the vertical synchronization interval.

d. All lines: Triggers and synchronizes on the first line of the video signal.

e. Specified lines: Triggers and synchronizes on the specified line. When this synchronization mode is selected, a line number can be specified. The configurable range of the line number depends on the selected video standard: 1–525 (NTSC), 1–625 (PAL/SECAM), 1–525 (525p), 1–625 (625p), 1–750 (720p), and 1–1125 (1080p/1080i).

**Note**: To observe detailed waveforms in the video signal, increase the memory depth.

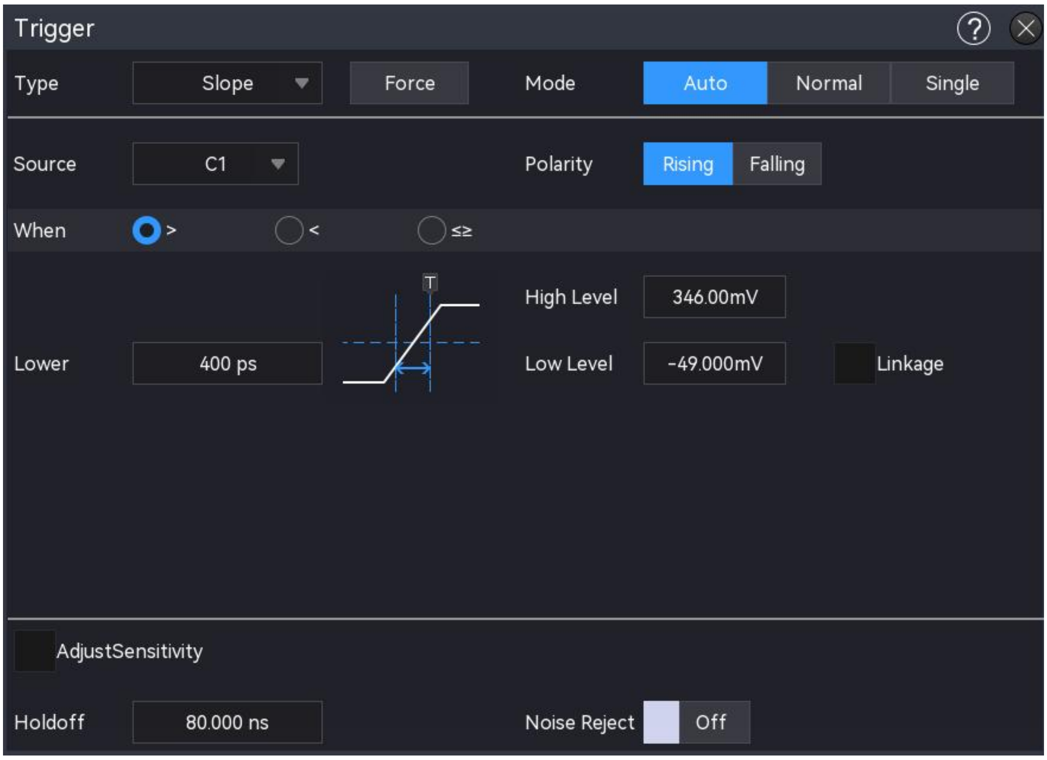
UPO6000LP series incorporates UNI-T proprietary digital 3D technology, featuring a multi-level greyscale display. This allows varying brightness levels to indicate the frequency of different signal components, helping users quickly assess signal quality and identify anomalies during debugging.

## 8.5. Slope Triggering

Slope triggering refers to triggering on the rising or falling edge of a signal within a specified time, making it suitable for observing sawtooth and triangular waves. The slope trigger menu sets the source, trigger mode, edge type (rising or falling), slope condition, upper and lower time limits, and high and low levels.

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Slope as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Edge Type

Select the input signal edge on which the oscilloscope triggers

a. Rising edge: Triggers on the rising edge of the input signal.

b. Falling edge: Triggers on the falling edge of the input signal.

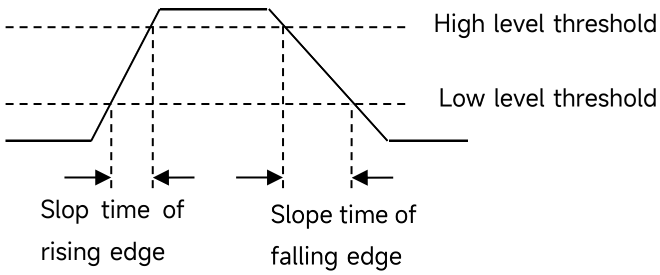
(5) Time Condition

a. > : When the slope time of the trigger signal is greater than the set slope time, the lower time limit can be set.

b. < : When the slope time of the trigger signal is less than the set slope time, the upper time limit can be set.

c. ⩽ ⩾: When the slope time of the trigger signal is the same as the set slope time or triggered within the set slope time, both the lower and upper time limits can be set.

**Note**: The slope time of the trigger signal refers to both the slope time of the rising edge and the slope time of the falling edge, as illustrated in the figure below.




(6) High/Low Level

The slope trigger requires both the high level and the low level to be set. The slope trigger can be generated stably only when all conditions are met.

Tap High Level or Low Level, then use the numeric keypad to adjust the corresponding level.

For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

Correlation: When the correlation is enabled, changing one level will cause the other level to change accordingly.

(7) Lower/Upper time limit

- When the trigger condition is set to > or <, click lower or upper limit input field to open the numeric keypad and set the corresponding time limit.
- When the trigger condition is set to ⩽ ⩾, click lower or upper limit input field to open the numeric keypad and set the corresponding time limit. The time range is from 00.4 ns to 1 s.

**Note**: The set slew rate is displayed at the bottom left of the screen.

The formula for calculating the slew rate:

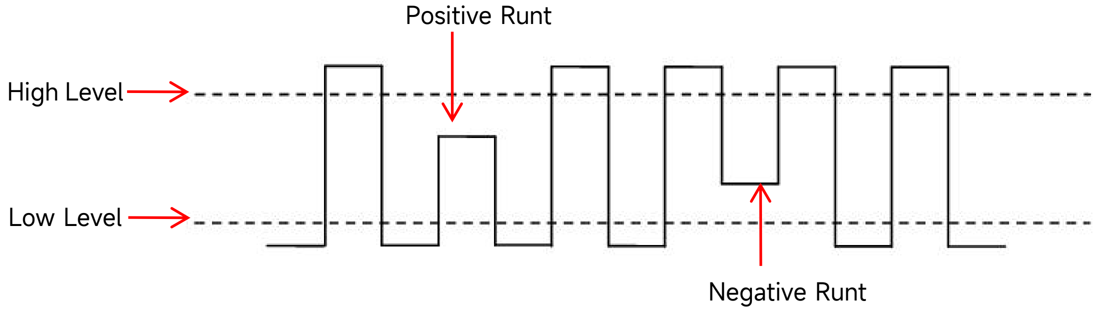
**(High level threshold - low level threshold) ÷ Time**

In this formula, Time refers to the set slope time.

## 8.6. Runt Pulse Triggering

The runt pulse triggering is used to trigger a pulse that has crossed one trigger level but not the other.

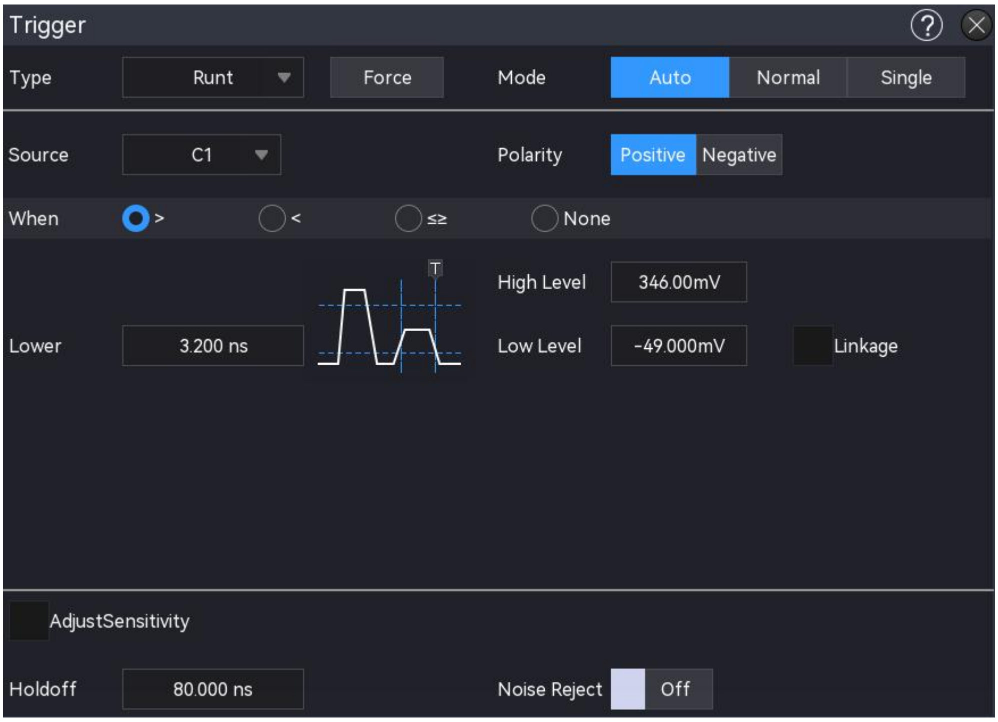
In this oscilloscope, the positive runt pulse is the pulse that crosses the lower limit of the trigger level but does not cross the upper limit of the trigger level; the negative runt pulse is the pulse that crosses the upper limit of the trigger level but does not cross the lower limit of the trigger level, as shown in the following figure.




The runt trigger menu sets the source, trigger mode, polarity (positive, negative), runt condition (irrelevance, <, >, ≤ ≥), the lower/upper time limit, and high/low level.

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Runt as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Polarity

a. Positive: Triggers on the positive runt pulse.

b. Negative: Triggers on the negative runt pulse.

(5) Runt Condition

a. >: When the runt pulse width is greater than the lower limit of the set pulse width, the lower time limit can be set.

b. <: When the runt pulse width is less than the upper limit of the set pulse width, the upper time limit can be set.

c. ≤ ≥: When the runt pulse width is equal to the lower or upper time limit, both the upper and lower time limits can be set.

d. Irrelevance: The runt pulse width is not compared to the time.

(6) High/Low Level

The runt plus trigger requires the high level and low level to be set. The runt plus trigger can only be stable generated when all conditions are met.

Tap High Level or Low Level, then use the numeric keypad to adjust the corresponding level.

For more details on *Trigger Level*, refer to the section Noun Explanation of Triggering System.

Correlation: When the correlation is enabled, changing one level will cause the other level to change accordingly.

(7) Lower/Upper time limit

- When the trigger condition is set to > or <, click lower or upper limit input field to open the numeric keypad and set the corresponding time limit.
- When the trigger condition is set to ≤ ≥, click lower or upper limit input field to open the numeric keypad and set the corresponding time limit. The time range is from 3.2 ns to 10 s.

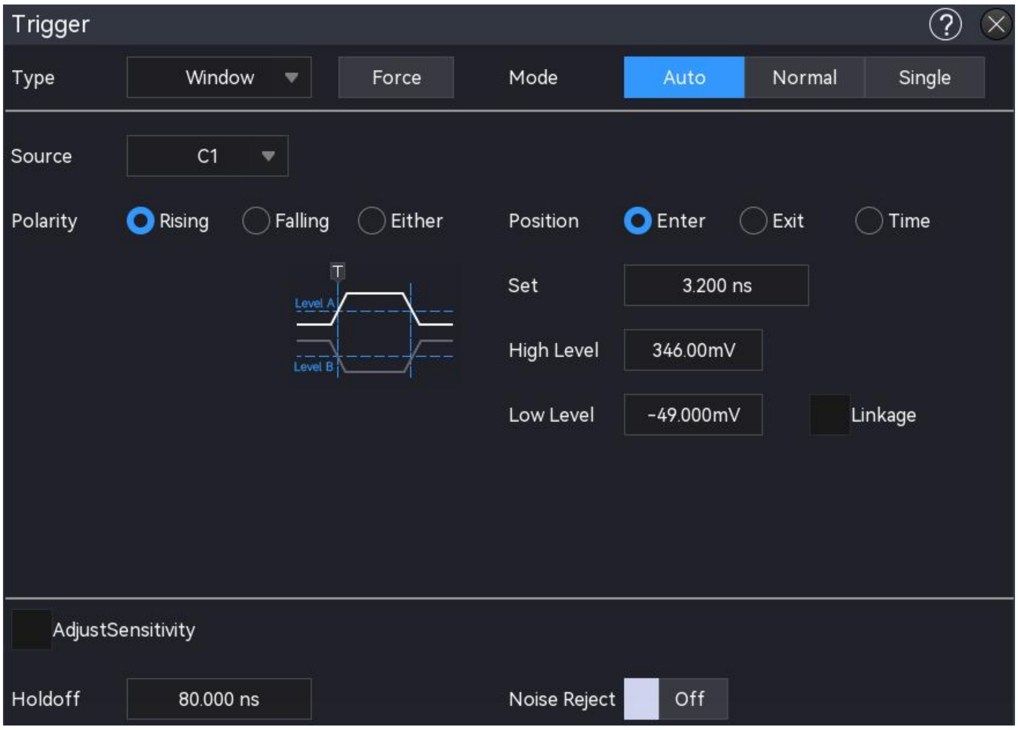
## 8.7. Window Triggering

The window triggering provides a high level and a low level. The oscilloscope triggers when the rising edge of the input signal crosses the high level or when the falling edge crosses the low level, as shown in the following figure.

The window triggering menu sets the source, trigger mode, edge type (rising edge, falling edge, or rising/falling), trigger position (enter, exit, or time), time, and high/low level.

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Window as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Edge Type

Set the input signal to trigger on the specified edge type, which can be rising, falling, or rising/falling edge. The current edge type is displayed in the trigger label at the top of the screen.

a. Rising edge: Triggers on the rising edge of the input signal when the voltage level exceeds the set high level.
b. Falling edge: Triggers on the falling edge of the input signal when the voltage level falls below the set low level.
c. Either (Rising/Falling): Triggers on the rising or falling edge of the input signal when the voltage level meets the set trigger level.

(5) Trigger Position

The trigger position can be set to enter, exit, or time. It is helpful to further confirm the trigger time.

a. Enter: Triggers when the input signal enters the specified trigger level.

b. Exit: Triggers when the input signal exits of the specified trigger level.

c. Time: Triggers when the accumulated hold time since the input signal entered the specified trigger level range is equal to the window time.

(6) Time

When the trigger position is set to Time, the time condition is applied and triggering occurs. Click Time input field to open the numeric keypad and set the window time. The window time can be set from 3.2 ns to 10 s.

(7) High/Low Level

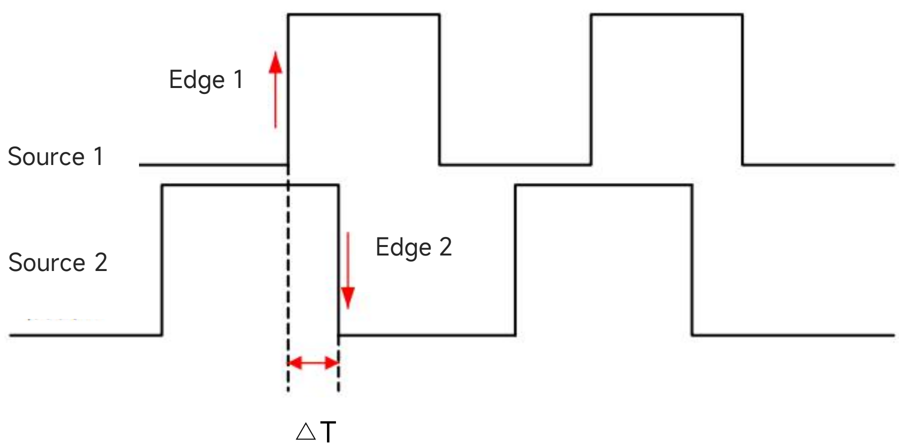
Window triggering requires the high level and low level to be set. It can trigger stably only when all conditions are met.

Tap High Level or Low Level, then use the numeric keypad to adjust the corresponding level. For more details on *Trigger Level*, refer to the section Noun Explanation of Triggering System.

Correlation: When the correlation is enabled, changing one level will cause the other level to change accordingly.

## 8.8. Delay Triggering

Delay triggering requires configuring both Trigger Source 1 and Trigger Source 2. When the time difference (△T) between the edge set by Source 1 (Edge 1) and the edge set by Source 2 (Edge 2) meets the set time limit, the oscilloscope will be triggered, as shown in the following figure.




Edge 1 is set as the rising edge. Edge 2 is set as the falling edge. △T is the area marked in red.

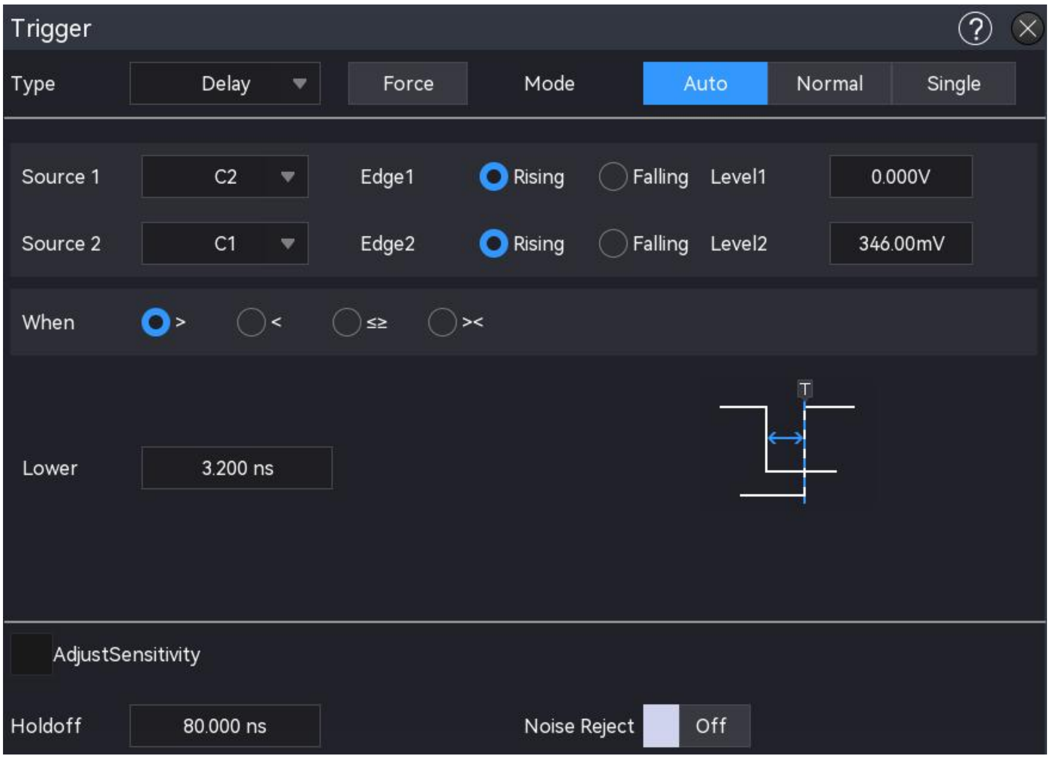
**Note**:

- Edge 1 and edge 2 must be adjacent edges.
- Only the channel that has a connected signal and to be the trigger source can be triggered

stably.

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Delay as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Source 1

Click Source 1 to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Edge 1

Click Edge 1 to set the delay trigger for Source 1, it can be set to rising or falling edge.

(5) Source 2

Click Source 2 to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(6) Edge 2

Click Edge 2 to set the delay trigger for Source 2, it can be set to rising or falling edge.

(7) Delay Condition

a. ＞: Triggers when the time difference (△T) between the specified edges of Source 1 and Source 2 is greater than the set lower time limit, and the lower time limit can be set.

b. ＜: Triggers when the time difference (△T) between the specified edges of Source 1 and Source 2 is greater than the set upper time limit, and the upper time limit can be set.

c. ⩽ ⩾: Triggers when the time difference (△T) between the specified edges of Source 1 and Source 2 is greater than or equal to the set lower time limit and less than or equal to the set upper time limit. The upper/lower time limits can be set.

d. ＞ ＜: Triggers when the time difference (△T) between the specified edges of Source 1 and Source 2 is less than the set lower time limit or greater than the set upper time limit. The upper/lower time limits can be set.

(8) Lower/Upper time limit

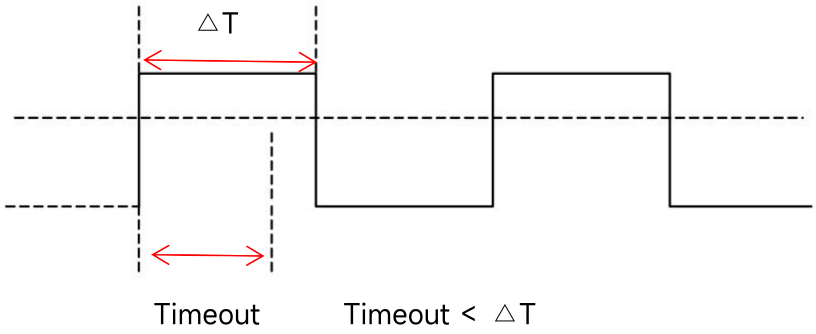
- When the trigger condition is set to > or <, click lower or upper limit input field to open the numeric keypad and set the corresponding time limit.
- When the trigger condition is set to ⩽ ⩾, click lower or upper limit input field to open the numeric keypad and set the corresponding time limit. The time range is from 3.2 ns to 10 s.

(9) Level 1, Level 2

The delay trigger requires both trigger Source 1 and trigger Source 2 to be set. It can trigger stably only when all conditions are met. Tap Level 1 or Level 2, then use the numeric keypad to adjust the corresponding level value. For more details on *Trigger Level*, refer to the section Noun Explanation of Triggering System.

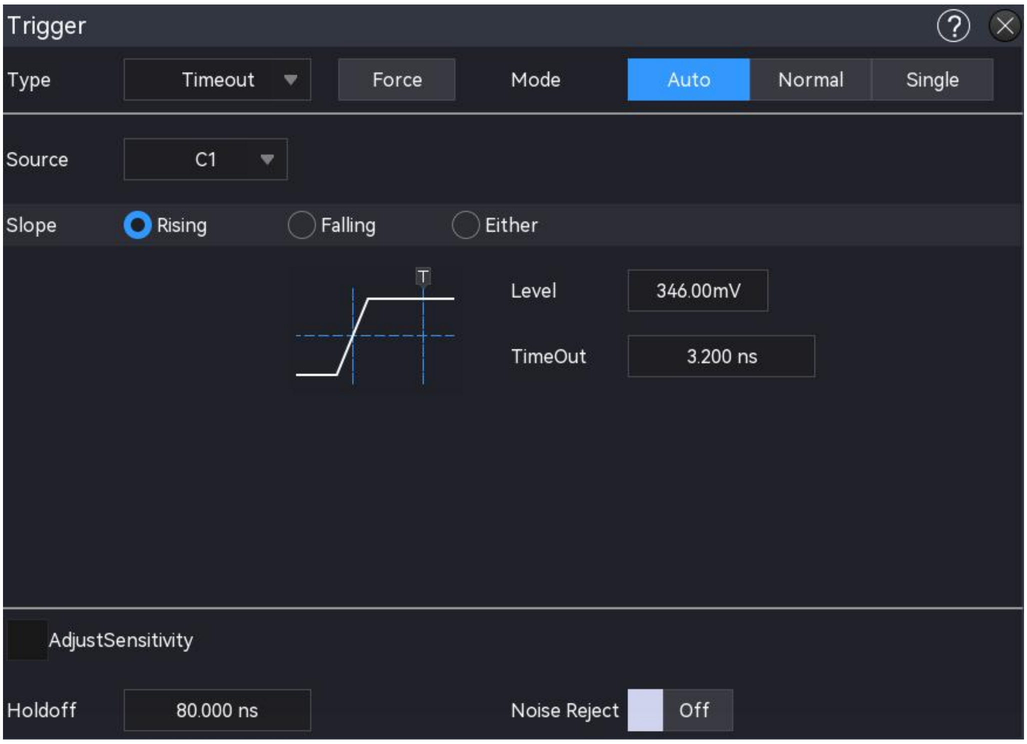
## 8.9. Timeout Triggering

The oscilloscope will be triggered when the time interval (△T) from the rising edge (or falling edge) of the input signal across the trigger level and to the adjacent falling edge (rising edge) across the trigger level is greater than the set timeout time, as shown in the following figure.




(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Timeout as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Edge Type

Select the edge type on which the input signal will trigger. The current edge type is displayed

in the trigger label at the top of the screen.

a. Rising edge: Counts the time when the rising edge of the input signal crosses the trigger level.

b. Falling edge: Counts the time when the falling edge of the input signal crosses the trigger level.

c. Either (Rising/Falling): Counts the time when either the rising or falling edge of the input signal crosses the trigger level.

(5) Timeout

Timeout is used to set the maximum hold time after the input signal across the trigger level. The oscilloscope will be generated when timeout < △T. Click Timeout input field to open the numeric keypad and set the timeout.

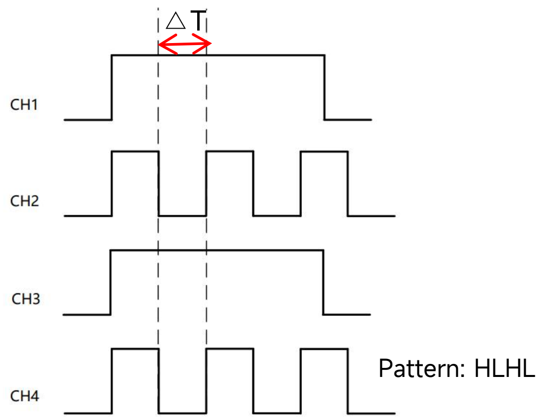
The timeout range can be set from 3.2 ns to 10 s.

(6) Trigger Level

Tap Level, then use the numeric keypad to adjust the level value. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

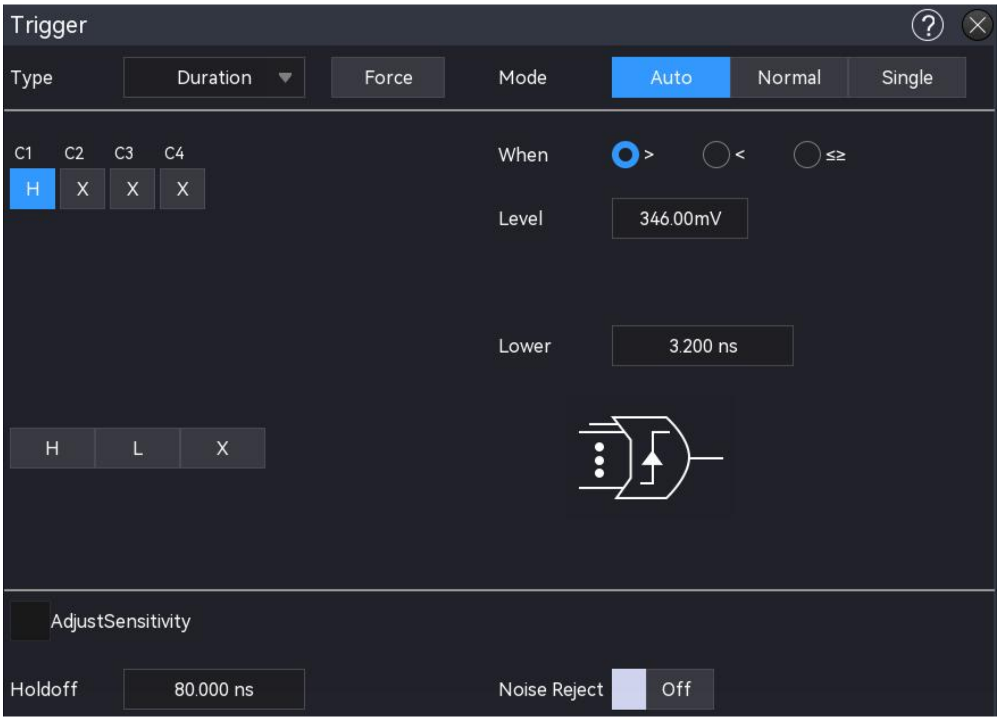
## 8.10. Duration Triggering

When the duration triggering is selected, the oscilloscope identifies the trigger condition by searching for the duration of the specified pattern. This pattern is a logical "AND" combination of channels. Each channel can be set to H (high), L (low), or X (ignore). The oscilloscope trigger when the duration (△T) of this pattern meets the set time, as shown in the following figure.




(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Duration as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Pattern Setting

The pattern can be set to H, L, or X, as shown in the figure above.

a. H: Sets the pattern for the selected channel to H, i.e. the voltage level is higher than the trigger level of the channel.

b. L: Sets the pattern for the selected channel to L, i.e. the voltage level is lower than the trigger level of the channel.

c. X: Sets the pattern for the selected channel to X, i.e. the channel is not part of the pattern, the oscilloscope will not be triggered if all channels in the pattern are set to X.

(5) Trigger Condition

a. >: Triggers when the duration is greater than the set lower time limit, and the lower time limit can be set.

b. <: Triggers when the duration is less than the set upper time limit, and the upper time limit can be set.

c. ≤ ≥: Triggers when the duration is less than or equal to the set upper time limit and greater than or equal to the lower time limit. The upper/lower time limits can be set.

(6) Lower/Upper time limit

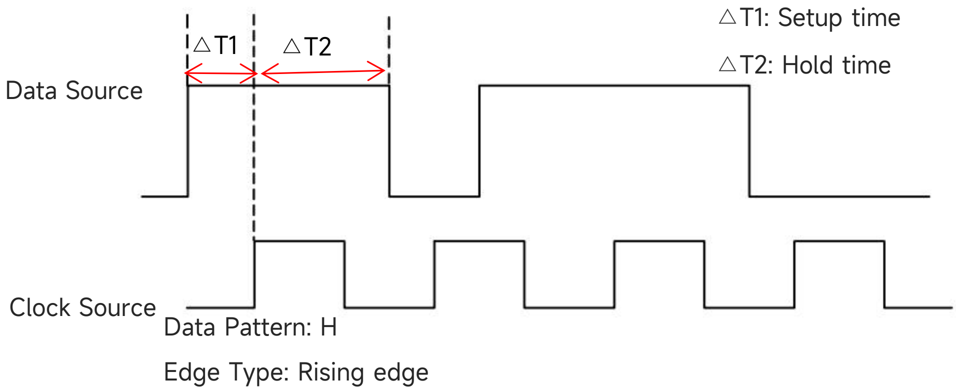
- When the trigger condition is set to > or <, click lower or upper limit input field to open the numeric keypad and set the corresponding time limit.
- When the trigger condition is set to ≤ ≥, click lower or upper limit input field to open the numeric keypad and set the corresponding time limit. The time range is from 3.2 ns to 10 s.

(7) Trigger Level

Tap Level, then use the numeric keypad to adjust the level value. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

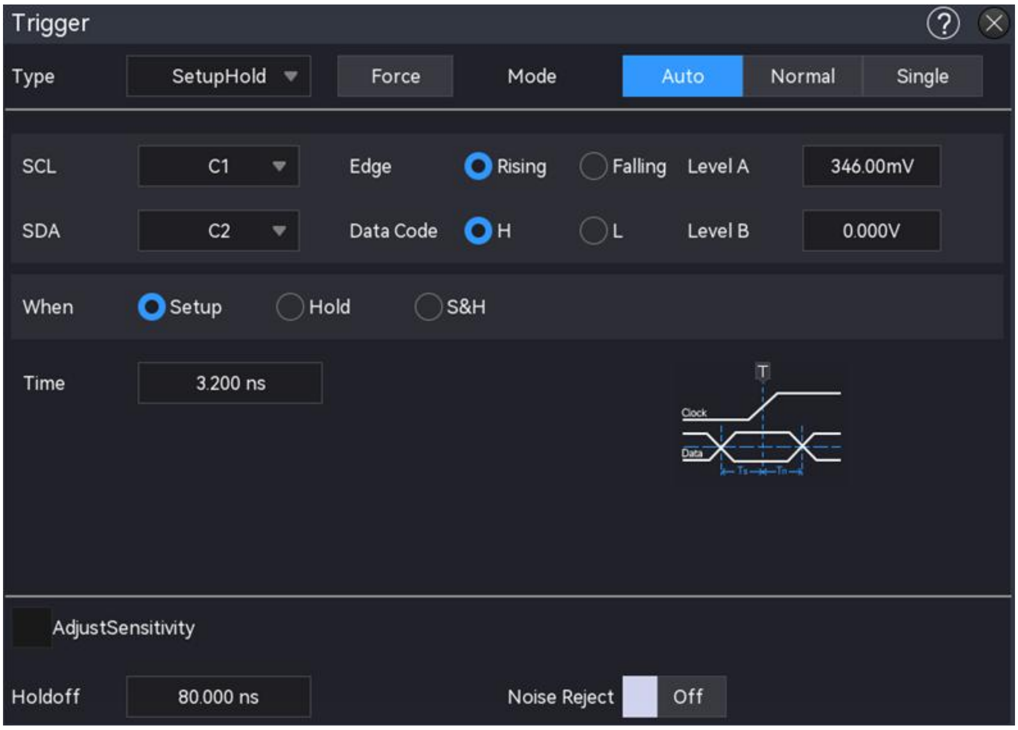
## 8.11. Setup & Hold Triggering

The setup/hold triggering requires the data signal line and clock signal line to be set. 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, as shown in the following figure. The oscilloscope triggers when the setup time or the hold time is less than the pre-set time. It is mainly used to locate and find the error pattern, and quickly find the signal that cannot meet setup and hold time.




(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Setup & Hold as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Data Source

Click SDA to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Data Type

Select the effective pattern for the data signal. It can be set to H or L.

a. H: Sets the effective pattern for the data signal to a high level.

b. L: Sets the effective pattern for the data signal to a low level.

(5) Clock Source

Click Clock Source to select C1-C4 as the clock source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(6) Edge Type

a. Rising edge: Sets the clock edge to the rising edge.

b. Falling edge: Sets the clock edge to the falling edge.

(7) Trigger Condition

a. Setup: Triggers when the setup time is less than the specified setup time.

b. Hold: Triggers when the hold time is less than the specified hold time.

c. Setup & Hold: Triggers when the setup and hold time is less than the specified time.

(8) Time

The setup and hold time △**T** is compared to the set time, and the oscilloscope generates a trigger when the condition is met. Click Time input field to open the numeric keypad and set the time.

The time range is from 3.2 ns to 10 s.

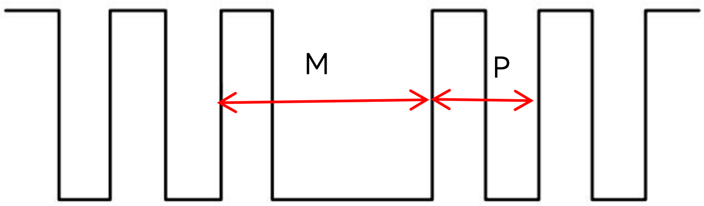
(9) Data Level, Clock Level

The setup & hold trigger requires both the data level and clock level to be set. It can trigger stably only when all conditions are met.

Tap Data Level or Clock Level, then use the numeric keypad to adjust the corresponding level value. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

## 8.12. Nth Edge Triggering

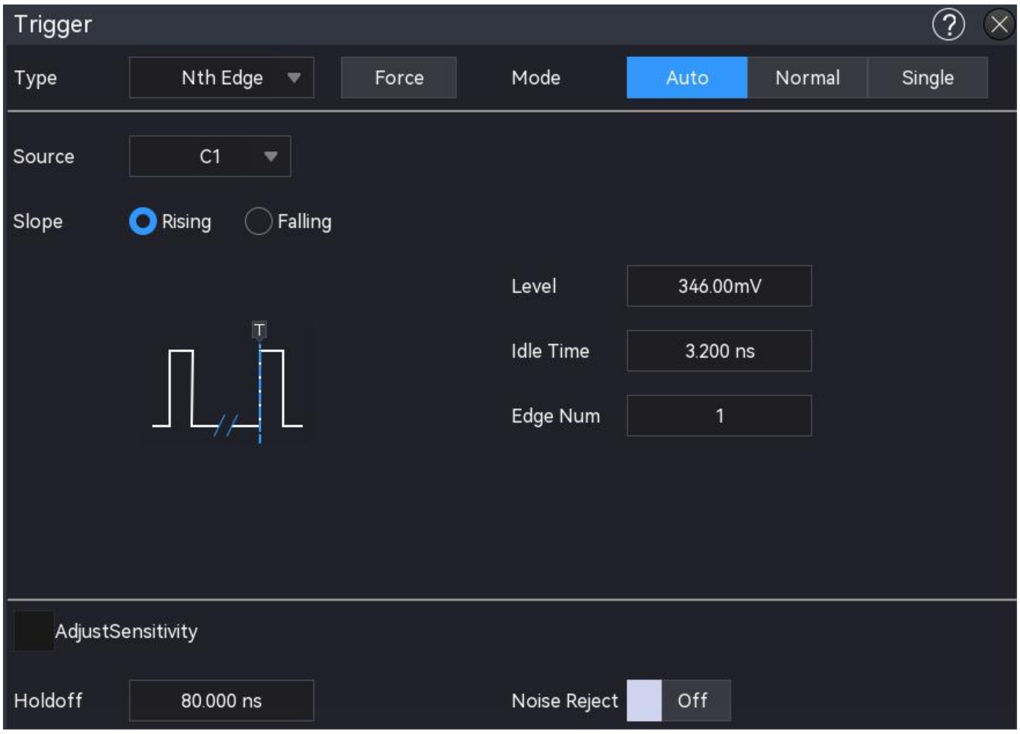
The Nth edge triggering refers to be triggers on the Nth edge after the specified idle time. For example, as shown in the following figure, if the oscilloscope is set to trigger on the 2nd rising edge following the specified idle time (the time between two adjacent rising edges), the idle time should be set such that P< idle time <M. Here, M is the time between the 1st rising edge and the next rising edge, and P is the maximum allowable time for counting the rising edges, as shown in the following figure.




P < Idle Time < M

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Nth Edge as the trigger type.




(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Edge Type

Select an input signal to trigger on which edge.

a. Rising edge: Triggers on the rising edge of the input signal.

b. Falling edge: Triggers on the falling edge of the input signal.

(5) Idle Time

The idle time is compared to the pulse time. The oscilloscope triggers when the condition is met.

Click Idle Time input field to open the numeric keypad and set the idle time.

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

(6) Edge count

The edge count represents the Nth edge value. Click Edge count input field to open the numeric keypad and set the edge count. The edge count range can be set from 1 to 65535.

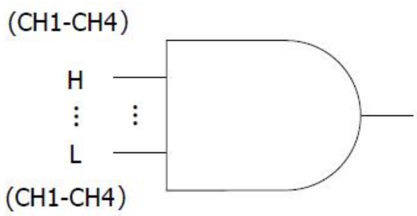
(7) Trigger Level

Tap Level, then use the numeric keypad to adjust the trigger level value. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

## 8.13. Pattern Triggering

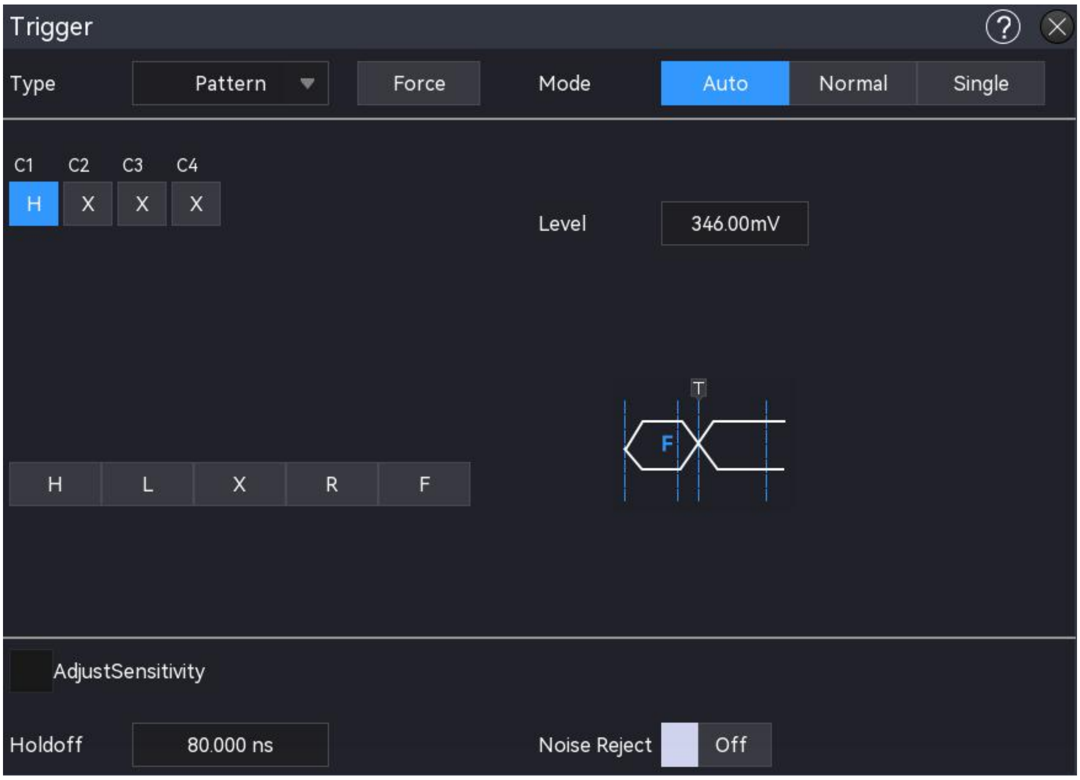
The pattern triggering identifies the trigger condition by searching for the specified patterns. This pattern is a logical "AND" combination of channels. Each channel can be set to H (high), L (low), or X (ignore).

A rising or falling edge (only one edge can be specified) can be specified for one channel included in the pattern. When an edge is specified, the oscilloscope triggers on the specified edge if the patterns set for the other channels are true (that is, the actual channel states match the set pattern). If no edge is specified, the oscilloscope triggers on the last edge that makes the pattern true. If all the channels in the pattern are set to X, the oscilloscope ddooes not trigger




(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Pattern as the trigger type.



(2) Trigger Mode

Set the trigger mode to auto, normal, or single. For more details on *Trigger Mode*, refer to the section Noun Explanation of Triggering System.

(3) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(4) Pattern Setting

The pattern can be set to H, L, X, rising edge, or falling edge. The pattern for each channel is displayed at the bottom of the screen.

a. H: Sets the pattern for the selected channel to H, i.e. the voltage level is higher than the trigger level of the channel.
b. L: Sets the pattern for the selected channel to L, i.e. the voltage level is lower than the trigger level of the channel.
c. X: Sets the pattern for the selected channel to X, i.e. the channel is not part of the code pattern, the oscilloscope will not be triggered if all channel in the code pattern is set to X.
d. Rising edge: Sets the pattern for the selected channel to the rising edge.
e. Falling edge: Sets the pattern for the selected channel to the falling edge.

(5) Trigger Level

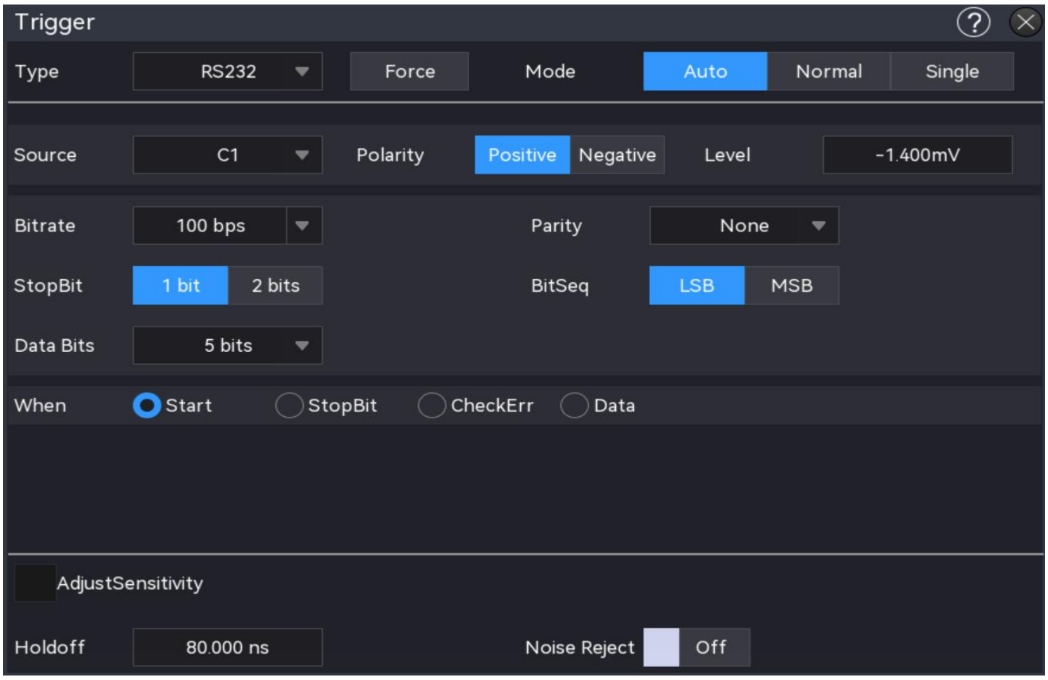
Tap Level, then use the numeric keypad to adjust the level value. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

## 8.14. RS232 Triggering

RS232 bus triggering is a serial communication method for transferring data between computers or between a computer and a terminal.

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select RS232 as the trigger type.




(2) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(3) Trigger Level

Double-click Level input field to open the numeric keypad and set the trigger level.

When the trigger level is changed, a dotted line appears on the screen to indicate the current trigger level. After the adjustment stops, the dotted trigger level line disappears automatically after approximately 2 seconds.

(4) Polarity

a. Negative: The reversed polarity of logic level, i.e. the high level is 0 and the low level is 1.
b. Positive: The normal polarity of logic level, i.e. the high level is 1 and the low level is 0.

(5) Parity Check

Set the parity check for data transmission. Click Parity to select none, even, or odd.

(6) Data Bit Width

Set the data bit width for RS232 signal. Click Data Bits to select 5 bits, 6 bits, 7 bits, or 8 bits.

(7) Bit Sequence

Set the data bit sequence for RS232 signal, click Bit Sequence to select MSB or LSB

a. MSB: The Most Significant Bit, i.e., the bit with the highest value in a sequence, transmitted first.

b. LSB: The Least Significant Bit, i.e., the bit with the lowest value in a sequence, transmitted last.

(8) Stop Bit

Set the stop bit for each data. Click Stop Bit to select 1 bit or 2 bits.

(9) Bitrate

When RS232 communication uses asynchronous transmission communication, no accompanying clock signal is present during data transmission process. To correctly determine the data bits, both communication parties must agree on the bit rate. The bit rate is generally defined as the number of bits transmitted in one second. For example, 9600 bps means that 9600 bits are transmitted in one second. The bitrate is not directly equal to the effective data transmission rate. Note that the start bit, data bit, parity bit, and stop bit are all included in the total bit count. The oscilloscope sets the bitrate according to the bit sampling.

The bitrate can be set to 2400 bps, 4800 bps, 9600 bps, 19200 bps, 38400 bps, 57600 bps, 115200 bps, or custom. Open the numeric keypad to set the custom bitrate.

It is recommended to configure the bitrate according to the RS232 communication hardware and software. Due to the characteristics of this transmission protocol, RS232 protocol is typically used for short distance (less than 20 m), and low speed (less than 1 Mbps) transmission occasions. Communication beyond this range is susceptible to interference and may become unreliable.

(10) Trigger Condition

a. Start frame: Triggers on the start bit of RS232. When transmitting a single string or repeatedly transmitting the same string, this trigger provides a stable signal waveform. If the transmitted data changes, the corresponding waveform also changes.

b. Stop bit: Triggers on the first stop bit in the RS232 protocol. This trigger is applied automatically, regardless of whether the Device Under Test (DUT) uses 1 or 2 stop bits. There is no need to specify the number of stop bits used by the DUT.

c. Parity error: When RS232 has the parity bit, set the parity bit to 0 or 1 according to the parity check method.

The parity checking rules are as follows:

- Odd parity check: The transmission is considered correct if the number of 1s in both the data bits and the parity bit is odd.
- Even parity check: The transmission is considered correct if the total number of 1s in both the data bits and the parity bit is even.

With this setting, the user can quickly locate and find the transmission process of parity error during the RS232 communication. It is useful for analyzing the fault.

d. Data: Triggers when data acquired by the oscilloscope matches the custom 2 bits in hexadecimal. With this option, the user can quickly locate the transmission signals containing specific data of interest.

When the data is selected, the data menu can be configured.

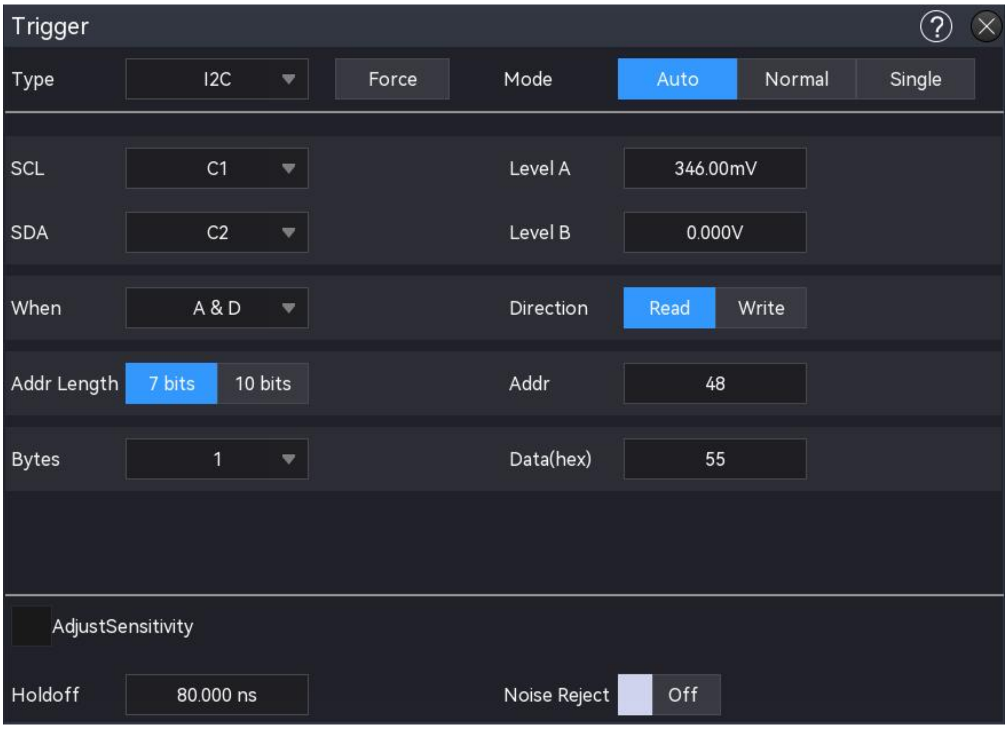
- Data: The data depends on the frame length. Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 00 to FF.

## 8.15. I²C Triggering

I²C bus triggering is a two-wire serial bus and used to connect the microcontroller and peripheral device. It's widely applied in micro-electronics area.

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select I2C as the trigger type.




(2) Source Setting

Set both the clock source and the data source. The oscilloscope will only trigger stably if the selected channel has a connected signal and is set as the trigger source.

a. Clock Source

Click SCL to select C1-C4 as the clock source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed

in the trigger label at the top of the screen.

b. Data Source

Click SDA to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

(3) Trigger Level

Tap Level A or Level B, then double-click corresponding input field to open the numeric keypad and set the trigger level.

(4) Operating Direction

Click Direction to select Read or Write.

a. Write: Triggers when $I^2C$ protocol Read/Write bit is set to Write.

b. Read: Triggers when $I^2C$ protocol Read/Write bit is set to Read.

(5) Trigger Condition

a. Start: Triggers at the start time.

b. Restart: Triggers on the restart time, i.e. a start signal appears again after a start signal has appeared, but a stop has not appeared yet.

c. Stop: Triggers on the stop bit, i.e. SDA signal goes from low to high when SCL is at a high level.

d. Loss confirmed: In $I^2C$ protocol, after every 8 bits of data are transmitted, the data receiver must send an acknowledgement signal. As shown in the figure above, the ACK bit occurs when SCL is high and SDA is low. A loss trigger occurs when both SCL and SDA are high during the ACK bit period.

e. Address: The oscilloscope triggers when the communication address matches the user setting address. This helps the user quickly locate address transmissions.

When the address length or address is selected, the corresponding menus can be configured.

- Address length: Sets the address bit width of $I^2C$ signal, click Address Length to select 7 bits or 10 bits.
- Address: Sets the trigger address, double-click Address input field to open the numeric keypad and set the address. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 00 to FF and from 000 to 3FF.

f. Data: Triggers when the data acquired by $I^2C$ matches the custom data. This function helps users quickly locate transmission signals containing the specified data of interest.

When the byte length or data is selected, the corresponding menus can be configured.

- Byte length: Click Byte Length input field to set the byte length for the specified data.

The byte length range can be set from 1 to 5.

- Data: The data depends on the frame length. Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 00 to FFFFFFFFFF (10 Fs).

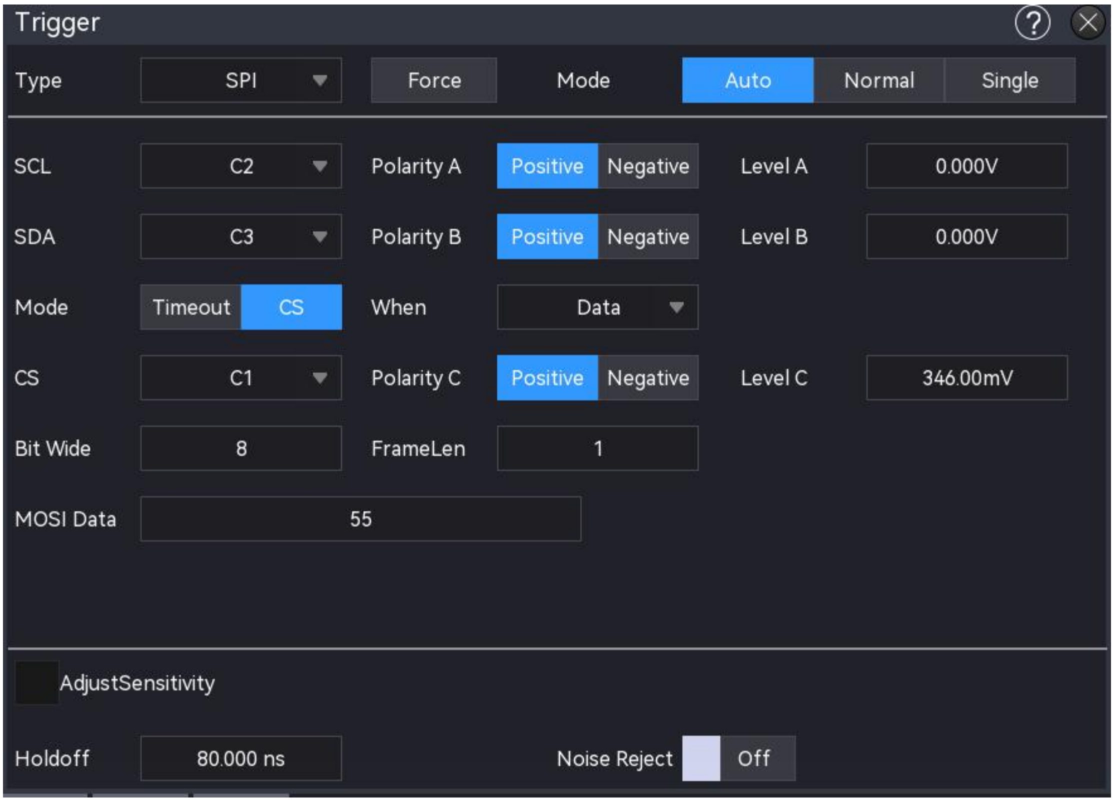
g. Address & Data: Triggers when the specified address is detected during the transmission and the data relationship meets the set condition. With this trigger condition, it can easily generate the specified address and data trigger of $I^2C$ and helpful for the user to analyze the transmission.

When the address length, address, byte length, or data is selected, the corresponding menus can be configured. For the setting of each parameter, refer to *Trigger Condition Address and Data* above.

## 8.16. SPI Triggering

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select SPI as the trigger type.




(2) Source Setting

Set the clock source, data source, and CS (Chip Selection) source. The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

a. Clock Source

Click SCL to select C1-C4 as the clock source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

b. Data Source

Click SDA to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

c. Chip Selection Source

It can be set when the mode is CS. Click CS to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

(3) Edge Setting

a. Clock Polarity

Click Polarity A to select Positive or Negative.

Positive: Triggers on the positive edge of the clock signal.

Negative: Triggers on the negative edge of the clock signal.

b. CS Polarity

It can be set when the mode is CS. Click Polarity C to select Positive or Negative.

Positive: The signal is set to 1 if it is greater than the threshold; otherwise, it is 0.

Negative: The signal is set to 1 if it is less than the threshold; otherwise, it is 0.

c. Data Polarity

Click Polarity B to select Positive or Negative.

Positive: The signal is set to 1 if it is greater than the threshold; otherwise, it is 0.

Negative: The signal is set to 1 if it is less than the threshold; otherwise, it is 0.

(4) Trigger Level

Tap Level A, Level B, or Level C, then double-click corresponding input field to open the numeric keypad and set the trigger level.

(5) Mode

Click Mode to select SPI mode. In SPI mode, timeout and chip selection can be set.

- Timeout: After the clock signal (CLK) remains idle for the specified time, the oscilloscope triggers when it detects data on MISO that meets the trigger conditions.
- Chip selection: When the Chip Select (CS) signal is valid, the oscilloscope triggers when it

detects data on SDA that meets the trigger conditions.

(6) Trigger Condition

Set the trigger condition for SPI. The start and data conditions can be configured.

a. Start: A clock signal will be generated when the timeout meets the trigger condition. When the timeout is selected, the corresponding menu can be configured.

- Timeout: Double-click Timeout input field to open the numeric keypad and set the timeout. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 3.2 ns to 1 s.

b. Data: A trigger will be generated when the clock signal meets the idle time and the data signal satisfies the data condition.

When the mode is set to timeout, timeout, data bit width, and data can be configured.

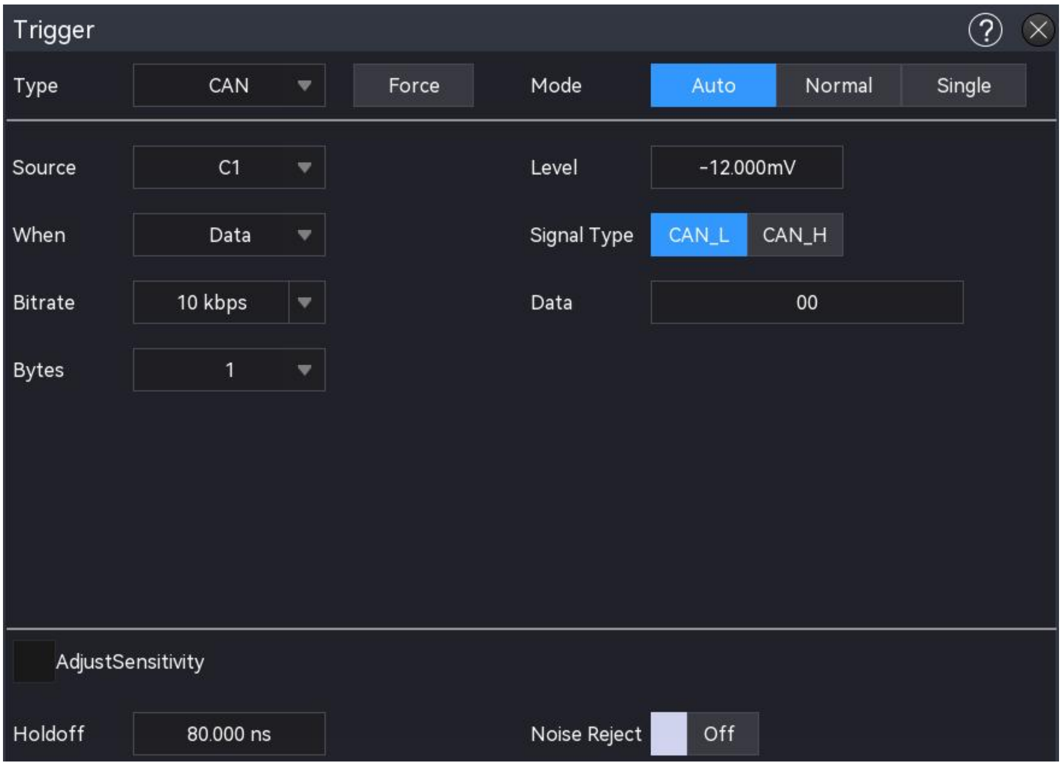
When the mode is set to CS, data bit width, frame length, and data can be configured.

- Timeout: Sets the timeout. For the configuration method, refer to Timeout mentioned above.
- Data bit width: Sets the bit width for each unit in SPI protocol, double-click Bit Width input field to open the numeric keypad and set the data bit width. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 4 to 32 bits.
- Frame length: Set the length for data unit. Double-click Frame Length input field to open the numeric keypad and set the frame length. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 32.
- Data: The data is related to the frame length. Double-click Data input field to open the numeric keypad and set the data value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 – FFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFF (32 Fs).

## 8.17. CAN Triggering

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select CAN as the trigger type.




(2) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(3) Trigger Level

Double-click Level input field to open the numeric keypad and set the trigger level.

When the trigger level is adjusted, a dotted line appears on the screen to indicate the current trigger level. After the adjustment stops, the dotted trigger level line disappears automatically after approximately 2 seconds.

(4) Signal Type

Select whether the current signal accessed by the source is a high data line signal or a low data line signal. Click Signal type to select CAN_H, or CAN_L.

(5) Bitrate

Select the bitrate for CAN serial bus data, click Bitrate to select 10 kbps, 19.2 kbps, 20 kbps, 33.3 kbps, 38.4 kbps, 50 kbps, 57.6 kbps, 62.5 kbps, 83.3 kbps, 100 kbps, 115.2 kbps, 125 kbps, 230.4 kbps, 250 kbps, 490.8 kbps, 500 kbps, 800 kbps, 921.6 kbps, 1 Mbps, 2 Mbps, 3 Mbps, 4 Mbps, 5 Mbps, or custom.

If Custom is selected, a custom bitrate can be entered.

(6) Trigger Condition

a. Start frame: Triggers on the start of the CAN signal frame.
b. Data frame: Triggers on the data frame that matches the CAN signal.
c. Remote frame: Triggers on the remote frame.
d. Error frame: Triggers on the error frame of the CAN signal.
e. Overload frame: Triggers on the overload frame of the CAN signal.
f. Identifier: Triggers on the data frame that matches the specified ID.
   When the identifier, frame format, or direction is selected, the corresponding menus can be configured.
   - Identifier: Double-click Identifier input field to open the numeric keypad and set the identifier. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 000 to 7FF and from 00000000 to 1FFFFFFF.
   - Frame format: Click Frame Format to set the format to standard or extend. The different frame formats have different ID ranges.
   - Direction: Click Direction to set the direction for the identifier. The direction can be set to Write, Read, or Read or Write.
     - Write: The oscilloscope will be generated when CAN protocol Read/Write bit is Write.
     - Read: The oscilloscope will be generated when CAN protocol Read/Write bit is Read.
     - Read or write: The oscilloscope will be generated when CAN protocol Read/Write bit is Read or Write.
g. Data: Triggers when the data acquired by CAN matches the custom data. This function helps users quickly locate transmission signals containing the specified data of interest. When the byte length or data is selected, the corresponding menus can be configured.
   - Byte length: Different byte lengths have different data ranges. Double-click Byte Length input field and select the byte length. The setting range is from 1 to 8.
   - Data: Set the trigger data, double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.
h. ID and Data: Triggers on the data frame that matches the specified ID and data.
   When the identifier, frame format, direction, byte length, or data is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *Identifier* and *Data* mentioned above.
i. End of frame: Triggers on the end of frame of CAN signal.
j. Loss confirmed: Triggers on the loss confirmed of CAN signal.
k. Bit stuff error: In the segment requiring bit filling, the waveform will trigger upon detecting an error from 6 consecutive bits of the same level.

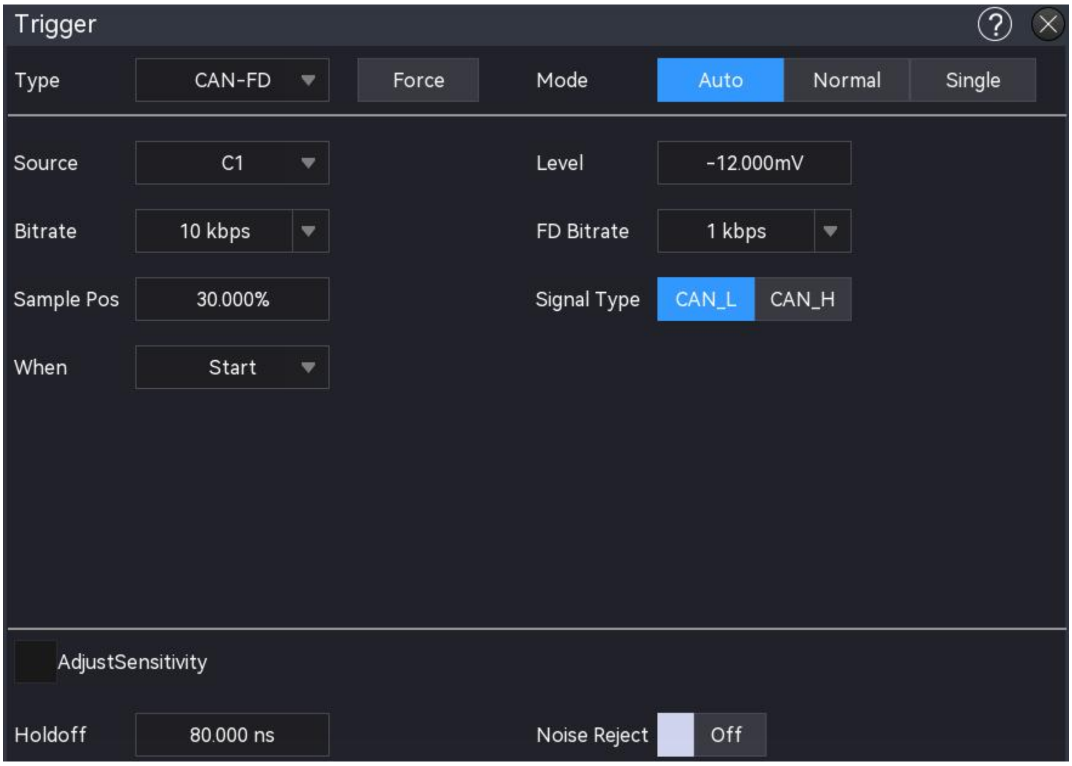
l. CRC error: Triggers when a CRC error occurs.

m. All error: Triggers on all errors, including bit stuff errors and CRC errors.

## 8.18. CAN-FD Triggering

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select CAN-FD as the trigger type.




(2) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(3) Trigger Level

Double-click Level input field to open the numeric keypad and set the trigger level.

When the trigger level is adjusted, a dotted line appears on the screen to indicate the current trigger level. After the adjustment stops, the dotted trigger level line disappears automatically after approximately 2 seconds.

(4) Signal Type

Select whether the current signal accessed by the source is a high data line signal or a low data line signal. Click Signal type to select CAN_H or CAN_L.

(5) Bitrate

Select the bitrate for CAN-FD serial bus data, click Bitrate to select 10 kbps, 19.2 kbps, 20 kbps, 33.3 kbps, 38.4 kbps, 50 kbps, 57.6 kbps, 62.5 kbps, 83.3 kbps, 100 kbps, 115.2 kbps, 125 kbps, 230.4 kbps, 250 kbps, 490.8 kbps, 500 kbps, 800 kbps, 921.6 kbps, 1 Mbps, 2 Mbps, 3 Mbps, 4 Mbps, 5 Mbps, or custom.

If Custom is selected, a custom bitrate can be entered.

(6) FD Bitrate

Select the FD bitrate for CAN-FD serial bus data, click FD Bitrate to select 250 kbps, 500 kbps, 800 kbps, 1 Mbps, 1.5 Mbps, 2 Mbps, 4 Mbps, 5 Mbps, 6 Mbps, 8 Mbps, or a custom FD bitrate.

If Custom is selected, a custom FD bitrate can be entered.

(7) Sampling Position

The sample position is the point in the bit time where the oscilloscope samples the bit level. The sample position is expressed as a percentage of the "Time from bit start to sample point" and the "Bit time".

Click Sampling position input field to open the numeric keypad and set the sampling position. The range can be set to from 30% to 90%.

(8) Trigger Condition

a. Start frame: Triggers on the start of the CAN-FD signal frame.

b. Data frame: Triggers on the data frame that matches the CAN-FD signal.

c. Remote frame: Triggers on the remote frame.

d. Error frame: Triggers on the error frame of the CAN-FD signal.

e. Overload frame: Triggers on the overload frame of the CAN-FD signal.

f. Identifier: Triggers on the data frame that matches the specified ID.

   When the ID or frame format is selected, the corresponding menus can be configured.

   - ID: Double-click ID input field to open the numeric keypad and set the ID. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The ID range can be set from 000 to 7FF and from 00000000 to 1FFFFFFF.
   - Frame format: Click Frame Format to set the format to standard, extend, FD standard, or FD extend. The different frame formats have different ID ranges. The “Standard, Extend” format is suitable for CAN signal. “FD standard, FD extend” format is suitable for CAN-FD signal.

g. Data: Triggers when the data acquired by CAN-FD matches the custom data. This function helps users quickly locate transmission signals containing the specified data of interest.

   When the byte length, data, offset, or byte offset is selected, the corresponding menus can be configured.

- Byte length: Different byte lengths have different data ranges. Double-click Byte Length input field and select the byte length. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 16.
- Data: Set the trigger data, double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.
- Offset: Set the data offset of byte data for delay trigger. Click Offset to turn it ON or OFF.
  ON: Displays the Byte Offset menu.
  OFF: Hides the Byte Offset menu.
- Byte offset: Double-click Byte Offset input field to open the numeric keypad and set the byte offset. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 63.

h. ID and Data: Triggers on the data frame that matches the specified ID and data.
   When the ID, frame format, byte length, data, offset, or byte offset is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *Identifier* and *Data* mentioned above.

i. End of frame: Triggers on the end of frame of CAN - FD signal.

j. Loss confirmed: Triggers on the loss confirmed of CAN - FD signal.

k. Bit stuff error: In the segment requiring bit filling, the waveform will trigger upon detecting an error from 6 consecutive bits of the same level.

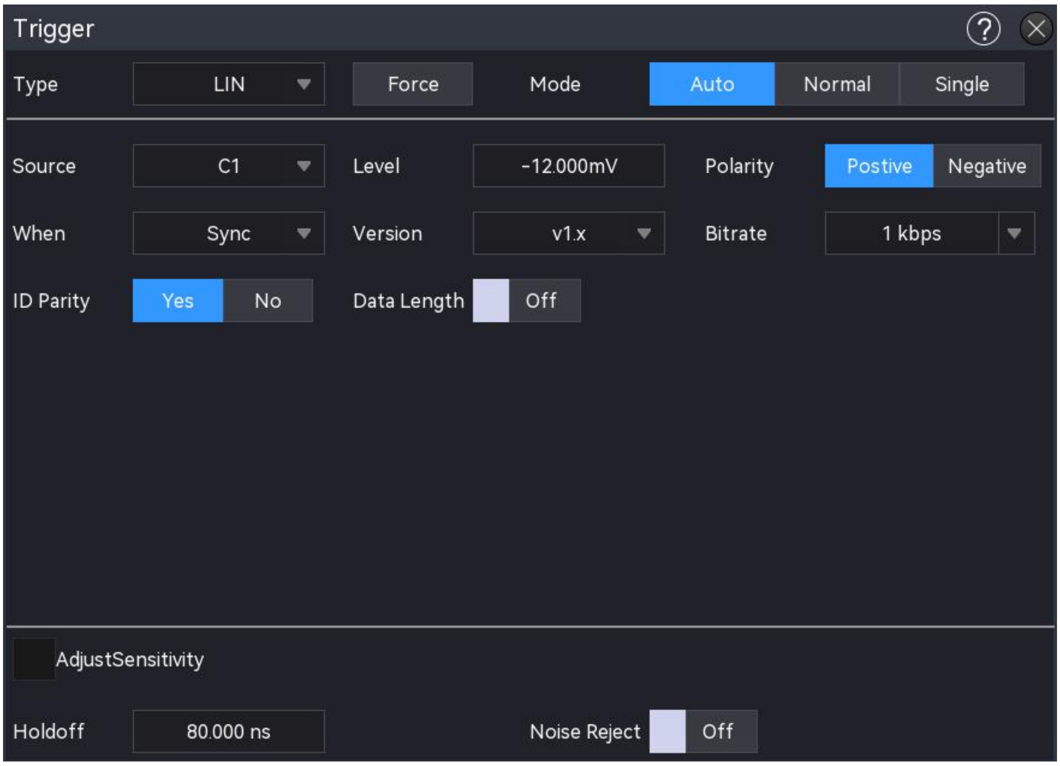
l. CRC error: Triggers when a CRC error occurs.

m. All error: Triggers on all errors, including bit stuff errors and CRC errors.

## 8.19. LIN Triggering

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select LIN as the trigger type.




(2) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(3) Polarity

Click Polarity to select the positive or negative.

(4) Trigger Level

Double-click Level input field to open the numeric keypad and set the trigger level.

When the trigger level is adjusted, a dotted line appears on the screen to indicate the current trigger level. After the adjustment stops, the dotted trigger level line disappears automatically after approximately 2 seconds.

(5) Version

Click Version to select the signal version to v1.x, v2.x, or either.

(6) Bitrate

Select the bitrate for LIN, click Bitrate to select 1.2 kbps, 2.4 kbps, 4.8 kbps, 9.6 kbps, 10.417 kbps, 19.2 kbps, 20 kbps, or custom.

If Custom is selected, a custom bitrate can be entered.

(7) ID Parity Check

Set ID Parity check to turn it ON or OFF.

ON: Includes the parity bit and ID.

OFF: Excludes the parity bit and ID.

(8) Data Length Menu

Set whether to display the data length menu. Click Data Length to turn it ON or OFF.

ON: Displays the data length menu.

OFF: Hides the data length menu.

(9) Data Length

Set LIN data length. Double-click Data Length input field to open the numeric keypad and set the data length. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data length can be set from 1 to 8. It is only available when the data length menu is set to ON.

(10) Trigger Condition

a. Synchronization: Triggers on a synchronization signal.

b. Identifier: Triggers when the detected ID matches the configured frame.

   When the ID is selected, the ID menu can be configured.

   - ID: Double-click ID input field to open the numeric keypad and set the ID. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

     When ID including parity bit is set to ON, the range is from 00 to FF.

     When ID including parity bit is set to OFF, the range is from 00 to 3F.

c. Data: Triggers when the data matches the custom data. This function helps users quickly locate transmission signals containing the specified data of interest.

   When the byte length or data is selected, the corresponding menus can be configured.

   - Byte length: Different byte lengths have different data ranges. Double-click Byte Length input field and select the byte length. The setting range is from 1 to 8.
   - Data: Set the trigger data. Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 00 to FFFFFFFFFFFFFFFF.

d. ID and Data: Triggers on the data frame that matches the specified ID and data.

   When the ID, byte length, or data is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *Identifier* and *Data* mentioned above.

e. Wake-up frame: Triggers on the signal's wake-up frame.

f. Sleep frame: Triggers on the signal's sleep frame.

g. Error: Triggers on the LIN signal’s sleep frame.

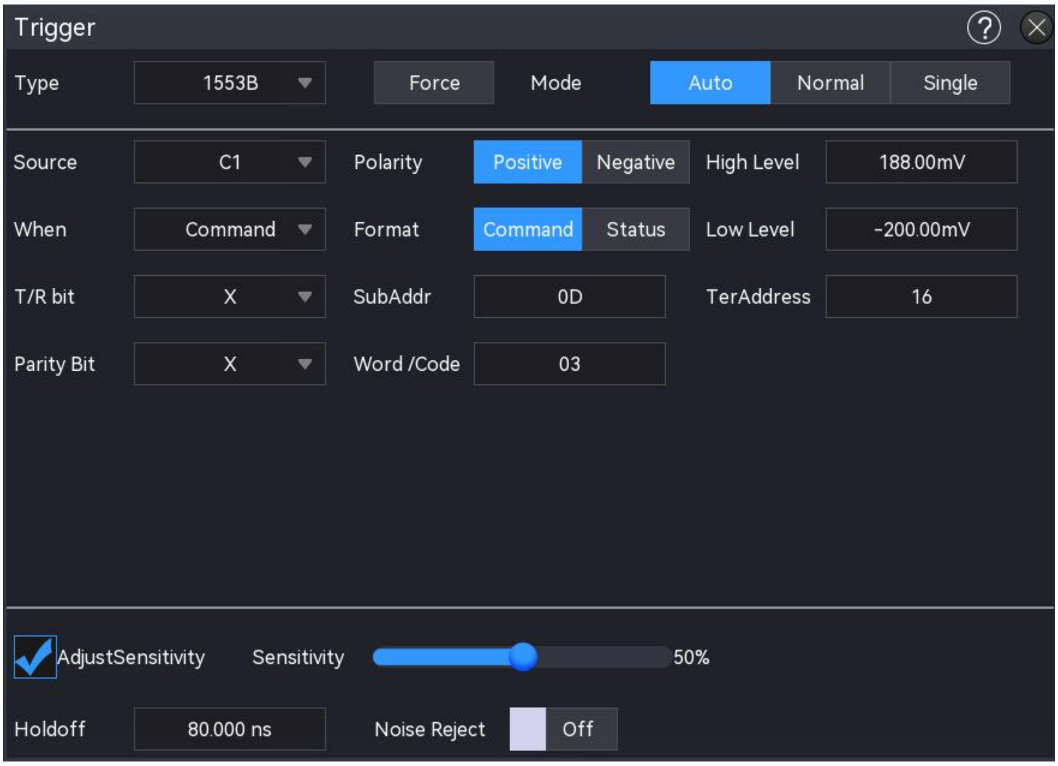
   When the error type is selected, the error type menu can be configured.

- Error type: Click Error Type to select synchronization, ID parity check, or checksum.
  - Synchronization: Synchronizing error
  - ID Parity check: ID parity check error
  - Checksum: Data check and error

## 8.20. 1553B Triggering

(1) Trigger Type

Tap the T trigger label at the top to open the “Trigger” menu. Tap “Trigger Type” to open the dropdown menu, then select “1553B” to configure the trigger settings.




(2) Source

Click on the “Source” to select C1 - C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the trigger label at the top of the screen.

The source can only trigger stably if the selected channel has a connected signal and is set as the trigger source.

(3) Polarity

Click on the “Polarity” to select “Positive” or “Negative”.

(4) High/Low Level

Double-click on “High Level (Low Level)” input field to open the numeric keypad to set the trigger level.

(5) Format

Click on the “Format” to set command word or state word.

If the format is the command word, the state trigger condition will be hidden.

If the format is the state word, the command trigger condition will be hidden.

(6) Trigger Condition

a. Synchronization: Triggered when a synchronization signal is detected.

b. Command: Triggered when the command fully matches the set parameters.

When the terminal address, T/R bit, sub-address/mode, word count/code, or parity check, the corresponding menus can be configured.

- Terminal address: Set the terminal address for a command word. Double-click on the “Terminal Address” input field to open the numeric keypad to set the terminal address. For details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting.
- T/R bit: Select the “T/R Bit” to set X, 0 (R), or 1 (T). The default is X.
- Sub-address/mode: Set the sub-address for a command word. Double-click on the “Sub-address” input field to open the numeric keypad to set the sub-address. For details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting.
- Word count/code: Set the word count/code for a command word, double-click on the “Word count/code” input field to open the numeric keypad to set the word count/code. For details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting.
- Parity check: Select the “Parity Check” to set X, 0, or 1. The default is X.

c. State: Triggered when the state word completely matches the set parameters. When the terminal address, error message (9), Instr (10), service request (11), BCR (15), Busy (16), system flag (17), DBCA (18), terminal flag (19), or parity check, the corresponding menus can be configured.

- Terminal address: Set the terminal address for a state word. Double-click on the “Terminal Address” input field to open the numeric keypad to set the terminal address. For details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting.
- Error message (9): Select the “Error message (9)” to set X, 0, or 1. The default is X.
- Instr (10): Select the “Instr (10)” to set X, 0, or 1. The default is X.
- Service request (11): Select the “Service request (11)” to set X, 0, or 1. The default is X.
- BCR (15): Select the “BCR (15)” to set X, 0, or 1. The default is X.
- Busy (16): Select the “Busy (16)” to set X, 0, or 1. The default is X.

- System flag (17): Select the "System flag (17)" to set X, 0, or 1. The default is X.
- DBCA (18): Select the "DBCA (18)" to set X, 0, or 1. The default is X.
- Terminal flag (19): Select the "Terminal flag (19)" to set X, 0, or 1. The default is X.
- Parity check: Select the "Parity Check" to set X, 0, or 1. The default is X.

d. Data: Triggered when the data word matches the set parameters.

When the data or parity check is selected, the corresponding menus can be configured.

- Data: Triggered on the specified data word. Double-click on the "Data" input field to open the numeric keypad to set the data. For details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting.
- Parity check: Select the "Parity Check" to set X, 0, or 1. The default is X.

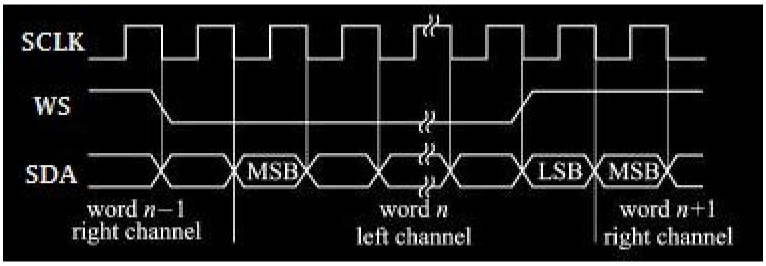
e. Error: Triggered on the specified error type.

When the error type (Parity check, synchronization, Manchester, and non-continuous data) is selected, the corresponding menus can be configured.

- Parity check: Triggered when the odd or even parity check is incorrect for the data in the word.
- Synchronization: Triggered when an invalid synchronizing pulse is detected.
- Manchester: Triggered when a Manchester error is detected.
- Non-continuous data: Triggered when non-continuous data is detected.

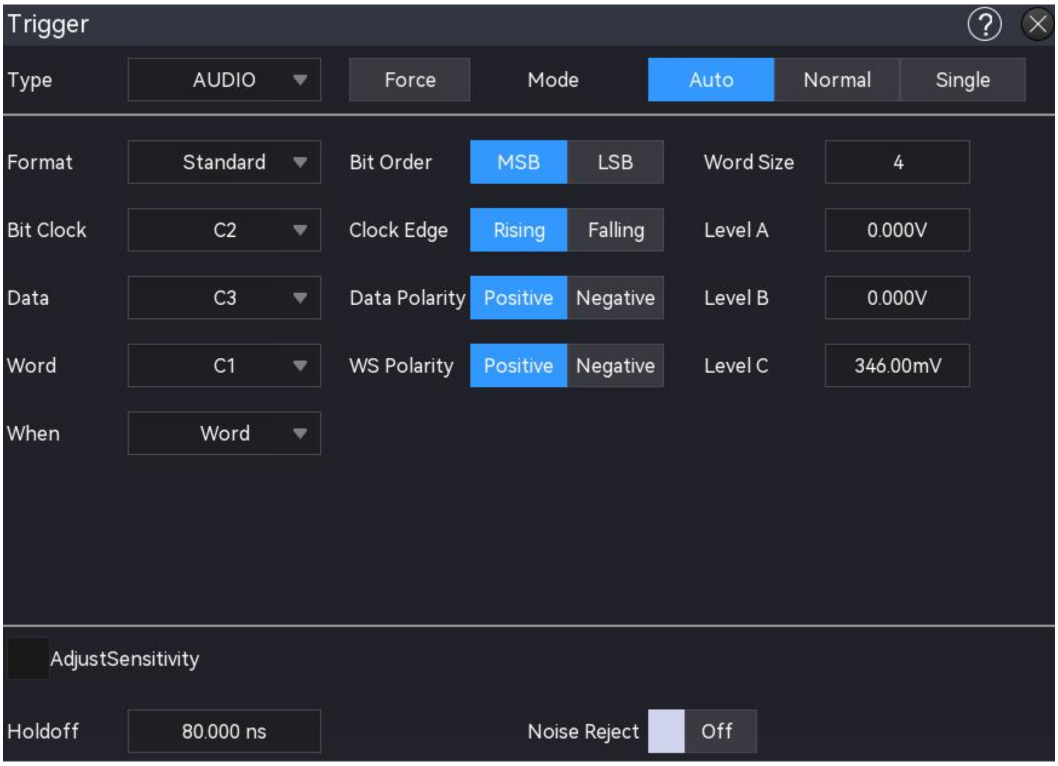
## 8.21. Audio Triggering

In the Audio triggering type, the oscilloscope recognizes the trigger condition by referencing the specified data value. The serial clock line (SCLK) should be specified, which provides one clock pulse for each bit of transmitted digital audio data, along with the frame clock line (WS), which toggles to indicate the active audio channel, and the serial data line (SDA), which transmits the audio data in two complement formats. The following figure shows the Audio bus timing diagram.




(1) Trigger Type

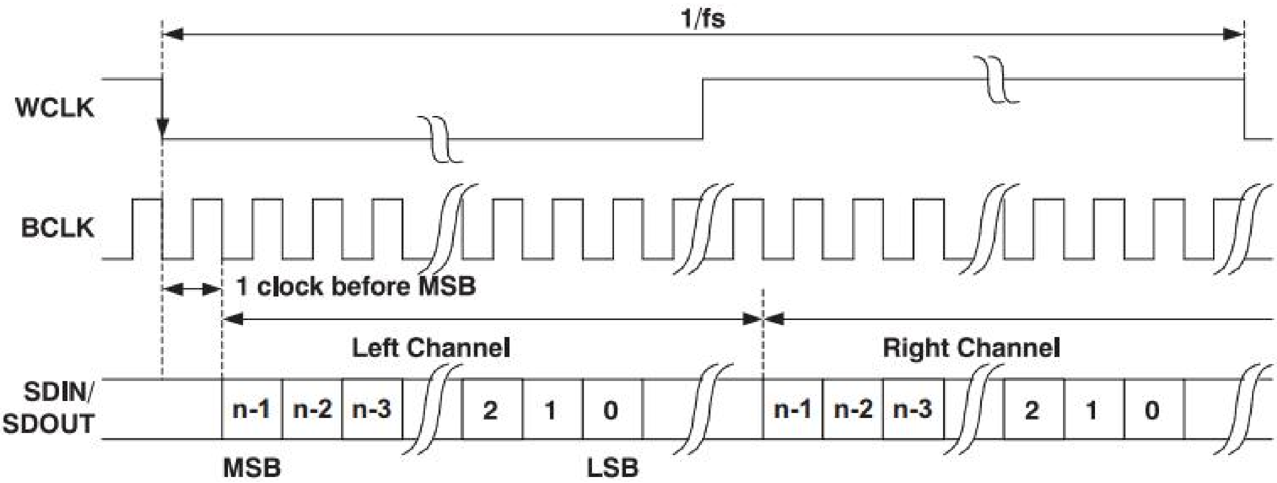
Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Audio as the trigger type.



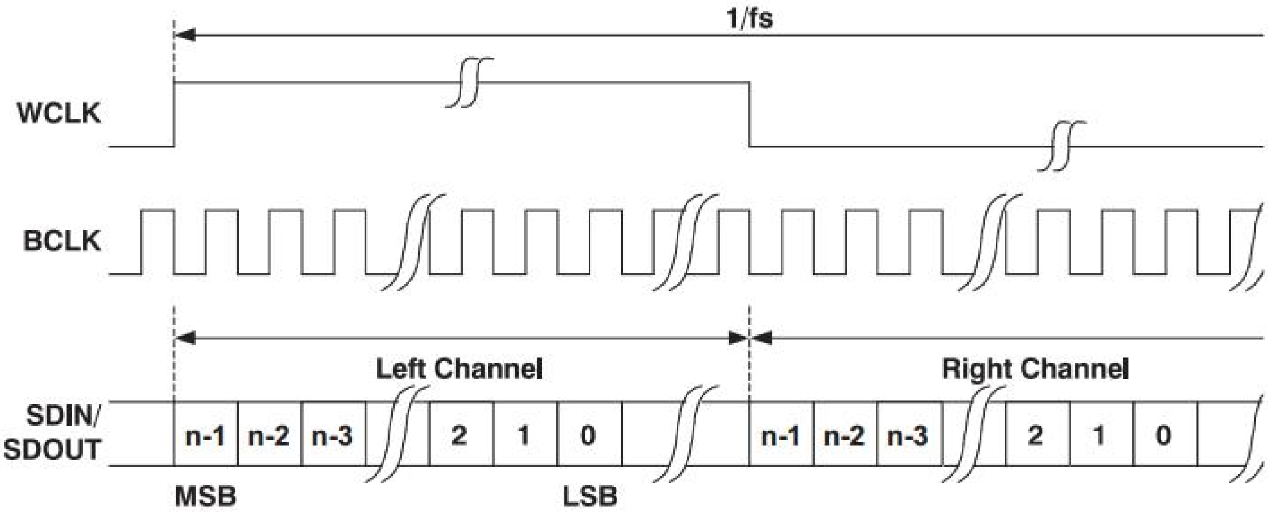

(2) Format

Click Format to select standard, left justifying, right justifying, or TDM.

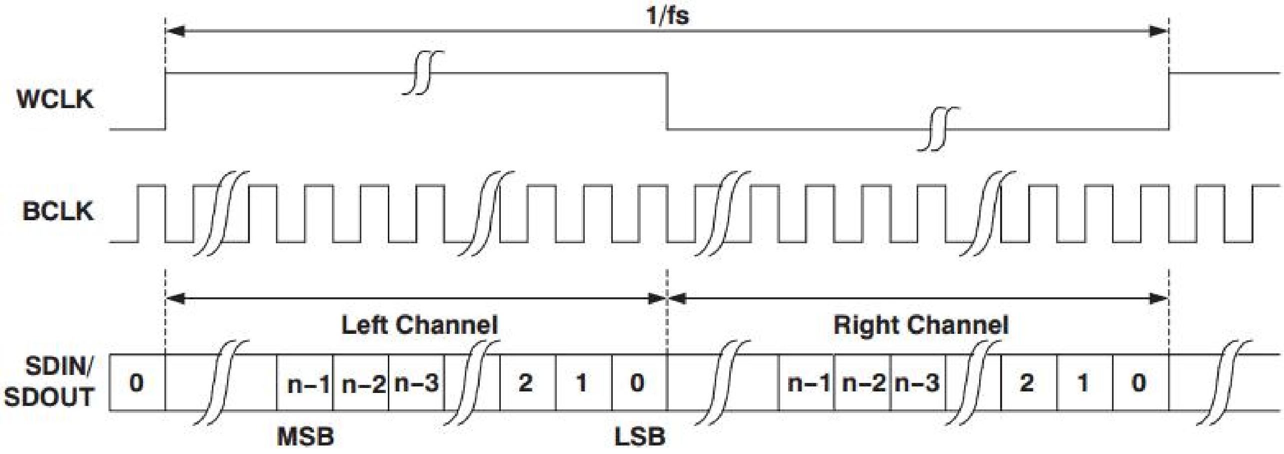
- Standard: The MSB of each sampled data is sent first, followed by the LSB. The MSB is displayed on the SDATA line one clock bit after the WS transition edge.



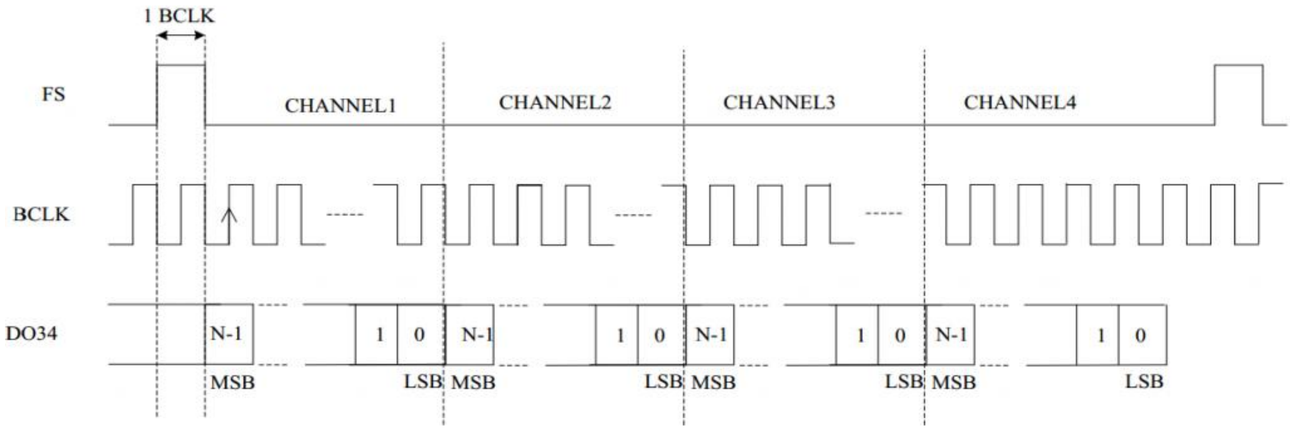

- Left justifying: Data transmission (MSB first) starts at the WS transition edge, without the one-bit delay used in the standard format.




- Right justifying: Data transmission (MSB first) is right aligned with the WS signal.




- TDM: (Time Division Multiplexing) mode allows the transmission of multi-channel data.




(3) Bit Order

Click Bit Order to select LSB or MSB. The default is MSB.

(4) Source Setting

Set the bit clock, bit selection, and data source. The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

a. Bit Clock

Click Bit Clock to select C1-C4 as the trigger source. For more details on *Trigger Source*, refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen. The clock line (SCLK) provides the clock signal for synchronizing audio data transmission.

b. Bit Selection

Click Bit Selection to select C1-C4 as the trigger source. For more details on *Trigger Source*, refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen. The bit selection indicates the audio data of the current transmission is left channel or right channel.

c. Data

Click Data to select C1-C4 as the trigger source. For more details on *Trigger Source*, refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen. The data line is used to transmit actual audio data.

d. Frame Synchronization

Click Frame Synchronization to select C1-C4 as the trigger source. For more details on *Trigger Source*, refer to the section Noun Explanation of Triggering System. The current source is displayed in the trigger label at the top of the screen.

(5) Edge Setting

a. Clock Edge

Click Clock to select Rising or Falling edge.

Rising edge: Samples SDA on the rising edge of the clock.

Falling edge: Samples SDA on the falling edge of the clock.

b. WS Polarity

Click WS Polarity to select Normal or Reverse. The WS polarity determines the valid level for the bit selection signal. The bit selection signal indicates the start frame and end of frame for the audio data.

c. Data Polarity

Click Data Polarity to select High=1 or High=0.

d. Polarity Synchronization

Click Polarity Synchronization to set the edge for the frame synchronization signal to the Rising or Falling edge.

(6) Trigger Level

Tap Level A, Level B, Level C, or Level D, then double-click corresponding input field to open the numeric keypad and set the trigger level.

(7) Data Format (do not select TDM)

When the data format is set to standard, left justifying, or right justifying is selected, the bit size and trigger mode (bit selection and data) menus can be configured.

a. Bit Size

The bit size can be set when the format is standard, left justifying, or right justifying. Double-click Bit Size input field to open the numeric keypad and set the bit size. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The byte offset range can be set from 4 to 32.

b. Trigger Type

- Bit Selection: Triggers on the bit selection.
- Data: The oscilloscope will be generated when the data meets the setting value in the sound channel.

  When the Audio or data is selected, the corresponding menus can be configured.

  - Audio: Click Audio to select any, left channel, or right channel.

  - Data: Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(8) Data Format (select TDM)

When the format is standard, left justifying, right justifying, data bit per channel, clock bit per channel, channel number per frame, bit delay, or trigger condition (frame synchronization, data, channel, and data) is selected, the corresponding menus can be configured.

a. Data Bit per Channel

Double-click Data Bit per Channel input field to open the numeric keypad and set this value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 4 to 32.

The set value of each channel data bit <= the set value of each channel clock bit.

b. Clock Bit per Channel

Double-click Clock Bit per Channel input field to open the numeric keypad and set this value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 4 to 32.

c. Channel Number per Frame

Double-click Channel Number per Frame input field to open the numeric keypad and set this value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 2 to 64.

d. Bit Delay

Double-click Bit Delay input field to open the numeric keypad and set this value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 31.

The set value of bit delay < The set value of each channel clock bit.

e. Trigger Condition

- Synchronization frame: Triggers on the synchronization frame.
- Data: Triggers when the data meets the setting value.

  When the data is selected, the data menu can be configured.

  - Data: Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.
- Channel and data: Triggers when the channel and data meet the setting value.

  When the channel or data is selected, the corresponding menus can be configured.

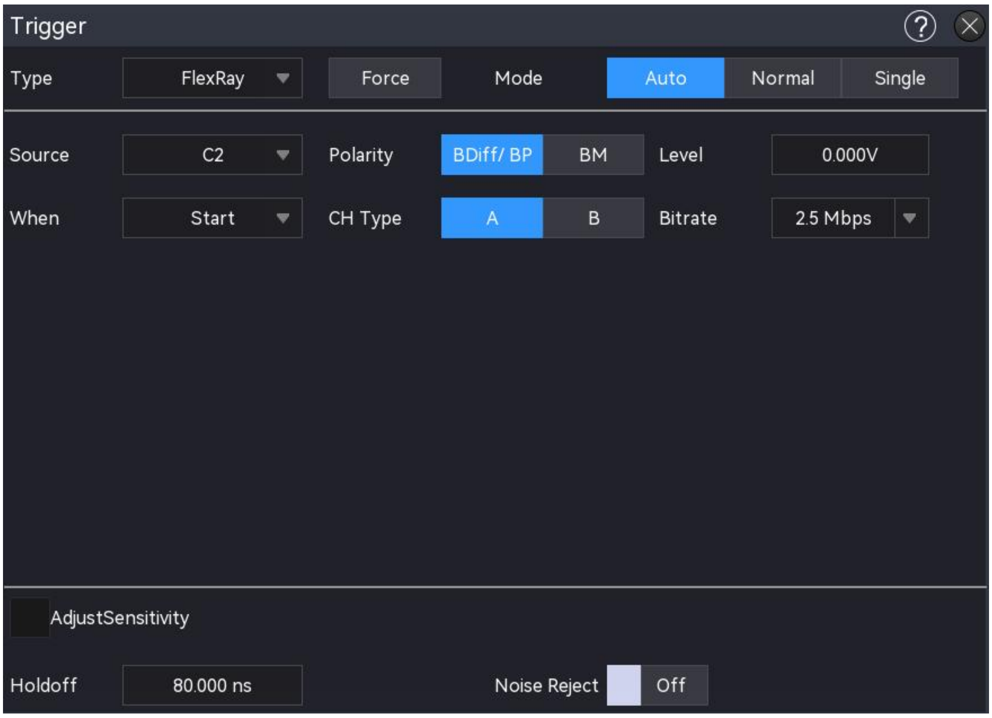
  -Channel: Double-click Channel input field to open the numeric keypad and set the channel number. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

  - Data: Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

## 8.22. FlexRay Triggering

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select FlexRay as the trigger type.




(2) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(3) Polarity

Click Polarity to select BDiff, BP, or BM.

(4) Trigger Level

Double-click Level input field to open the numeric keypad and set the trigger level.

(5) Channel Type

Click Channel Type to select A or B.

(6) Bitrate

Click Bitrate to select 2.5 Mbps, 5 Mbps, 10 Mbps, or custom.

If Custom is selected, a custom bitrate can be entered.

(7) Trigger Condition

a. Start frame: Triggers at the start of a frame.

b. Indicating bit: Triggers when the acquired data matches the set indicating bit.

   When the indicating bit is selected, the indicating bit menu can be configured.

   - Indicating bit: Sets the indicating bit for FlexRay trigger, click Indicating Bit to set normal (01XX), static load (11XX), null (00XX), synchronization (XX10), or start (XX11).

c. Identifier: Triggers when the acquired data matches the set identifier.

   When the ID is selected, the ID menu can be configured.

   - ID: Double-click ID input field to open the numeric keypad and set the ID. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 000 to 7FF.

d. Period number: Triggers when the acquired period number matches the configured period number.

   When the period number is selected, the period number menu can be configured.

   - Period number: Double-click Period Number input field to open the numeric keypad and set the period number. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 00 to 3F.

e. Header filed: Triggers when header filed matches the setting.

   When the identifier bit, ID, static load, header CRC, or period number is selected, the corresponding menus can be configured.

   - Identifier bit: Double-click Identifier Bit input field to open the numeric keypad and set the identifier bit. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 00 to 1F.

- ID: For setting ID, refer to Trigger Condition *Identifier* mentioned above.
- Static load: Double-click Static Load Length input field to open the numeric keypad and set the static load length. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 00 to 7F.
- Header CRC: Double-click Header CRC input field to open the numeric keypad and set the header CRC. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 000 to 7FF.
- Period number: For setting the period number, refer to Trigger Condition *Period Number* mentioned above.

f. Data: Triggers when the acquired data matches the custom data. This function helps users quickly locate transmission signals containing the specified data of interest.

When the byte length, offset, data, or byte offset is selected, the corresponding menus can be configured.

- Byte length: Different byte lengths have different data ranges. Double-click Byte Length input field and select the byte length. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The range of byte length can be set from 1 to 16.
- Offset: Click Offset input field to turn it ON or OFF.
- Data: Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The byte length range can be set from 00 to FFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFF (32 Fs).
- Byte offset: Set the byte offset and display the off-screen data on the screen. Double-click Byte Offset input field to open the numeric keypad and set the byte offset. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The byte offset range can be set from 0 to 253.

g. ID and Data: Triggers on the data frame that matches the specified ID and data.

When the ID, byte length, bias, data, or byte offset is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *Identifier* and *Data* mentioned above.

h. End of frame: Triggers at the end of frame of the oscilloscope.

When the frame type is selected, the frame type menu can be configured.

- Frame type: Click Frame Type to select static, dynamic (DTS), and all.
  - Static frame: Triggers on the static frame.
  - Dynamic frame (DTS): Triggers on the dynamic frame.

- All: Triggers on the static and dynamic frame.

i. Error: The oscilloscope will be generated when the bus error occurs.

When the error is selected, the error menu can be configured.

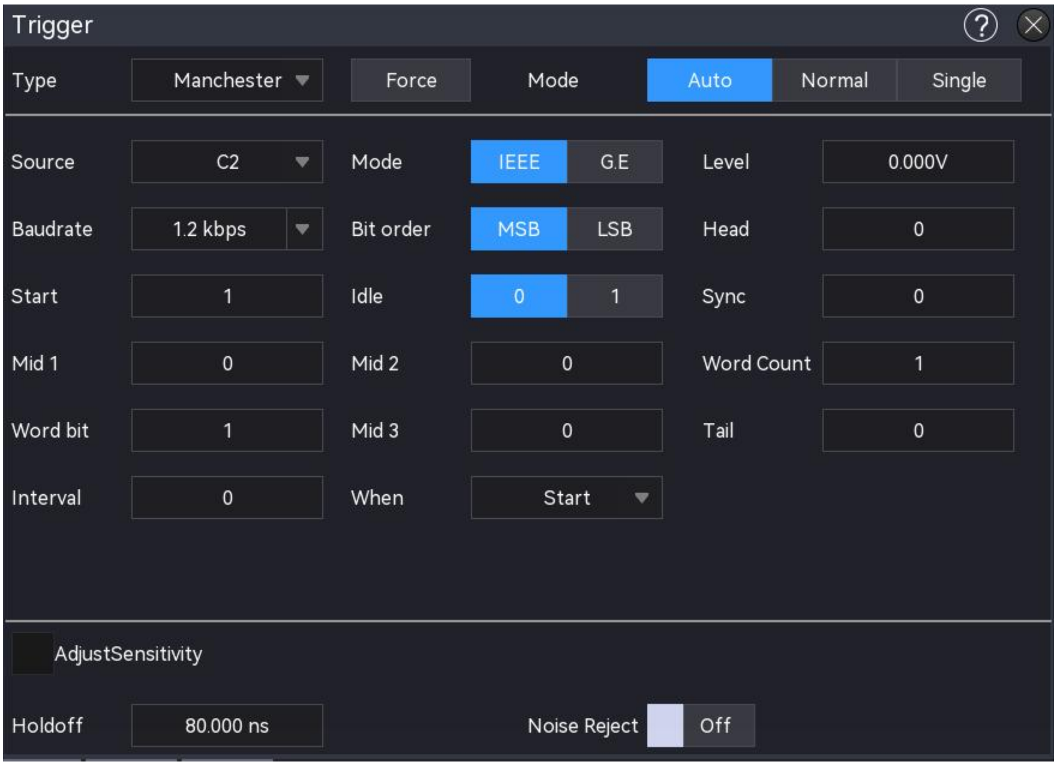
■ Error: Click Error to select header CRC, end of frame CRC, empty frame static error, empty dynamic error, synchronization frame, or start frame.

- Header CRC: Header CRC error of bus
- End of frame CRC: End of frame CRC error of bus
- Empty frame static error: Empty frame static error of bus
- Empty dynamic error: Empty dynamic error of bus
- Synchronization frame: The header frame of FlexRay has a dedicated indicating bit, the data frame will be the synchronization frame when the indicating bit is valid.
- Start frame: The start frame of FlexRay has a dedicated indicating bit, the data frame will be the synchronization frame when the indicating bit is valid.

## 8.23. Manchester Triggering

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select Manchester as the trigger type.




(2) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(3) Trigger Level

Double-click Level input field to open the numeric keypad and set the trigger level.

(4) Encode Mode

Click Encode Mode to switch between IEEE and G.E.

- IEEE: 1 indicates a low-to-high transition. 0 indicates a high-to-low transition.
- G.E: 0 indicates a low-to-high transition. 1 indicates a high-to-low transition.

(5) Bitrate

Click Bitrate to select the bitrate for DUT to 1.2 kbps, 2.4 kbps, 4.8 kbps, 9.6 kbps, 10.417 kbps, 19.2 kbps, 125 kbps, 250 kbps, 500 kbps, 1 Mbps, 2 Mbps, 5 Mbps, 10 Mbps, or custom. The custom baud bitrate must match the DUT, the default bitrate is 1.2 kbps.

(6) Bit Order

Click Bit Order to switch between MSB and LSB.

- MSB: The Most Significant Bit, i.e., the bit with the highest value in a sequence, transmitted first.
- LSB: The Least Significant Bit, i.e. the bit with the least value in a sequence, transmitted first.

(7) Idle State

Click Idle State to switch between 0 or 1.

- 0: The bus state is at a low level when no data is present.
- 1: The bus state is at a high level when no data is present.

(8) Trigger Condition

a. Start frame: Triggers at the start of a frame.

b. Header field: Triggers on the header field when the condition is met.

   When the header field is selected, the header field menu can be configured.

   - Header field: Sets the trigger data for the header field. The data length that can be set is limited by the header field length. Double-click Header field input field to open the numeric keypad and set the header field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

c. Data field: Triggers on the data field when the condition is met.

   When the data field is selected, the data field menu can be configured.

   - Bitrate: Sets the data length for the triggering data of data field, double-click Bitrate input field to open the numeric keypad and set the bitrate value. For details on the use of the numeric keyboard, refer to the section of 5.7 Parameter Setting.

- Data field: Sets the trigger data for the data field. The data length that can be set is limited by the data bit and bit size. Double-click Data Field input field to open the numeric keypad and set the data field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

d. End field: Triggers on the end field when the condition is met.

- End field: Sets the trigger data for the end field. The data length that can be set is limited by the end field. Double-click End Field input field to open the numeric keypad and set the end field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

e. Error field: Triggers on the error field.

(9) Start Frame Bit

Click Start Frame Bit input field to open the numeric keypad to enter the start frame bit. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The range can be set from 1 to 32.

(10) Synchronization Field

Click Synchronization Field input field to open the numeric keypad to enter the synchronization field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

(11) Middle Field 1

Click Middle Field 1 input field to open the numeric keypad to enter the middle field 1. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

(12) Header Field

When the header field is triggered, the setting of this parameter is applied. Click Header Field input field to open the numeric keypad and set the header field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

(13) Middle Field 2

Click Middle Field 2 input field to open the numeric keypad and set the middle field 2. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

(14) Word Count

Click Word Count input field to open the numeric keypad and set the word value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to

255.

(15) Word Bit

Click Word Bit input field to open the numeric keypad and set the word. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 8.

(16) Middle Field 3

Click Middle Field 3 input field to open the numeric keypad and set the middle field 3. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

(17) End Field

When the end field is triggered, the setting of this parameter is applied. Click End Field input field to open the numeric keypad and set the end field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

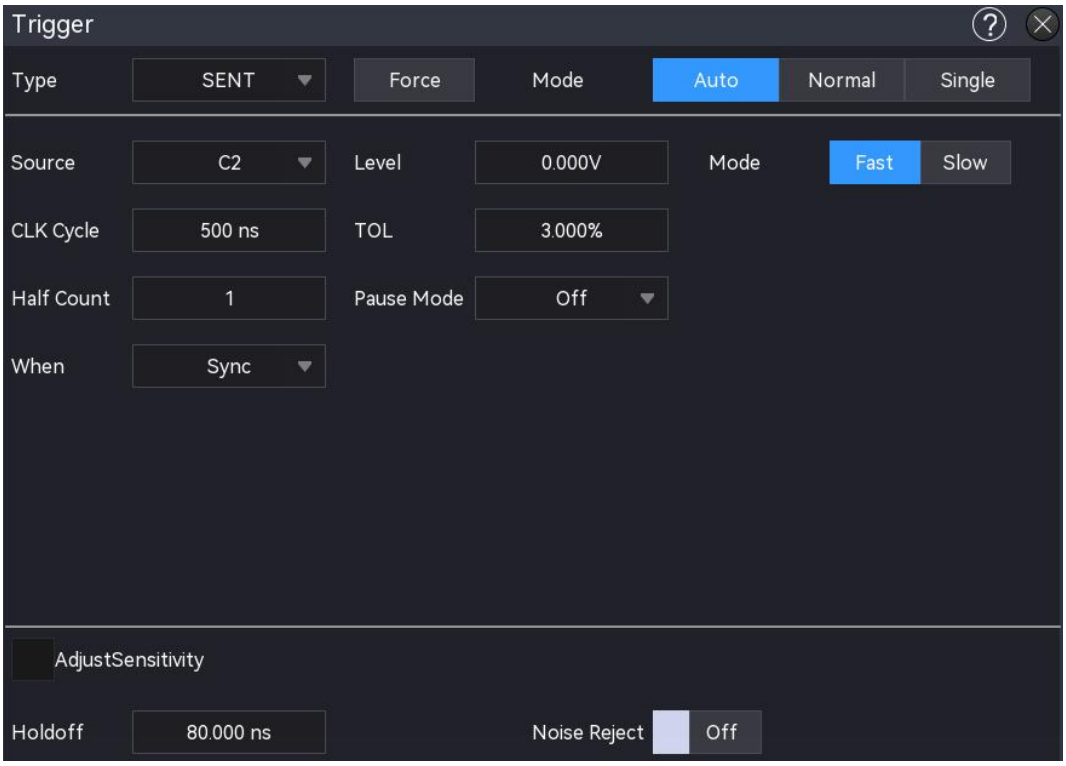
(18) Inter-frame Space

Click Inter-frame Space input field to open the numeric keypad and set the inter-frame space. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

## 8.24. SENT Triggering

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select SENT as the trigger type.




(2) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(3) Trigger Level

Double-click Level input field to open the numeric keypad and set the trigger level.

(4) Clock Period

Double-click Clock Period input field to open the numeric keypad and set the clock period. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting rang can be set from 500 ns to 300 μs.

(5) Tolerance

Set the percentage tolerance to specify a percentage tolerance for determining whether the sync pulse is valid for decoding data. If the time of the measured sync pulse is within the percentage tolerance of the rated clock period, then the decoding will continue; otherwise, the sync pulse causes an error, and data decoding will not be performed. The tolerance range can be set from 3% to 30%.

(6) Half Byte

Set the half byte for fast channel message, double-click Half Byte input field to open the numeric keypad and set the half byte. For details on using the numeric keypad, refer to section

5.7 Parameter Setting. The setting range can be set from 1 to 6.

(7) Pause Mode

Click Pause Mode to set whether there is a pause pulse between the fast channel messages. It can be switched to ON or OFF.

- OFF: There is no pause pulse between the fast channel messages.

  When no paise pulse is present, there is no idle time on the SENT serial bus. During normal operation, the fast channel decode line shows a continuous stream of packets, i.e. one packet closes, and a new packet opens immediately.
- ON: Inserts a pause pulse between fast channel messages, ensuring that the frames arrive at equal intervals.

  When the pause pulse is enabled, the idle time is displayed between messages.

(8) Mode

Click Mode to switch the trigger signal mode to fast or slow.

(9) Trigger Condition

The trigger condition can be set when the fast mode is selected, i.e. set the trigger condition under SENT fast mode.

When synchronization, state, data, CRC, state and data, state+data+CRC, fast CRC error, or continuous pulse error is selected, the corresponding menus can be configured.

a. Synchronization: Triggers on the synchronization data.

b. State: Triggers on the specified state condition.

   When the state data is selected, the state data menu can be configured.

   - State data: Double-click State Data input field to open the numeric keypad and set the state data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

c. Data: Triggers on the specified data condition.

   When the half byte, data field, or data field is selected, the corresponding menus can be configured.

   - Half byte: Double-click Half Byte input field to open the numeric keypad and set the half byte. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. Trigger half byte ≤ Half byte. The setting range is from 1 to 6.
   - Data field: Double-click Data Field input field to open the numeric keypad and set the data field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.
   - Data field format: Click Data Field Format to set the format to half byte or fast channel.

     - Half byte: Triggers when the acquired half-byte count and corresponding data match the configured settings.

- Fast channel: Triggers based on the fast channel data field.

d. CRC: CRC data will be triggered when the condition is met.

When the CRC data is selected, the CRC data menu can be configured.

■ CRC data: Double-click CRC Data input field to open the numeric keypad and set the CRC data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

e. State and data: Triggers when both the state and data conditions are met.

When the half byte, data field, data field format, or state data is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *State* and *Data* mentioned above.

f. State + data + CRC: Triggers when the state, data, and CRC conditions are met.

When the half byte, data field, data field format, state data, or CRC data is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *State*, *Data*, and *CRC* mentioned above.

g. Fast CRC error: Triggers on fast CRC error.

h. Continuous pulse error: Triggers on continuous pulse error.

(10) Frame Type

In slow mode, the frame type can be set. Click Frame Type to switch the trigger signal mode to A or B.

(11) Slow Speed Trigger Condition

In slow mode, the trigger condition for slow Speed can be set, i.e. set the trigger condition for slow SENT signal.

When the synchronization, short ID, short data, short CRC, short ID and data, enhanced ID, enhanced data, enhanced CRC, enhanced ID and data, or slow channel CRC error is selected, the corresponding menus can be configured.

a. Synchronization: Triggers on the synchronization data.

b. Short ID: Triggers when the short ID condition is met.

When the short ID menu is selected, the short ID menu can be configured.

■ Short ID: Double-click Short ID input field to open the numeric keypad and set the short ID. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

c. Short data: Triggers when the short data condition is met.

When the short data is selected, the short data menu can be configured.

■ Short data: Double-click Short Data input field to open the numeric keypad and set the short data. For details on using the numeric keypad, refer to section 5.7 Parameter

Setting.

d. Short CRC: Triggers when the short CRC condition is met.

When the short CRC is selected, the short CRC menu can be configured.

- Short CRC: Double-click Short CRC input field to open the numeric keypad and set the short CRC. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

e. Short ID+ data: Triggers when both the short ID and short data conditions are met.

When the short ID or short data is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *Short ID* and *Short Data* mentioned above.

f. Enhanced ID: Triggers when the enhanced ID condition is met.

When the enhanced ID is selected, the enhanced ID menu can be configured.

- Enhanced ID: Double-click Enhanced ID input field to open the numeric keypad and set the enhanced ID. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

g. Enhanced data: Triggers when the enhanced data condition is met.

When the enhanced data is selected, the enhanced data menu can be configured.

- Enhanced data: Double-click Enhanced Data input field to open the numeric keypad and set the enhanced data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

h. Enhanced CRC: Triggers when the enhanced CRC condition is met. When the enhanced CRC is selected, the enhanced CRC menu can be configured.

- Enhanced CRC: Double-click Enhanced CRC input field to open the numeric keypad and set the enhanced CRC. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

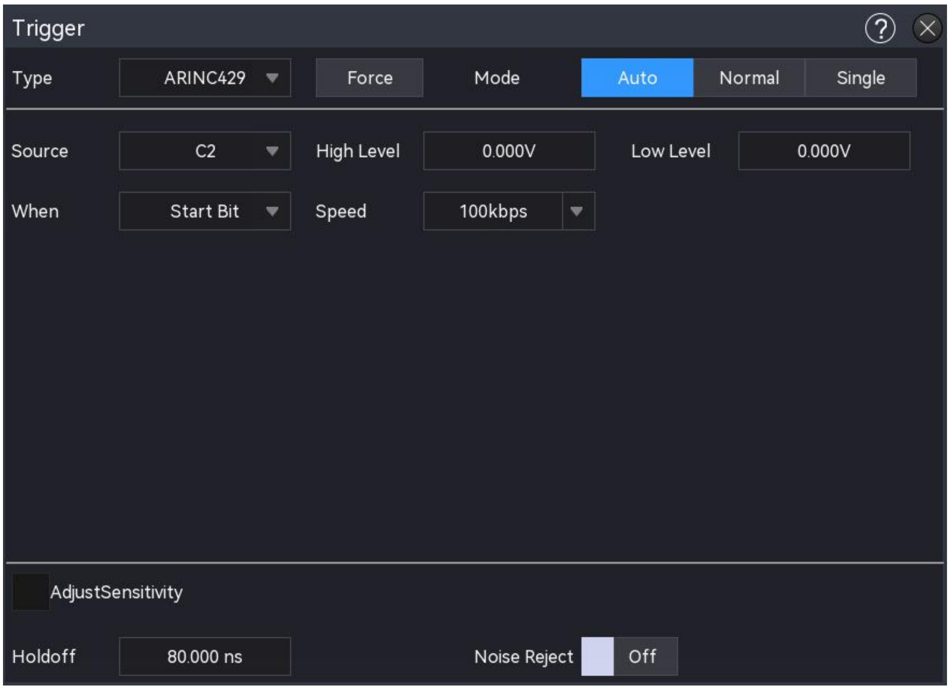
i. Enhanced ID+ data: Triggers when both the enhanced ID and enhanced CRC conditions are met. When the enhanced ID or enhanced CRC is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *Enhanced ID* and *Enhanced data* mentioned above.

j. Slow channel CRC error: Triggers on slow channel CRC error.

## 8.25. ARINC429 Triggering

(1) Trigger Type

Tap the Trigger label at the top of the screen to open the trigger menu, then click Trigger Type drop-down menu and select ARINC429 as the trigger type.




(2) Trigger Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

The selected source can trigger stably only when the corresponding channel has a valid input signal and is set as the trigger source.

(3) High/Low Level

Double-click High Level (Low Level) input field to open the numeric keypad and set the trigger level.

(4) Speed

Click Speed to set the transmission rate to high (100kb/s), low (12.5kb/s), or custom.

(5) Trigger Condition

a. Start bit: Triggers on the start bit of frame.

b. End bit: Triggers on the end bit of frame.

c. Label: Triggered on when the specified label occurs.

When the label menu is selected, the label menu can be configured.

- Label: Double-click Label input field to open the numeric keypad and set the label. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The range can be set from 00 to FF.

d. SDI: Triggers when the specified SDI occurs.

When the SDI is selected, the SDI menu can be configured.

- SDI: Double-click SDI input field to open the numeric keypad and set the SDI. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.The range can be set from 0 to 3.

e. Data: Triggers when data is acquired via the ARINC429 protocol. This function helps users quickly locate transmission signals containing the specified data of interest.
When the data is selected, the data menu can be configured.
   - Data: Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data range can be set from 00000 to 7FFFF.

f. SSM: Triggers when the Symbol State Matrix matches the custom Symbol State Matrix.
When the SSM is selected, the SSM menu can be configured.
   - SSM: Double-click SSM input field to open the numeric keypad and set the SSM. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The range can be set from 0 to 3.

g. Label + Bit: Triggers when the specified label and other fields match the configured conditions.
When the label, data, SSM, or SDI is selected, the corresponding menus can be configured. For the setting of each parameter, refer to Trigger Condition *Address, SDI, Data*, and *SSM* mentioned above.

h. Parity check error: Triggers when a parity check error occurs.

i. Bit error: Triggers when a bit error occurs.

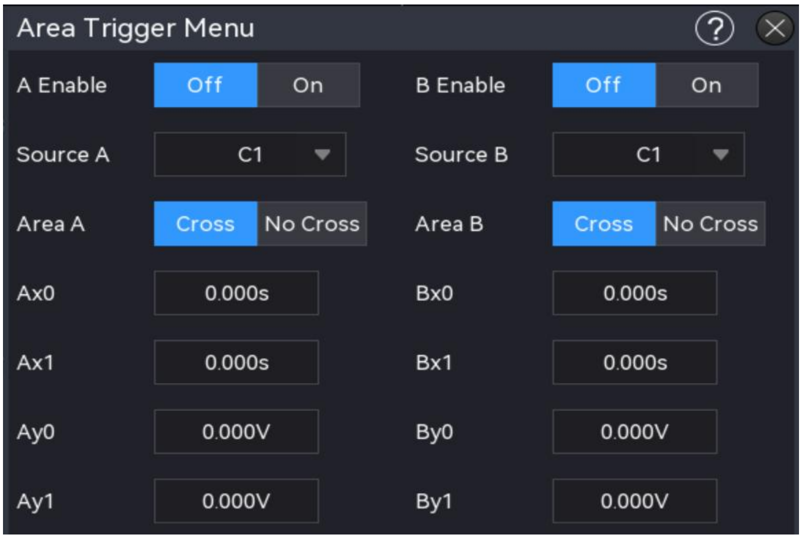
j. Interval error: Triggers when an interval error occurs.

k. All error: Triggers when one of the above errors occurs.

## 8.26. Zone Triggering

For complicated and fast-changing circuit signals during debugging, the oscilloscope with high waveform capture rate can easily observe fleeting accidental abnormal signals. 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.

The UPO6000LP Series adds the zone triggering function, providing users with an advanced and easy-to-use trigger option. To use it, enable the area drawing function and draw one or two rectangular areas on the signal of interest, allowing rapid separation and observation of specific signal portions. Zone Triggering can be combined with basic, advanced, and protocol triggers, and it also supports decoding, waveform recording, and pass/fail testing. This feature is especially useful for debugging complex signals.




Zone triggering provides two rectangle areas: Area A and Area B. Each area supports setting the trigger condition to intersection or non-intersection. Both areas also allow assigning the corresponding enable sources C1–C4.

(1) Area drawing setting menu: Enable Area A, Source A, Area A, Enable Area B, Source B, and Area B.

a. Enable Area A: Toggles Area A to ON or OFF.
   If a region box exists on the screen,
   ON: Displays the region box.
   OFF: Hides the region box.

b. Source A: Sets the source for Area A, it can be set to C1-C4.

c. Area A: Sets whether Area A intersects with Source A.

d. Enable Area B: Toggles Area B to ON or OFF.
   If a region box exists on the screen,
   ON: Displays the region box.
   OFF: Hides the region box.

e. Source B: Sets the source for Area B, it can be set to C1-C4.

f. Area B: Sets whether Area B intersects with Source B.

(2) Area box setting menu: Cancel, A: intersection, A: non-intersection, B: intersection, B: non-intersection.

a. Cancel: Closes the currently drawn area and cancel the condition setting.

b. A: intersection: Sets the currently drawn region as Area A.
   Trigger condition: Area A triggers if it intersects with the waveform and does not trigger if it does not intersect.

c. A: non-intersection: Sets the currently drawn region as Area A.

Trigger condition: Area A triggers if it does not intersect with the waveform and does not trigger if it intersects.

d. B: intersection: Sets the currently drawn region as Area B.

Trigger condition: Area B triggers if it intersects with the waveform and does not trigger if it does not intersect.

e. B: non-intersection: Sets the currently drawn region as Area B.

Trigger condition: Area B triggers if it does not intersect with the waveform and does not trigger if it intersects.

(3) Area boundary setting

Area trigger boxes can be quickly drawn using gestures. For fine-tuning the boundaries, enable the corresponding boundary input windows and adjust the values.

a. Ax0, Ax1, Ay0, and Ay1 represent the left, right, top, and bottom boundaries of Area A, respectively.

b. Bx0, Bx1, By0, By1 represent the left, right, top, and bottom boundaries of Area B, respectively.

Intersecting areas are displayed with blue borders, while non-intersecting areas are displayed with gray borders.

The setting menu can be opened by clicking the area trigger box on the screen, or by touching the horizontal and vertical positions of the area trigger box in the movable area.

When adjusting the waveform's time base or volts/div, the zone trigger boxes scale accordingly, expanding or compressing with the waveform.

Open the region trigger on the abnormal signal, as shown in the following figure.


If the currently selected area already exists, the new area trigger settings will replace the original settings, and the area trigger box will be closed. Area trigger settings are not saved when the instrument is rebooted.

**Note**: If both Area A and Area B are enabled simultaneously, the final trigger condition is the logical "AND" operation of both areas.

# 9. Protocol Decoding



Users can efficiently locate errors, debug hardware, and accelerate development by using protocol decoding, ensuring high-speed and high-quality project completion. The UPO6000LP provides four bus decoder modules (Decod 1, Decod 2, Decod 3, and Decod 4) for decoding common protocols for analog channel input signals. The UPO6000LP series supports protocol decoding for RS232, I2C, SPI, and optionally for CAN, CAN-FD, LIN, FlexRay, Audio, Manchester, SNET, and ARINC429.

As Decode 1, Decode 2, Decode 3, and Decode 4 have the same decoding function and setting method, this chapter uses Decode 1 as an example.

Access the decoding setting menu using the following steps.

- Click Home icon in the top-right corner of the screen, click decoding icon to enter the decoding setting menu.
- If the decoding function is added in the toolbar, click decoding icon in the top-right corner of the screen to enter the decoding setting menu.

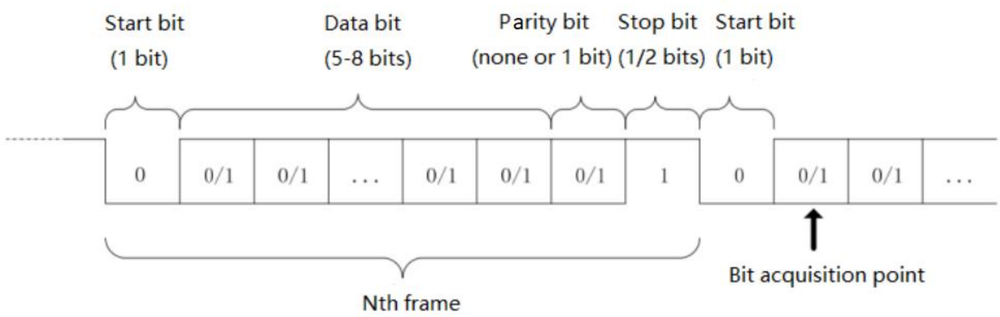
# 9.1. RS232 Decoding

RS232 is an asynchronous transmission standard interface established by Electronic Industries Association. It usually includes two application formats DB-9 or DB-25. It is suitable for the communication that the data transmission rate within the range 0-29491200/s.

It is widely used in microcomputer interface, the data to be transmitted is combined into a specified set of serial bits according to the protocol rules and sent it in an asynchronous serial way.

The data to be transmitted for each time, composed by the following rules.

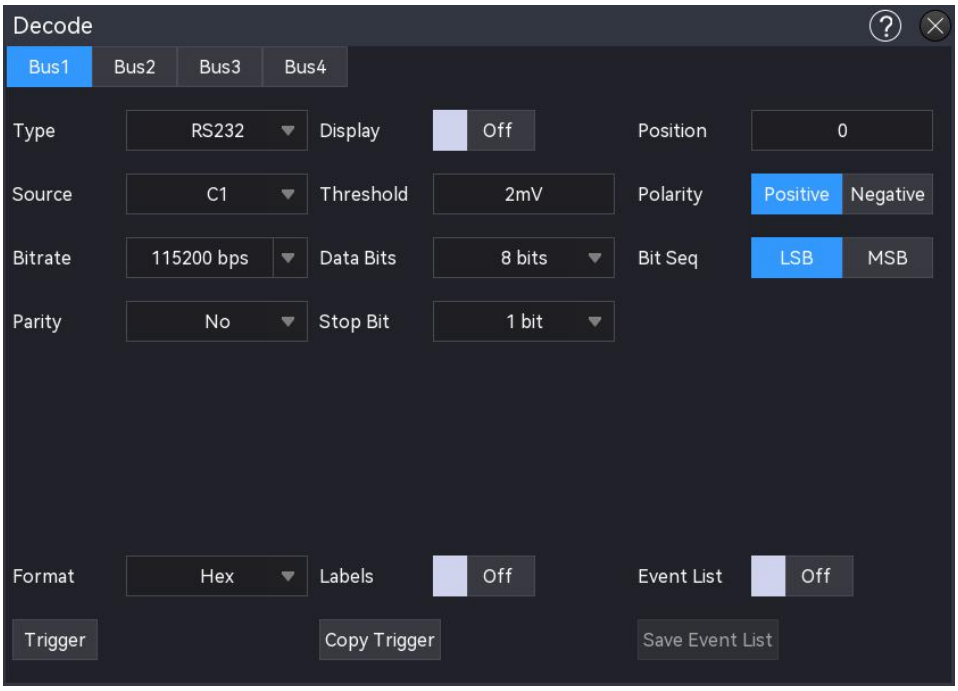
Send one start bit first, and send 5-8 data bits, and send optional parity check bit, and send one or two stop bits at last. The number of data bits is agreed by both communicating parties, it can be 5-8 bits, with no parity check bit, odd parity check bit, or even parity check bit. The stop bit can be set to one bit or two bits. In the following description, a transmission of a data string is referred to as a frame.




(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select RS232 and configure the decoding settings.



b. Source

Click Source to select C1-C4. The current source is displayed in the top-right corner of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. Threshold

Set the threshold of source. Double-click Threshold input field to open the numeric keypad and set the threshold. The threshold range is related to the vertical scale and vertical offset of the source.

d. Polarity

Click Polarity to select positive or negative.

- Negative: The reversed polarity of logic level, i.e. the high level is 0 and the low level is 1.
- Positive: The normal polarity of logic level, i.e. the high level is 1 and the low level is 0.

e. Parity Check

Set the parity check of data transmission. Click Parity to select none, even, or odd.

f. Data Bit

Set the data bit width for the specified RS232 decoding signal. Click Data Bits to select 5 bits, 6 bits, 7 bits, or 8 bits.

g. Bit Sequence

Set the data bit sequence for RS232 protocol signal. Click Bit Sequence to select MSB or LSB.

- MSB: The Most Significant Bit, i.e., the bit with the highest value in a sequence, transmitted first.
- LSB: The Least Significant Bit, i.e., the bit with the lowest value in a sequence, transmitted last.

h. Stop Bit

Set the stop bit for each data, click Stop Bit to select 1 bit or 2 bits.

i. Bitrate

When RS232 communication uses asynchronous transmission communication, no accompanying clock signal is present during data transmission process. To correctly determine the data bits, both communication parties must agree on the bit rate. The bit rate is generally defined as the number of bits transmitted in one second. For example, 9600 bps means that 9600 bits are transmitted in one second. The bitrate is not directly equal to the effective data transmission rate. Note that the start bit, data bit, parity bit, and stop bit are all included in the total bit count. The oscilloscope sets the bitrate according to the bit sampling.

The bitrate can be set to 2400 bps, 4800 bps, 9600 bps, 19200 bps, 38400 bps, 57600 bps, 115200 bps, or custom. Open the numeric keypad to set the custom bitrate.

It is recommended to configure the bitrate according to the RS232 communication hardware and software. Due to the characteristics of this transmission protocol, RS232 protocol is typically used for short distance (less than 20 m), and low speed (less than 1 Mbps) transmission occasions. Communication beyond this range is susceptible to interference and may become unreliable.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to open the numeric keypad and set the position. The range can be set from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, binary, or ASCII.

d. Label

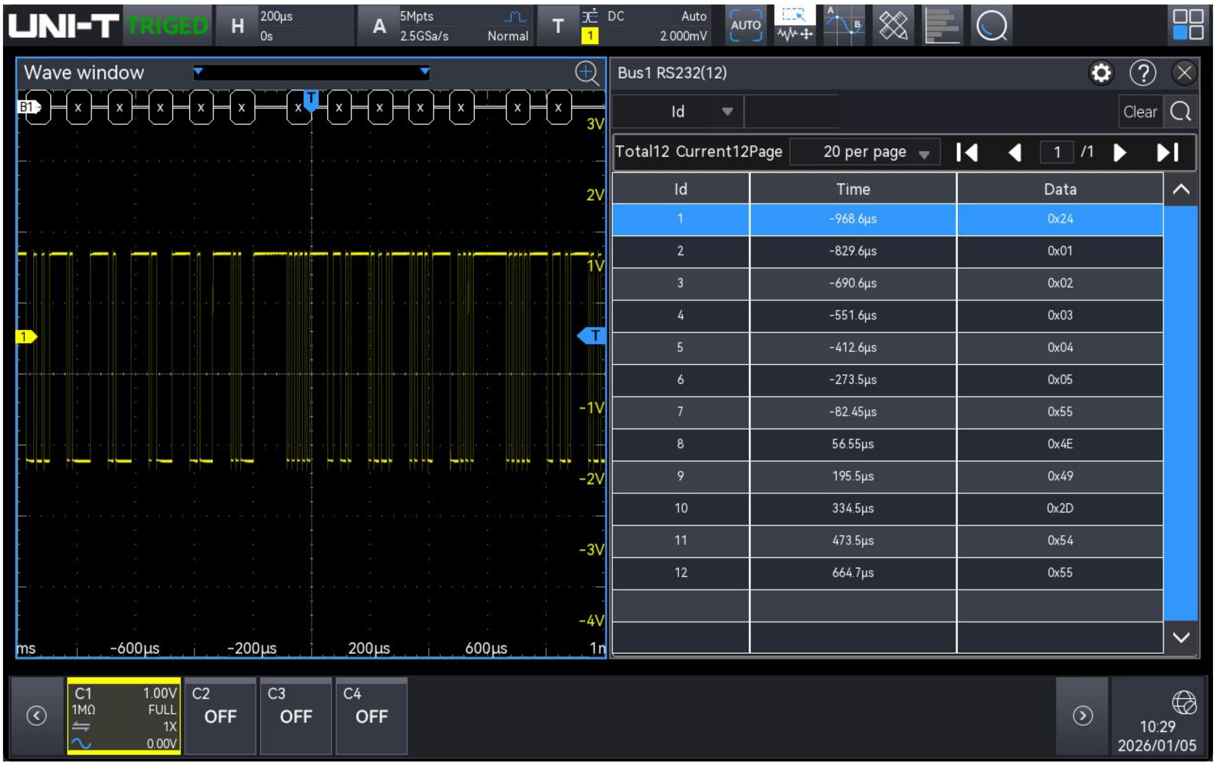
Click Label to turn the decoding bus label on or off. ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type. OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.




(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

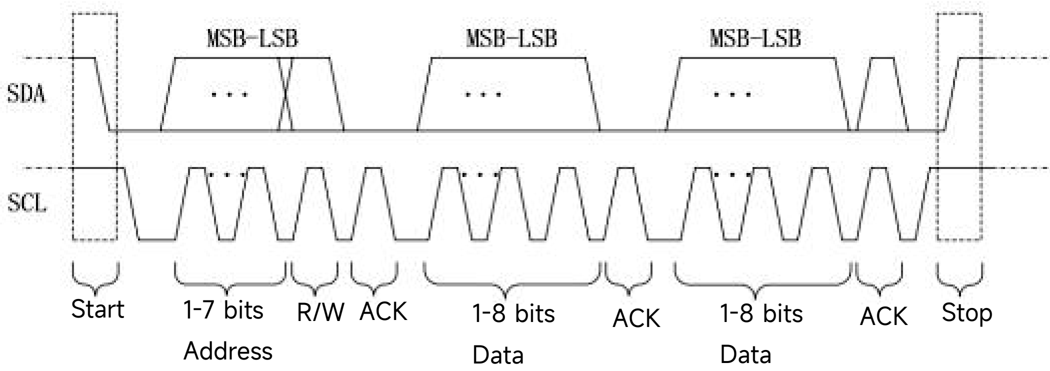
(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

## 9.2. $I^2C$ Decoding

$I^2C$ trigger is usually used to connect microcontrollers and peripheral devices, it's widely used in microelectronics area. This bus protocol has two lines to transmit, one line is serial data SDA, and another line is serial clock SCL. Use a master-slave system for communication, enabling two-way communication between the master and slave computers.

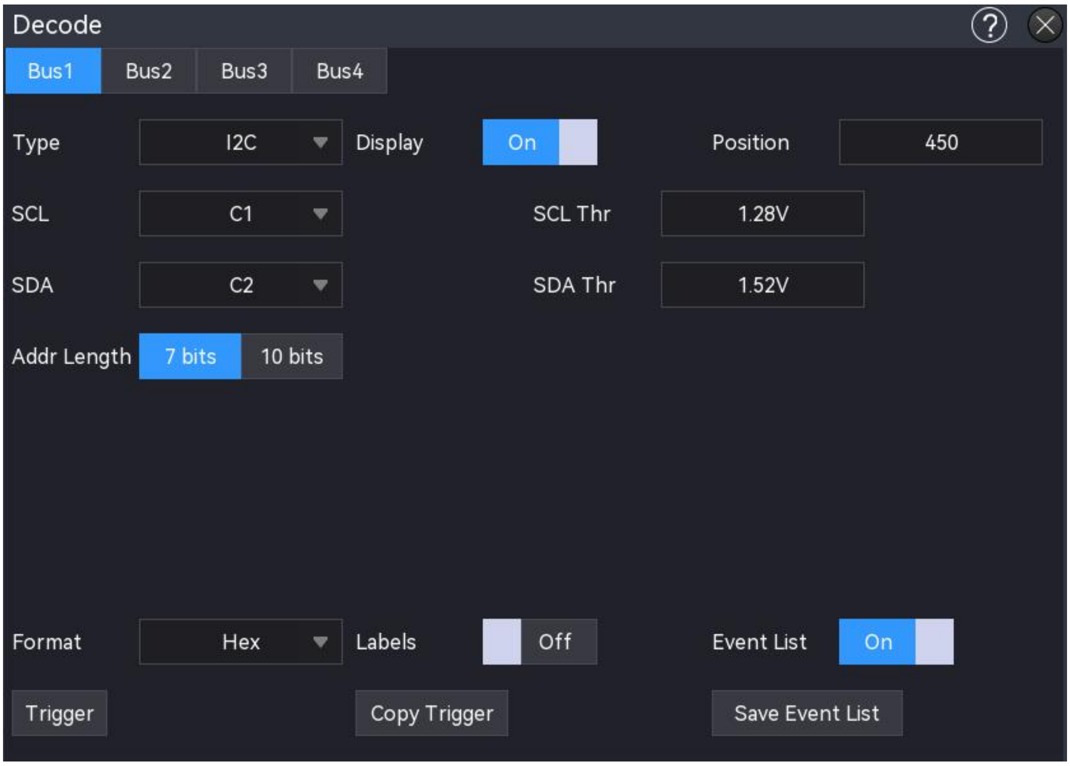
This bus is the bus of multiple masters, preventing data corruption through conflict demodulation and arbitration mechanisms. It should be noted that the $I^2C$ bus supports two address bit width: 7 bits and 10 bits. 10 bits and 7 bits address are compatible and can be used together. SCL and SDA on the $I^2C$ bus can both be connected to the positive supply via a pull-up resistor. When the bus is idle, both lines are high level. When any device on the bus outputs a low level, it causes the bus signal to become low, i.e. a logical "AND" between the signals of multiple devices. This special logical relationship is the key to realizing bus arbitration. The protocol requires the data signal SDA to remain stable while the clock signal SCL is high, and data is usually transmitted in MSB form, as shown in the following figure.




(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select I2C and configure the decoding settings.



b. Source Setting

Set the clock source, and data source. Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

- Clock Source

  Click Clock Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

- Data Source

  Click Data Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

c. Threshold Setting

Select SCL threshold or SDA threshold, then double-click corresponding Threshold input field to open the numeric keypad and set the threshold.

d. Address Length

Set the address bit width of $I^2C$ signal, click Address Length and select 7 bits or 10 bits.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to open the numeric keypad and set the position. The range can be set from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Label

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

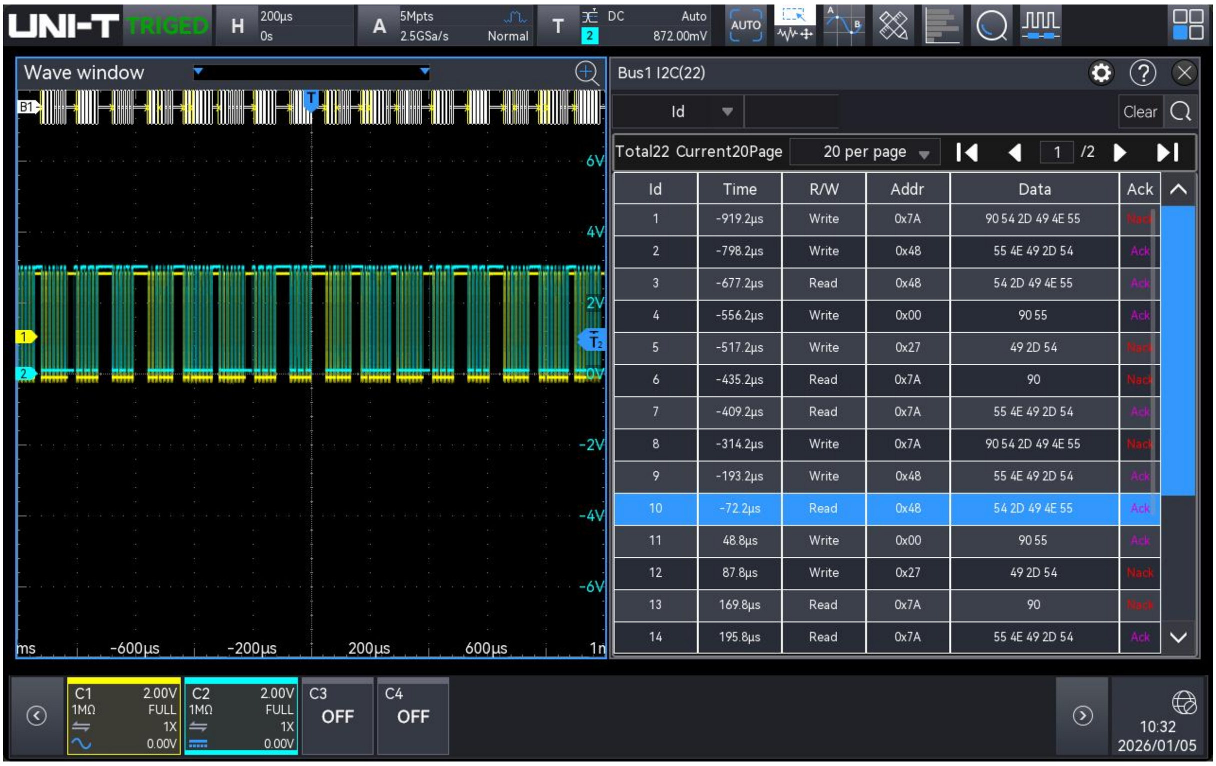
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.




(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

## 9.3. SPI Decoding

SPI (Serial Peripheral Interface) allows the host to communicate with peripheral devices in a serial way. It's full-duplex and synchronous communication bus.

It's usually use 4 signal connection line: MOSI (Master Out, Slave In): Data output from the master device, and input to the slave device.

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

SCLK: Clock signal generated by the master device.

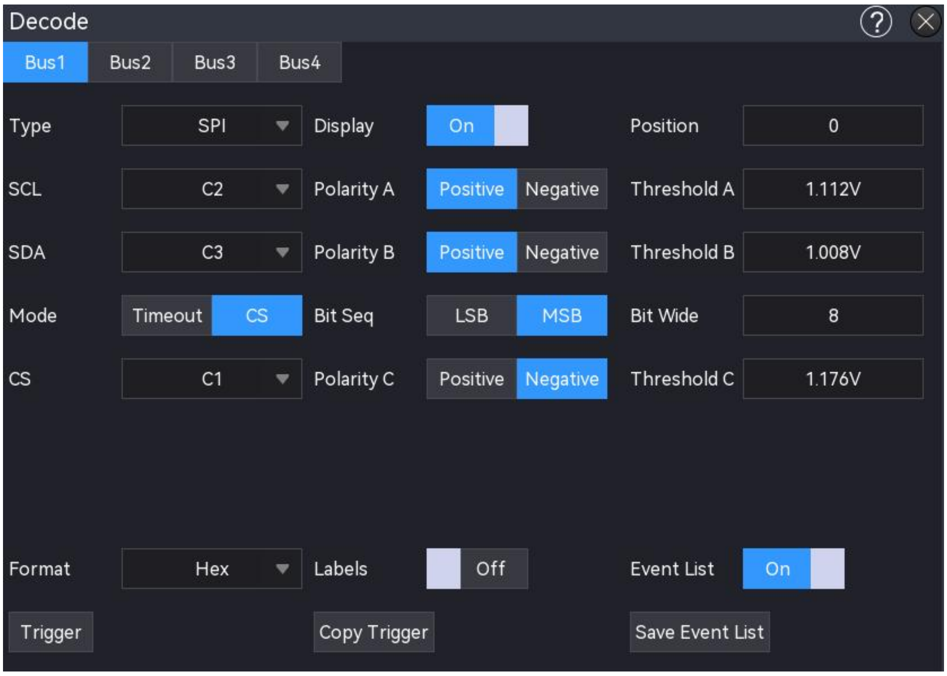
CS: Chip select used to enable the slave device.

The SPI interface is mainly used for synchronous serial data transfer between the host and low-speed peripherals. Data is transferred bit by bit under the shift pulse of the master device, and the transfer format is MSB. SPI interface is widely used because it does not require slave address addressing, it is full duplex communication and the protocol is simple. The transmission of SPI protocol is shown in the following figure.

(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select SPI and configure the decoding settings.




b. Source Setting

Set the clock source, data source, and CS source. Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

- Clock Source

  Click Clock Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

- Data Source

  Click Data Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed

in the trigger label at the top of the screen.

- Chip Source

  Click Chip Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

c. Edge Setting

- Clock Polarity

  Click Clock Polarity to select Positive or Negative.

  Positive: Triggers on the positive edge of the clock signal.

  Negative: Triggers trigger on the negative of clock signal.

- CS Polarity

  Click CS Polarity to select Positive or Negative.

  Positive: The signal is set to 1 if it is greater than the threshold; otherwise, it is 0.

  Negative: The signal is set to 1 if it is less than the threshold; otherwise, it is 0.

- Data Polarity

  Click Data Polarity to select Positive or Negative.

  Positive: The signal is set to 1 if it is greater than the threshold; otherwise, it is 0.

  Negative: The signal is set to 1 if it is less than the threshold; otherwise, it is 0.

d. Threshold Setting

Select Threshold A, Threshold B, or Threshold C, then double-click corresponding Threshold input field to open the numeric keypad and set the threshold.

e. Mode

Click Mode to select SPI. When SPI is selected, timeout and CS menus can be configured.

- Timeout: After the clock signal (CLK) remains idle for the specified time, the oscilloscope triggers when it searches for data that meets the trigger conditions (MISO).
- CS: When the CS is valid, the oscilloscope triggers when it searches for data that meets the trigger conditions (SDA).

f. Bit Order

Set the bit sequence for RS232, click Bit Order to select MSB or LSB.

- MSB: The Most Significant Bit, i.e., the bit with the highest value in a sequence, transmitted first.
- LSB: The Least Significant Bit, i.e., the bit with the lowest value in a sequence, transmitted last.

g. Bit Width

Set the bit width for each data unit in the SPI protocol signal. Double-click Bit Width input field to open the numeric keypad and set the bit width. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 4 to 32.

h. Timeout

Double-click Timeout input field to set the timeout. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 3.2 ns to 1 s.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to set the position. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Label

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

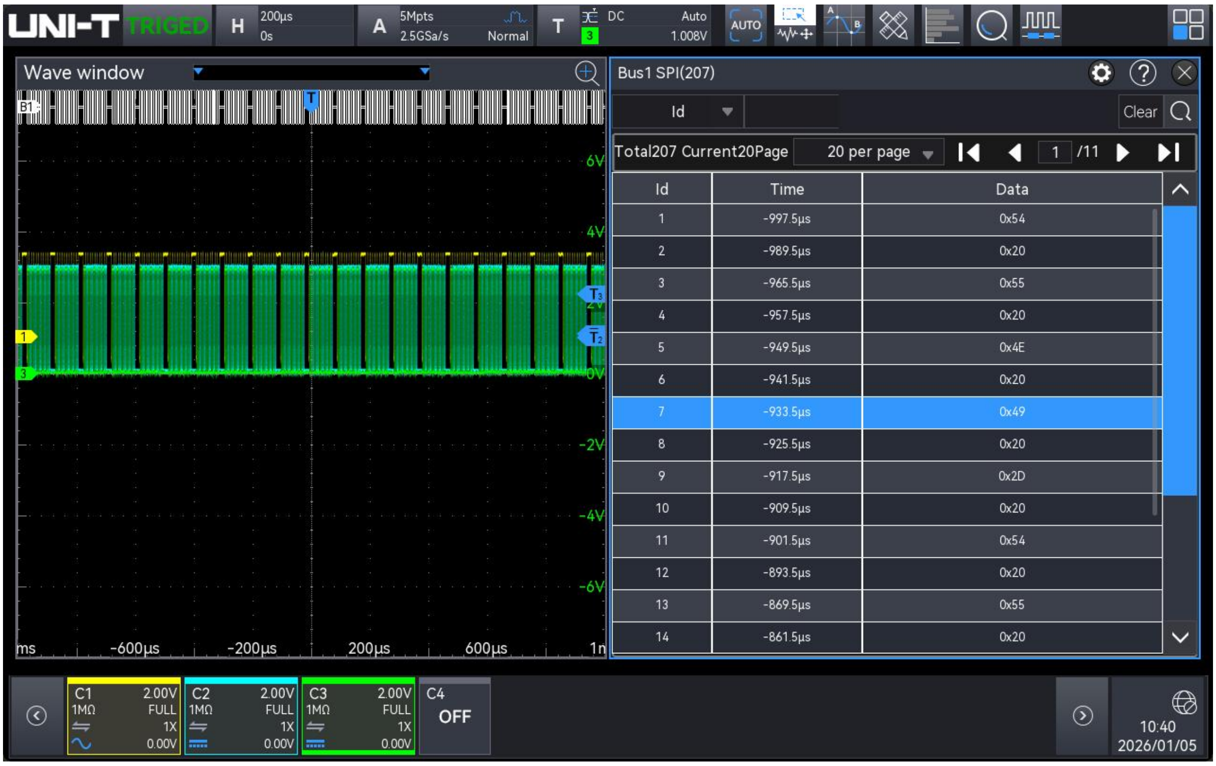
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.




(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

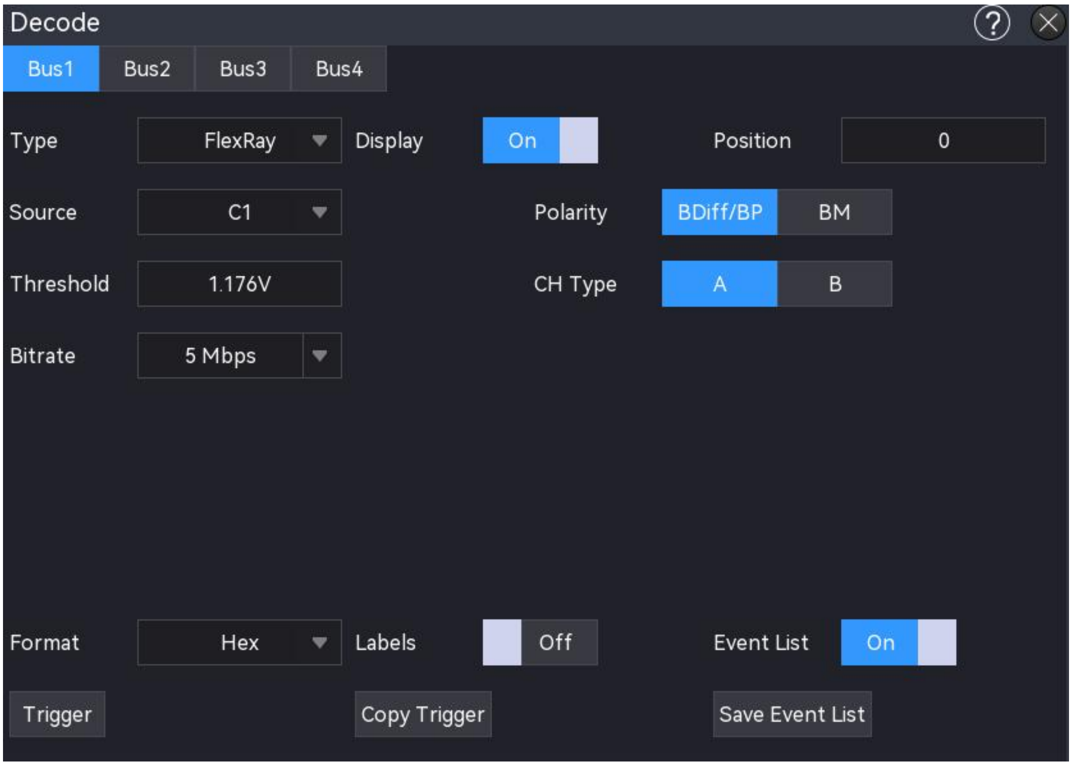
## 9.4. FlexRay Decoding

FlexRay is a differential serial bus configured with three consecutive segments (header, payload and tail). The oscilloscope samples the FlexRay signal at the specified sampling position and determines whether each data point is a logic "1" or a logic "0" based on a set threshold level. FlexRay decoding requires the signal type and rate to be specified.

(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select FlexRay and configure the decoding settings.




b. Source

Click Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. Polarity

Click Polarity to select BDiff, BP, or BM.

d. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold.

e. Channel Type

Click Channel Type to select A or B.

f. Bitrate

Click Bitrate to select 2.5 Mbps, 5 Mbps, 10 Mbp, or custom.

If Custom is selected, a custom bitrate can be entered.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to set the position. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Label

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

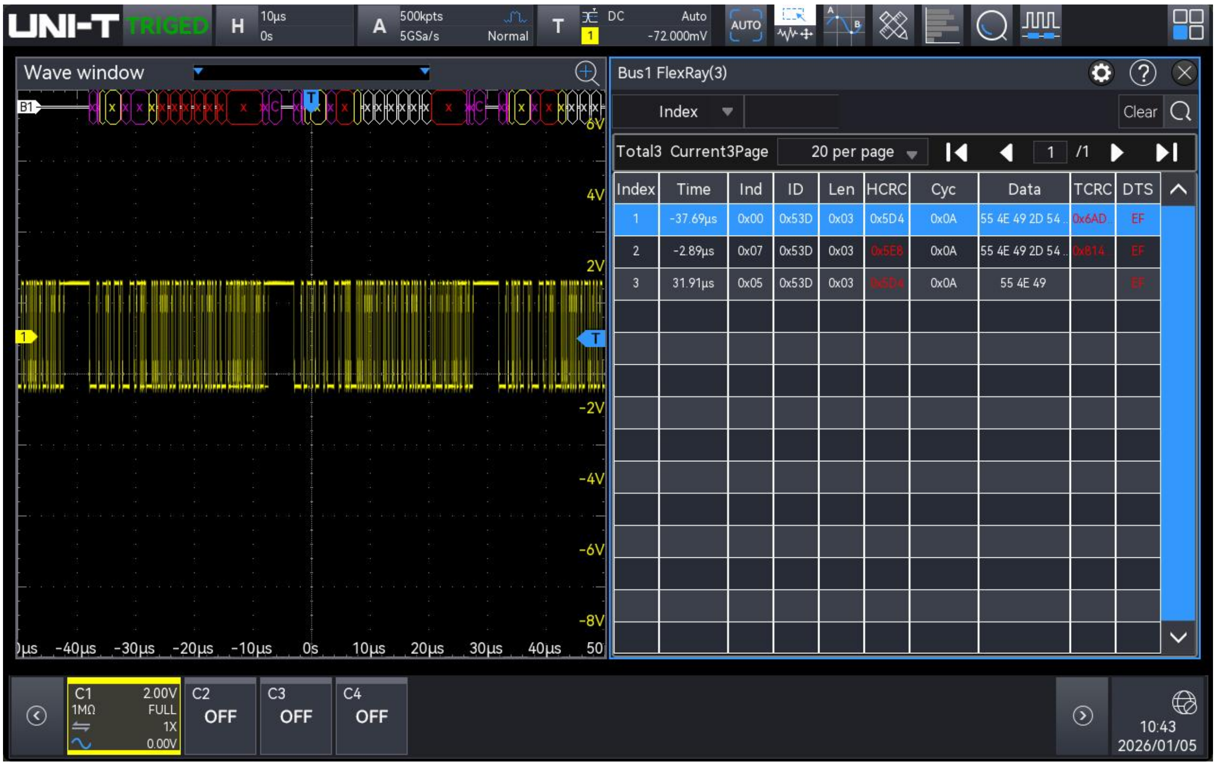
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.




(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

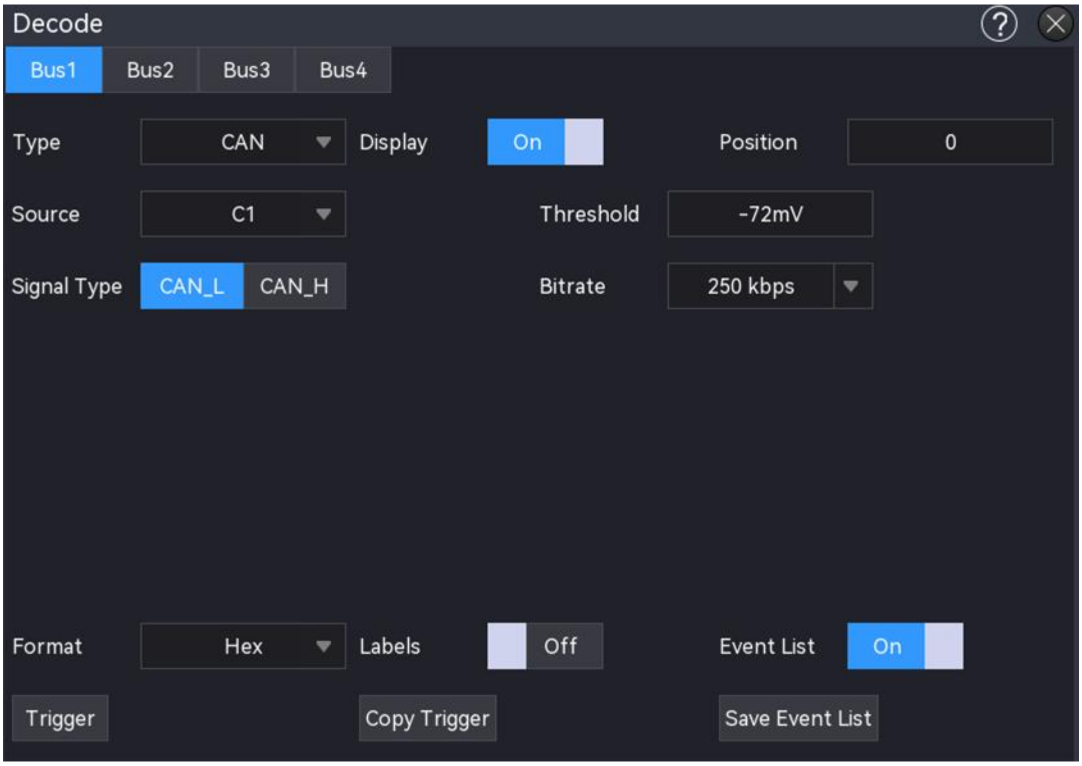
When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

## 9.5. CAN Decoding

(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select CAN and configure the decoding settings.




b. Source

Click Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold. When the threshold is changed, a dotted line appears on the screen to indicate the current trigger level. After the adjustment stops, the dotted trigger level line disappears automatically after approximately 2 seconds.

d. Signal Type

Select whether the current signal accessed by the source is a high data line signal or a low data line signal. Click Signal Type to select CAN_H, or CAN_L.

e. Bitrate

Select the bitrate for CAN serial bus data. Click Bitrate to select 10 kbps, 19.2 kbps, 20 kbps, 33.3 kbps, 38.4 kbps, 50 kbps, 57.6 kbps, 62.5 kbps, 83.3 kbps, 100 kbps, 115.2 kbps, 125 kbps, 230.4 kbps, 250 kbps, 490.8 kbps, 500 kbps, 800 kbps, 921.6 kbps, 1 Mbps, 2 Mbps, 3 Mbps, 4 Mbps, 5 Mbps, or custom.

If Custom is selected, a custom bitrate can be entered.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. The display position of decoding line can be adjusted by using Multipurpose A rotary knob. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, binary, or ASCII.

d. Label

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.


(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

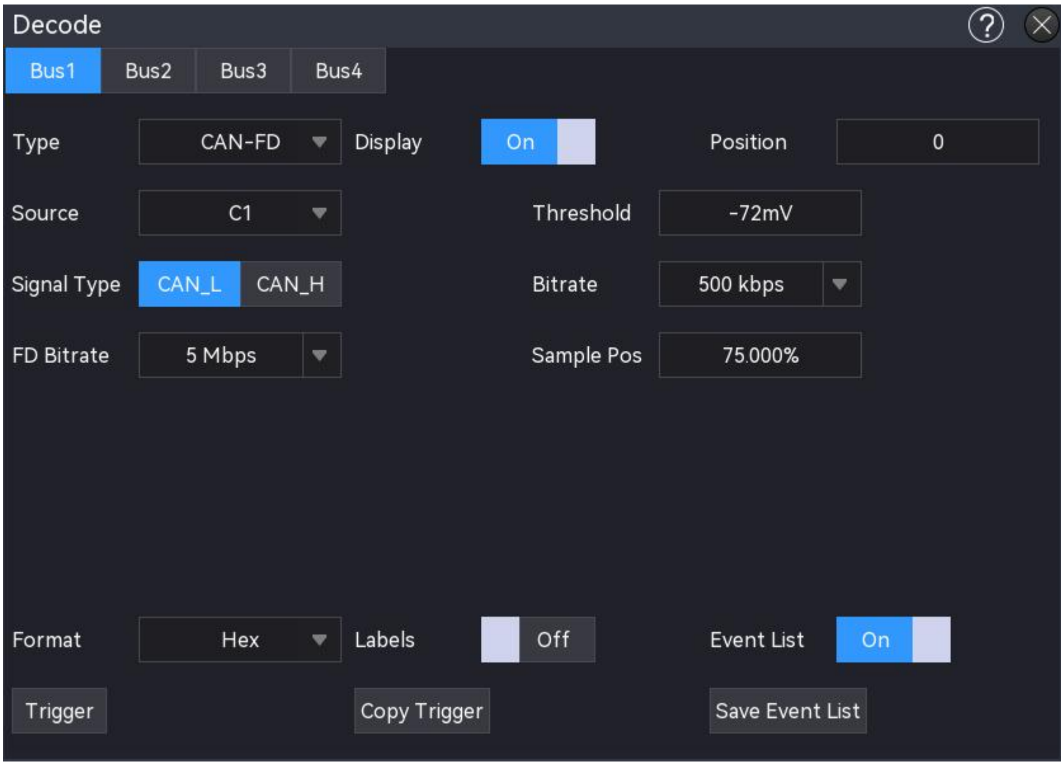
When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

## 9.6. CAN-FD Decoding

(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select CAN-FD and configure the decoding settings.




b. Source

Click Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold. When the threshold is changed, a dotted line appears on the screen to indicate the current trigger level. After the adjustment stops, the dotted trigger level line disappears automatically after approximately 2 seconds.

d. Signal Type

Select whether the current signal accessed by the source is a high data line signal or a low data line signal. Click Signal Type to select CAN_H or CAN_L.

e. Bitrate

Select the bitrate for CAN-FD serial bus data. Click Bitrate to select 10 kbps, 19.2 kbps, 20 kbps, 33.3 kbps, 38.4 kbps, 50 kbps, 57.6 kbps, 62.5 kbps, 83.3 kbps, 100 kbps, 115.2 kbps, 125 kbps, 230.4 kbps, 250 kbps, 490.8 kbps, 500 kbps, 800 kbps, 921.6 kbps, 1 Mbps, 2 Mbps, 3 Mbps, 4 Mbps, 5 Mbps, or custom.

If Custom is selected, a custom bitrate can be entered.

f. FD Bitrate

Select the FD bitrate for CAN-FD serial bus data, click Bitrate to select 250 kbps, 500 kbps, 800 kbps, 1 Mbps, 1.5 Mbps, 2 Mbps, 4 Mbps, 6 Mbps, 8 Mbps or custom.

g. Sampling Position

The sample position is the point in the bit time where the oscilloscope samples the bit level. The sample position is expressed as a percentage of the "time from bit start to sample point" and the "bit time".

Click Sampling Position input field to open the numeric keypad and set the sampling position. The range can be set from 30% to 90%.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, binary, or ASCII.

d. Label

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

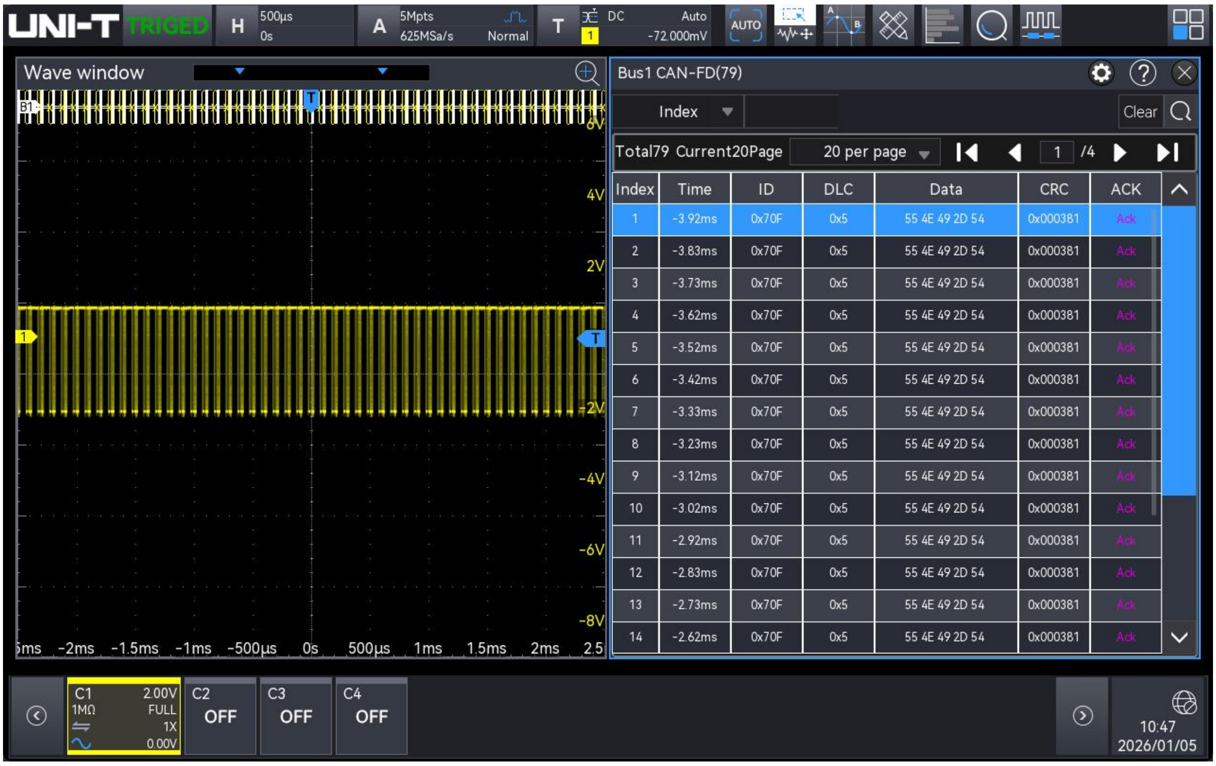
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.



(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

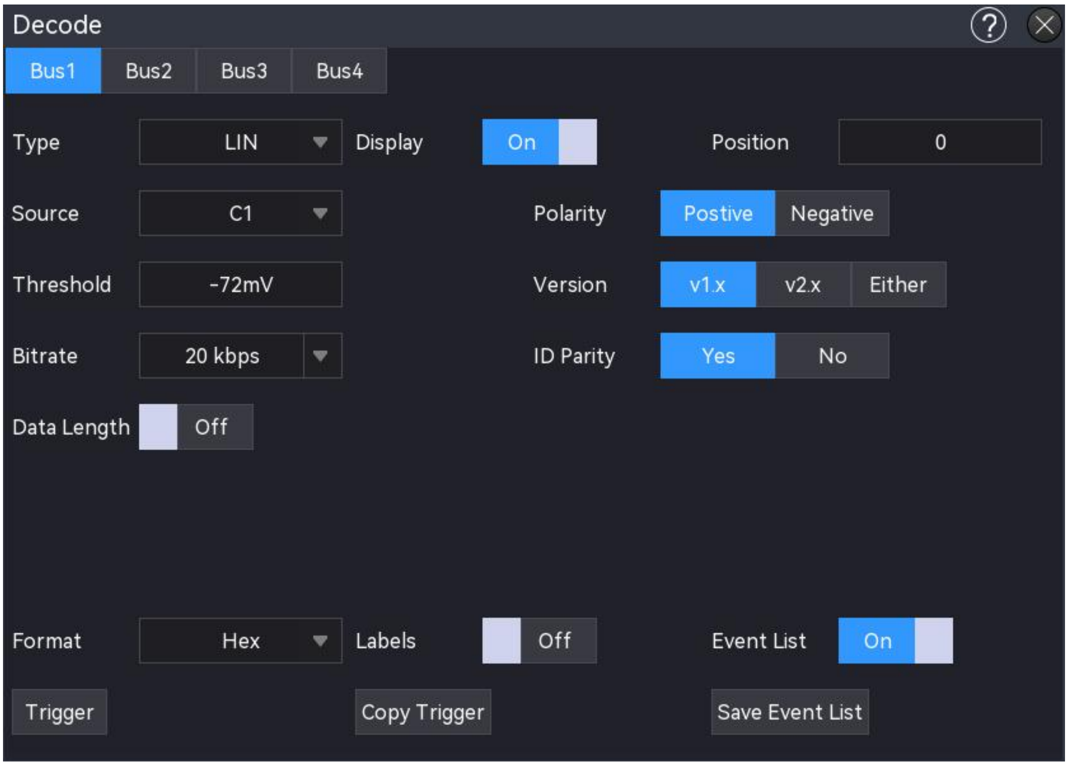
When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

## 9.7. LIN Decoding

(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select LIN and configure the decoding settings.




b. Source

Click Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. Polarity

Click Polarity to select the polarity to positive or negative.

d. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold. When the threshold is changed, a dotted line appears on the screen to indicate the current trigger level. After the adjustment stops, the dotted threshold line disappears automatically after approximately 2 seconds.

e. Version

Click Version to select the signal version to v1.x, v2.x, or arbitrary.

f. Bitrate

Select the bitrate for LIN, click Bitrate to select 1.2 kbps, 2.4 kbps, 4.8 kbps, 9.6 kbps, 10.417 kbps, 19.2 kbps, 20 kbps, or custom.

If Custom is selected, a custom bitrate can be entered.

g. ID Parity check

Set ID Parity check to turn it ON or OFF.

ON: Includes the parity bit and ID.

OFF: Excludes the parity bit and ID.

h. Data Length Menu

Set whether to display the data length menu, click Data Length to turn it ON or OFF.

ON: Displays the data length menu.

OFF: Hides the data length menu.

i. Data Length

Set LIN data length, double-click Data Length input field to open the numeric keypad and set the data length. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The data length can be set from 1 to 8. It is only available when the data length menu is set to ON.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Label

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

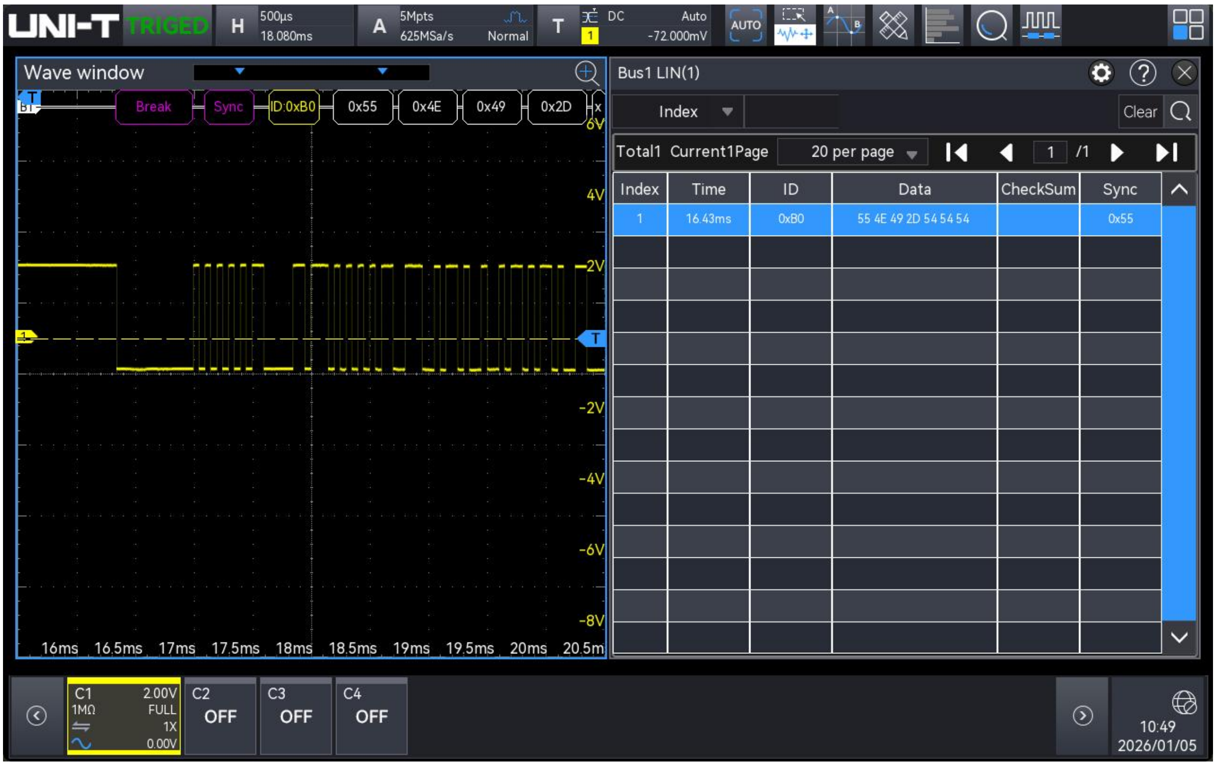
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.




(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

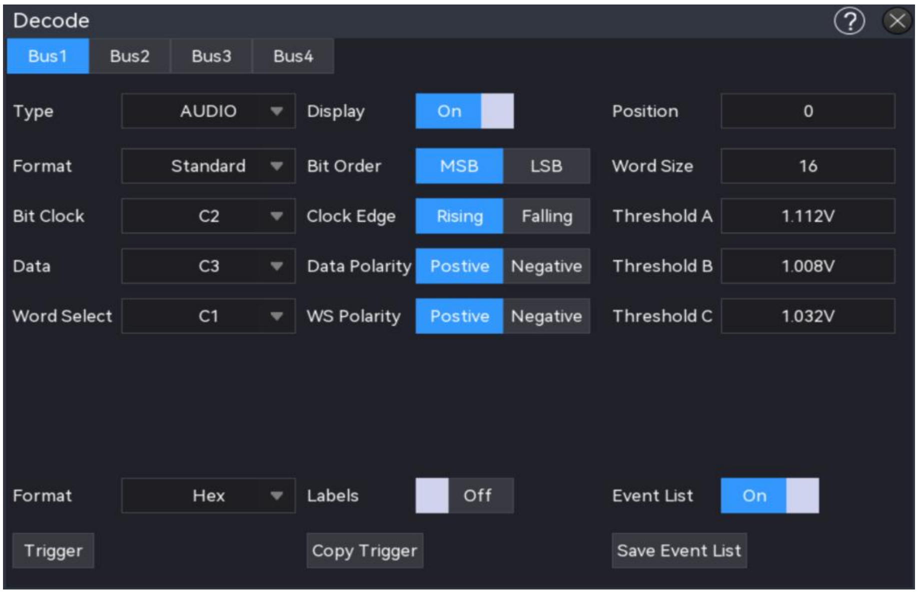
## 9.8. Audio Decoding

Audio (Inter—IC Sound) bus, also known as the Integrated Circuit Built-in Audio Bus, is a bus standard developed by Philips for the transmission of audio data between digital audio devices.

(1) Decoding Menu Setting

a. Protocol Type

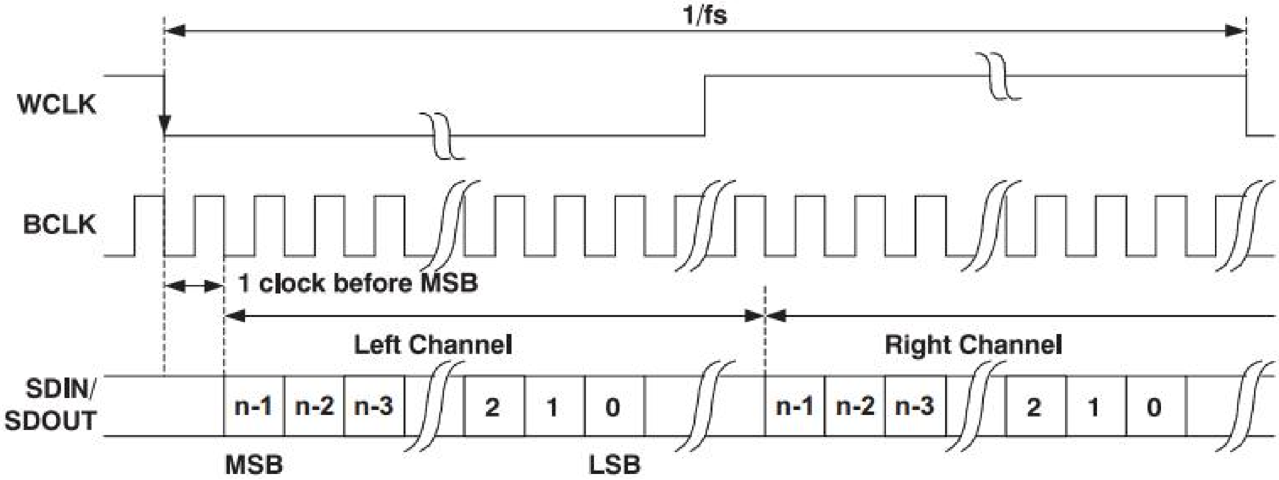
Click Protocol Type to select Audio and configure the decoding settings.



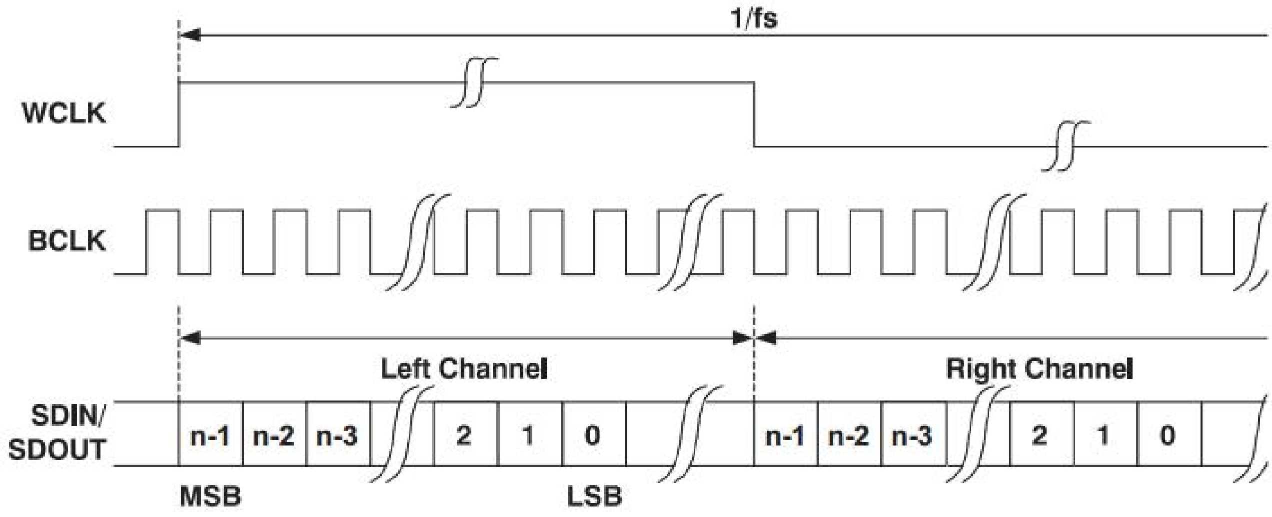

b. Format

Click Format to select standard, left justifying, right justifying, or TDM.

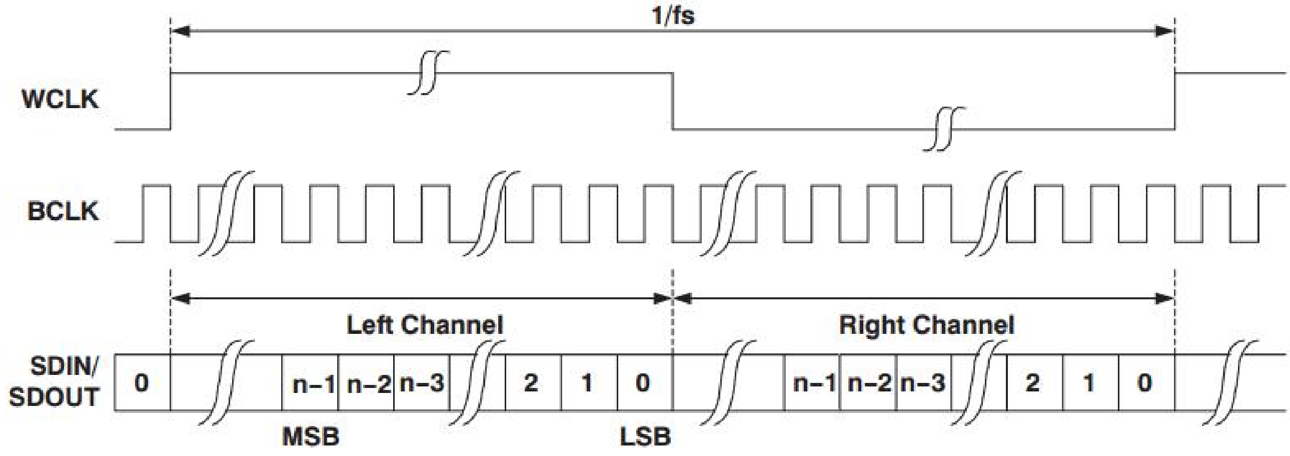
- Standard: MSB of each sampled data is sent first and the LSB is sent last. MSB is displayed on the SDATA line, which at one clock bit clock after the edge of WS transition.



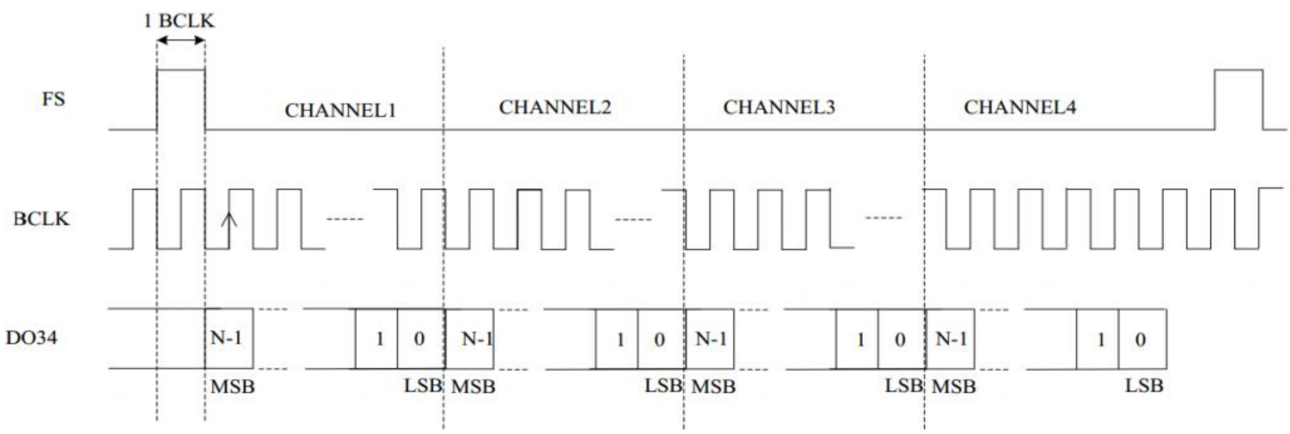

- Left justifying: Data transmission (MSB first) starts at the edge of the WS conversion (without the one-bit delay used by the standard format).




- Right justifying: Data transmission (MSB first) is right justifying with WS.




- TDM: (Time Division Multiplexing) mode can transmit multi-channel data.




c. Bit Order

Click Bit Order to select LSB or MSB. The default is MSB.

d. Source Setting

Set the bit clock, bit selection, and data source. Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

- Bit Clock

Click Bit Clock to select C1-C4. For more details on *Trigger Source,* refer to the section

Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen. The clock line (SCLK) provides the clock signal for synchronizing audio data transmission.

- Bit Selection

  Click Bit Selection to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

  The bit selection indicates the audio data of the current transmission is left channel or right channel.

- Data

  Click Data to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen. The data line is used to transmit the actual audio data.

- Frame Synchronization

  Click Frame Synchronization to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the trigger label at the top of the screen.

e. Edge Setting

- Clock Edge

  Click Clock to select Rising/Falling edge.

  Rising edge: Samples SDA on the rising edge of clock.

  Falling edge: Samples SDA on the falling edge of clock.

- WS Polarity

  Click WS Polarity to select Normal or Reverse. The WS polarity determines the valid level for the bit selection signal. The bit selection signal indicates the start frame and end of frame for the audio data.

- Data Polarity

  Click Data Polarity to select High=1 or High=0.

- Polarity Synchronization

  Click Polarity Synchronization to set the edge for signal synchronization. It can be set to Rising or Falling edge.

f. Threshold

Double-click Threshold A-Threshold D input field to open the numeric keypad and set the corresponding threshold. For details on using the numeric keypad, refer to section 5.7

Parameter Setting.

g. Bit Size

The bit size can be set when the format is standard, left justifying or right justifying. Double-click Bit Size input field to open the numeric keypad and set the bit size. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 4 to 32.

h. Data Bit of Each Channel

Double-click Data Bit of Each Channel input field to open the numeric keypad to set this value. For details on the use of the numeric keyboard, refer to the section of 5.7 Parameter Setting. The range of data bit can be set from 4 to 32.

Data bit setting for each channel <= Clock bit setting for each channel

i. Clock Bit of Each Channel

Double-click Clock Bit of Each Channel input field to open the numeric keypad to set this value. For details on the use of the numeric keyboard, refer to the section of 5.7 Parameter Setting. The range of clock bit can be set from 4 to 32.

j. Frame of Each Channel

Double-click Frame of Each Channel input field to open the numeric keypad to set this value. For details on the use of the numeric keyboard, refer to the section of 5.7 Parameter Setting. The range of frames can be set from 2 to 64.

k. Bit Delay

Double-click Bit Delay input field to open the numeric keypad to set this value. For details on the use of the numeric keyboard, refer to the section of 5.7 Parameter Setting. The range of bit delay can be set from 0 to 31.

Bit delay < Clock bit setting for each channel

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to set the position. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Label

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

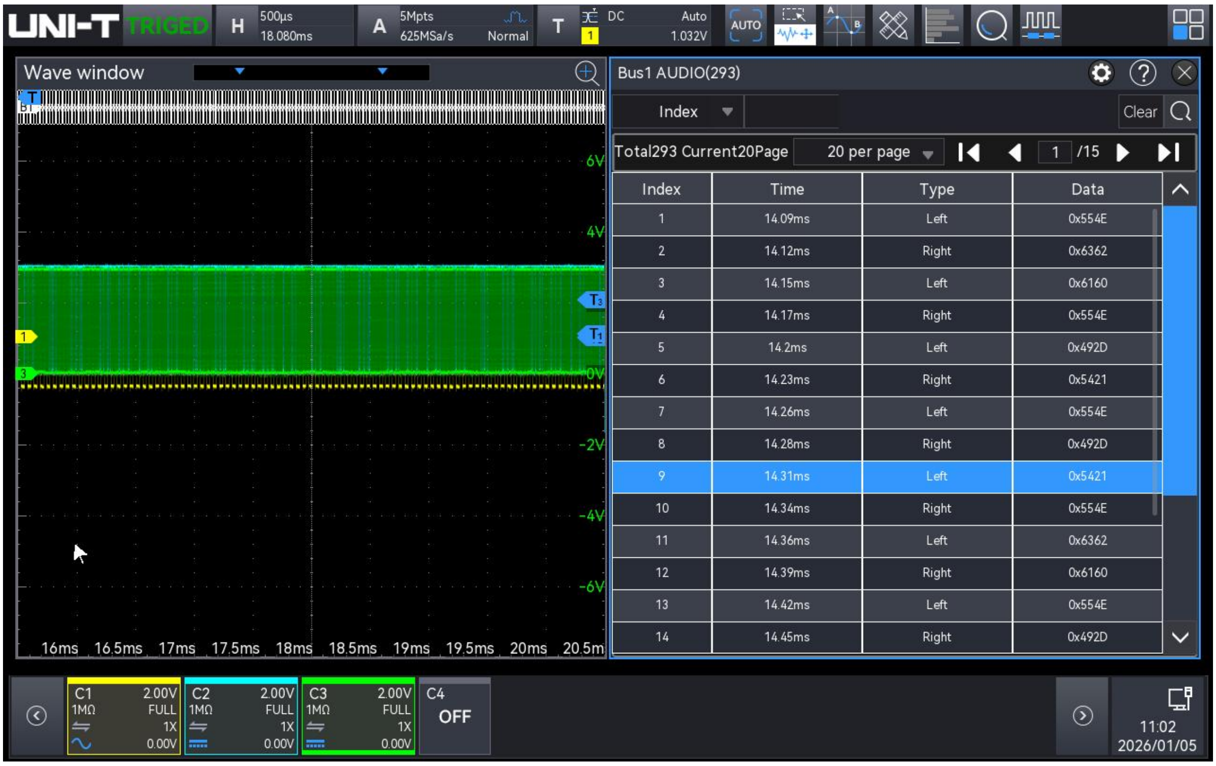
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.




(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

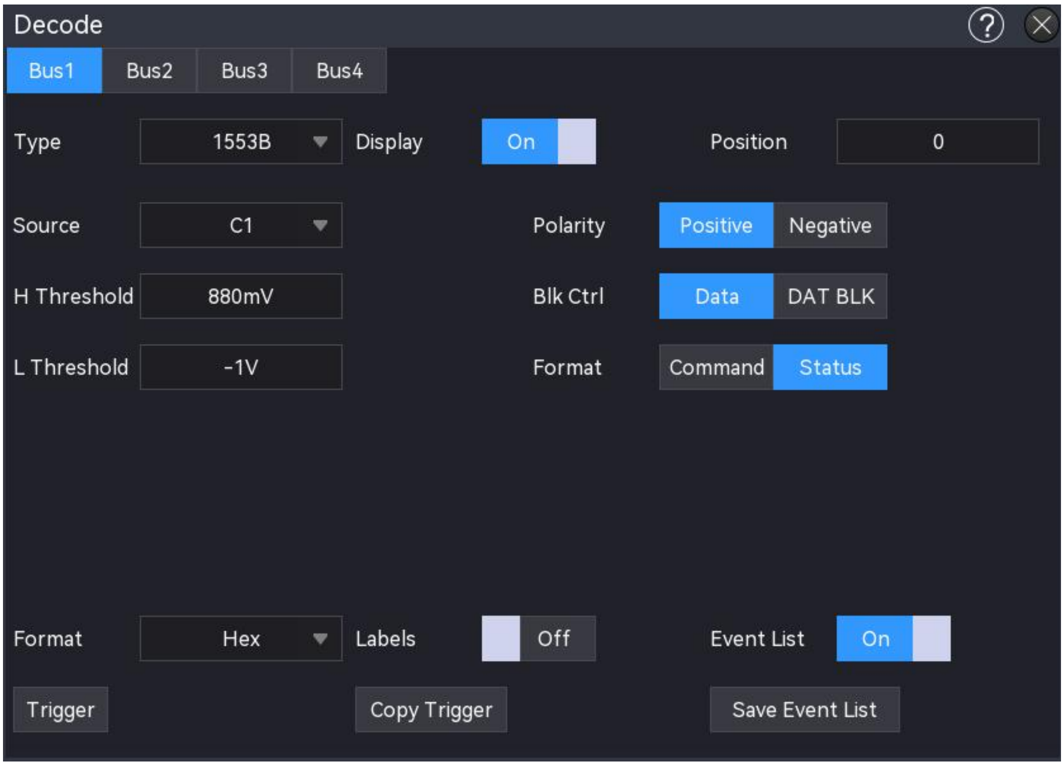
## 9.9. 1553B Decoding

The oscilloscope samples the 1553B signal and determines whether each data point is a logic "1" or "0" based on a set threshold. 1553B decoding requires the data channel source and threshold to be specified.

(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select 1553B and configure the decoding settings.




b. Source

Click Source to select C1-C4 as the trigger source. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. Polarity

Click Polarity to select Positive or Negative. The default is positive.

d. High/Low Threshold

Double-click Threshold (High/Low) input field to open the numeric keypad and set the threshold. The threshold range is related to the vertical scale and vertical offset of the source.

e. Block Control

Click Block Control to select Data or Data block to decoding. The default is data.

f. Format

Click Format to set command word or state word.

If the format is the command word, the "state" trigger condition will be hidden.

If the format is the state word, the "command" trigger condition will be hidden.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to set the position. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Label

Click Label to turn the decoding bus label on or off. ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type. OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off. ON: The event list is displayed, as shown in the following figure. OFF: Click close × icon in the upper-right corner of the screen to close the event list.


(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

## 9.10. Manchester Decoding

Manchester encoding, also called phase encoding (PE), is a synchronous clock coding technique used by the physical layer to encode the clock and data of a synchronous bit stream. Manchester encoding is used in Ethernet media systems.

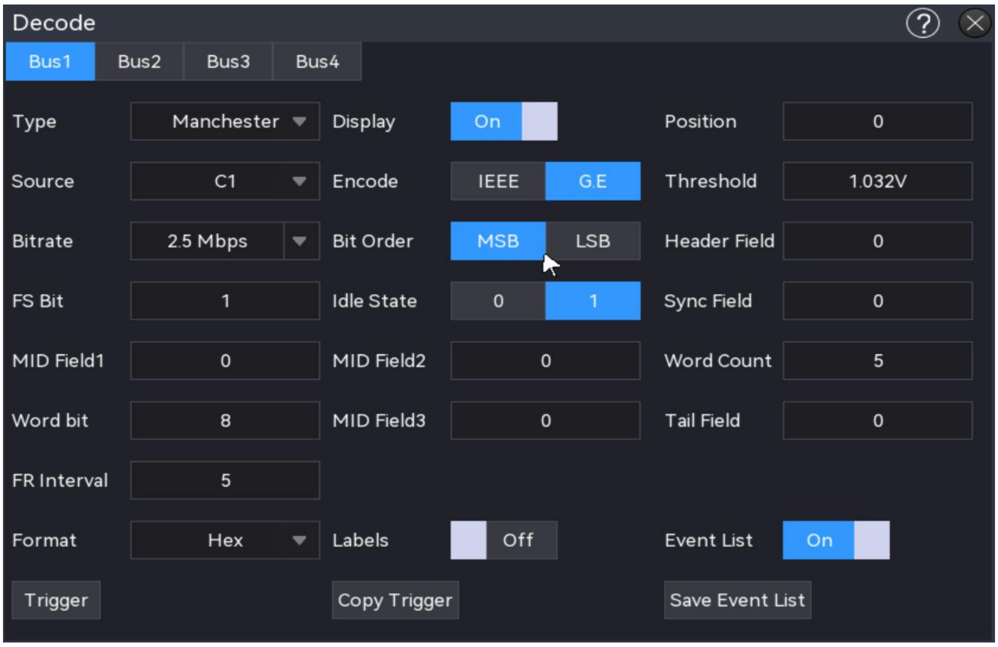
Manchester encoding provides a simple way of encoding simple binary sequences without long cycles and without conversion levels, thus preventing loss of clock synchronization or analogue link bit errors due to low frequency shifts with poor compensation. In this technique, the actual binary data transmitted through the cable is not sent as a sequence of logical 1's or 0's (technically known as Non-Return to Zero (NRZ). Instead, these bits are converted into a slightly different format, which has many advantages than the use of binary coding.

Manchester coding is commonly used in LAN transmission. Manchester coding is used to encode binary data '0' and '1' by level jumps.

(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select Manchester and configure the decoding settings.




b. Source

Click Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold.

d. Encode Mode

Click Encode Mode to switch between IEEE and G.E.

- IEEE: 1 indicates a low-to-high transition. 0 indicates a high-to-low transition.
- G.E: 0 indicates a low-to-high transition. 1 indicates a high-to-low transition.

e. Bitrate

Click Bitrate to select the baud rate of DUT to 1.2 kbps, 2.4 kbps, ps4.8 kbps, 9.6 kbps, 10.417 kbps, 19.2 kbps, 125 kbps, 250 kbps, 200 kbps, 1 Mbps, 2 Mbps, 5 Mbps, 10 Mbps, or custom. The custom bitrate must match with the DUT, the default bitrate is 1.2 kbps.

f. Bit Order

Click Bit Order to switch between MSB and LSB.

- MSB: The Most Significant Bit, i.e., the bit with the highest value in a sequence, transmitted first.
- LSB: The Least Significant Bit, i.e. the bit with the least value in a sequence, transmitted first.

g. Idle State

Click Idle State to switch between 0 or 1.

- 0: The bus state is at a low level when no data is present.
- 1: The bus state is at a high level when no data is present.

h. Start Frame Bit

Click Start Frame Bit input field to open the numeric keypad and set the start frame bit. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The range can be set from 1 to 32.

i. Synchronization Field

Click Synchronization Field input field to open the numeric keypad and set the synchronization field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

j. Middle Field 1

Click Middle Field 1 input field to open the numeric keypad and set the middle field 1. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

k. Header Field

Click Header Field input field to open the numeric keypad and set the header field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

l. Middle Field 2

Click Middle Field 2 input field to open the numeric keypad and set the middle field 2. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.The setting range is from 0 to 32.

m. Word Count

Click Word Count input field to open the numeric keypad and set the data bit. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 255.

n. Word Bit

Click Word Size input field to open the numeric keypad and set the bit size. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 8.

o. Middle Field 3

Click Middle Field 3 input field to open the numeric keypad and set the middle field 3. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

p. End Field

Click End Field input field to open the numeric keypad and set the end field. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

q. Inter-frame Space

Click Inter-frame Space input field to open the numeric keypad to enter the inter-frame space. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 to 32.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to set the position. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Label

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

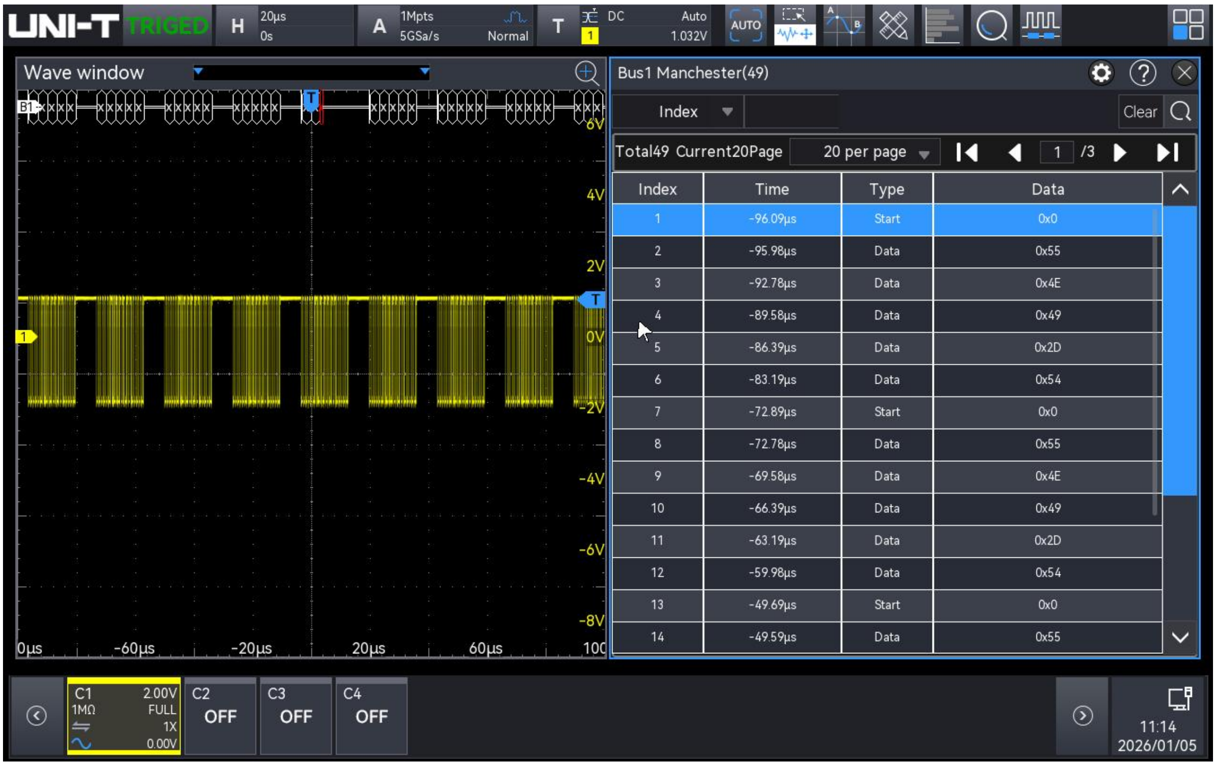
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.




(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

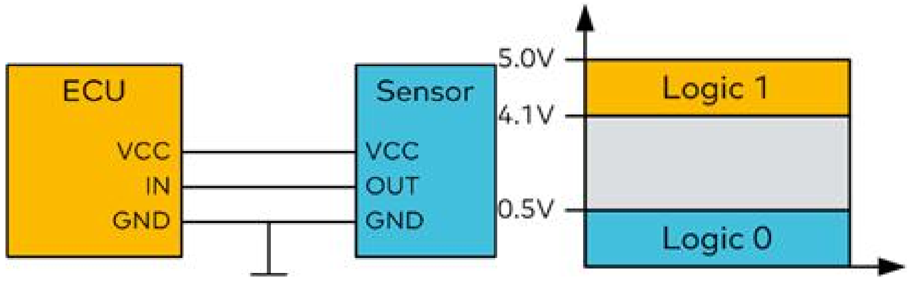
(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

## 9.11. SENT Decoding

SENT (Single Edge Nibble Transmission) protocol is a point-to-point, unidirectional transmission scheme introduced by SAE, which is used for data transmission between on-board sensors and electronic control units (ECUs).

SENT high and low signal level requirements: 0-0.5 V for logic level 0, 4.1-5 V for logic level 1.

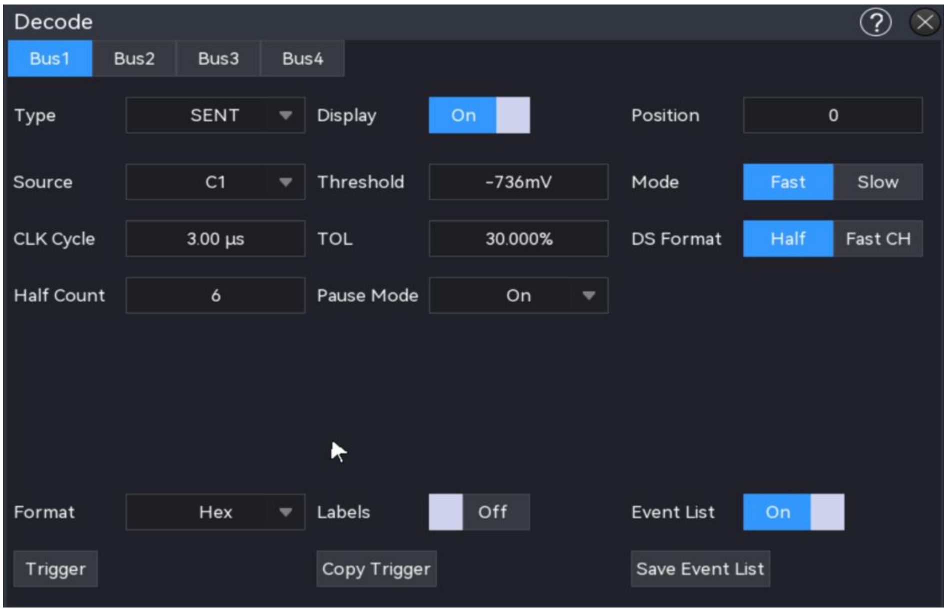



The data of the SENT protocol is coded using a half-byte nibble, i.e. 4 bits, and a half-byte nibble is defined by the time difference between two falling edges.

(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select SENT and configure the decoding settings.




b. Source

Click Source to select C1-C4. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold.

d. Mode

Click Mode to switch the trigger signal mode to fast or slow.

e. Clock Period

Double-click Clock Period to open the numeric keypad and set the clock period. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 500 ns to 300 μs.

f. Tolerance

Set the percentage tolerance to specify a percentage tolerance for determining whether the sync pulse is valid for decoding data. If the time of the measured sync pulse is within the percentage tolerance of the rated clock period, then the decoding will continue, otherwise, the sync pulse occurs an error and data decoding will not be performed.

g. Half Byte

Set the half byte for fast channel message, double-click Half Byte input field to open the numeric keypad and set the half byte. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

h. Pause Mode

Click Pause Mode to set whether there is a pause pulse between the fast channel messages. It can be switched to ON or OFF.

- OFF: There is no pause pulse between the fast channel messages.

  There is no idle time on the SENT serial bus without pause pulses. This means that during normal operation the fast channel decode line shows a continuous stream of packets, i.e. one packet closes, and a new packet opens immediately.
- ON: Inserts a pause pulse between fast channel messages, ensuring that the frames arrive at equal intervals.

  When the pause pulse is enabled, the idle time is displayed between messages.

i. Data Field Format

Set the display format of decoding data field. It can be set to half byte or fast channel.

- Half byte: The decoding data of data field is displayed in half byte.
- Fast channel: The decoding data of data field is displayed together.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to set the position. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Format

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

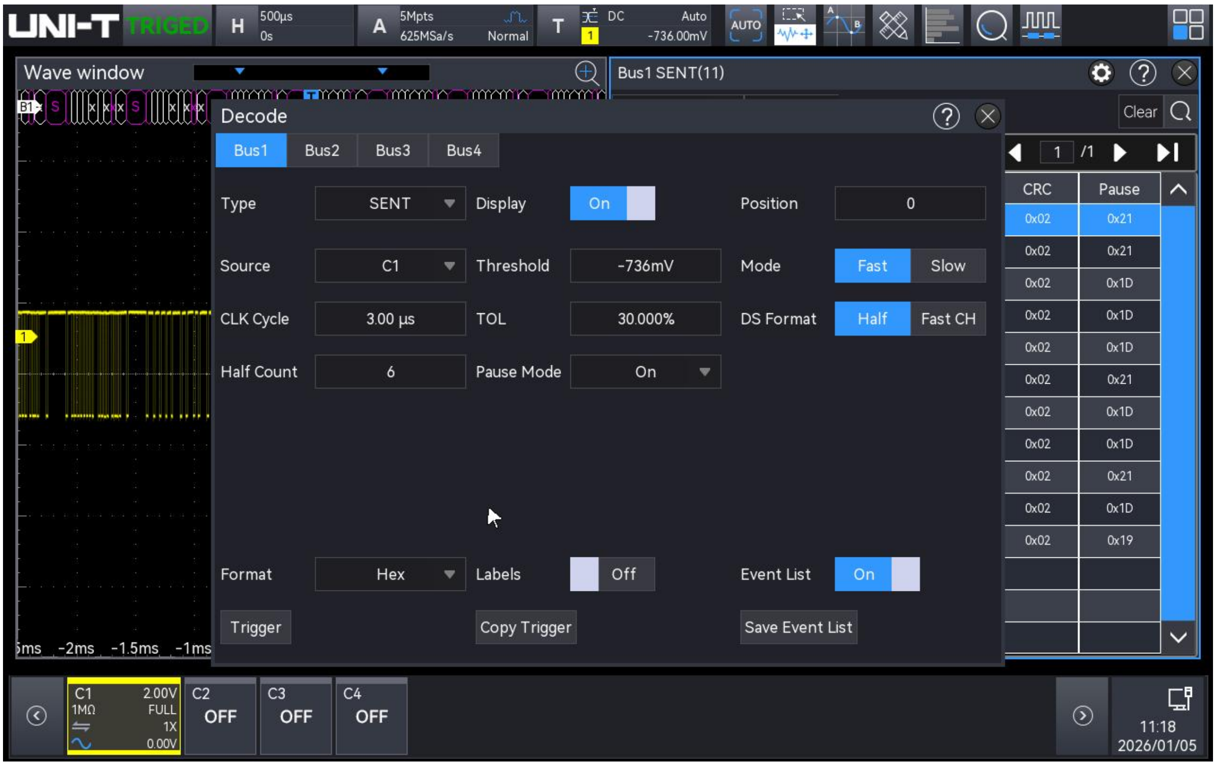
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.




(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

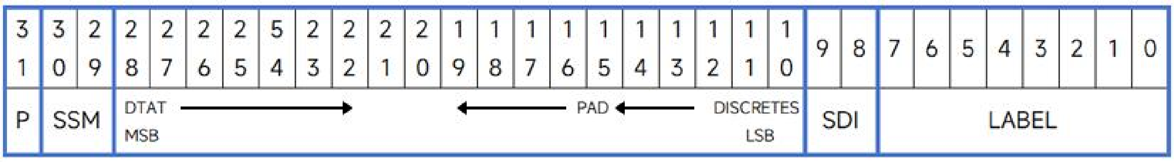
Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

# 9.12. ARINC429 Decoding

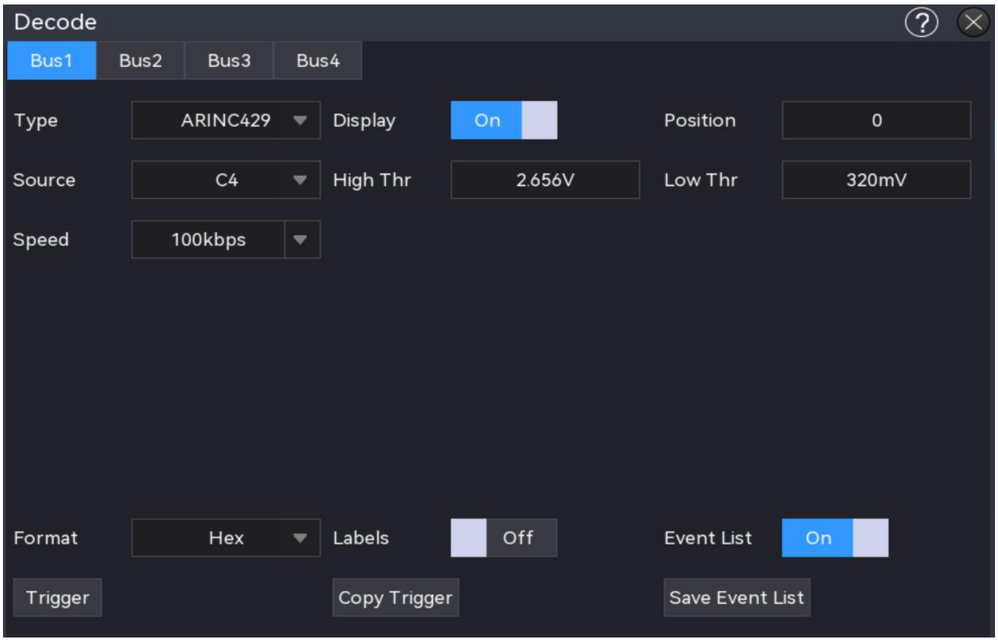
The ARINC 429 bus is a serial standard, interface-oriented, unidirectional broadcast transmission bus.




(1) Decoding Menu Setting

a. Protocol Type

Click Protocol Type to select ARINC429 and configure the decoding settings.




b. Source

Click Source to select C1-C45. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current trigger source is displayed in the trigger label at the top of the screen.

**Note**: Decoded data can be obtained only when the signal source is set to the channel with a connected signal.

c. High/Low Threshold

Double-click Threshold (High/Low) input field to open the numeric keypad and set the threshold.

d. Speed

Click Speed to set the transmission rate to high (100 kb/s), low (12.5 kb/s), or custom.

(2) Decoding Bus Setting

a. Bus Switch

Click Bus Switch to turn the bus function on or off.

b. Decoding Line

Set the display position of decoding bus on the screen. Double-click Decoding Line input field to set the position. The setting range is from 0 to 546.

c. Format

Set the display format for the decoding bus and event list decoding. Click Format to select hexadecimal, decimal, or binary.

d. Format

Click Label to turn the decoding bus label on or off.

ON: The decoding bus label is displayed in the upper-left corner and shows the current protocol type.

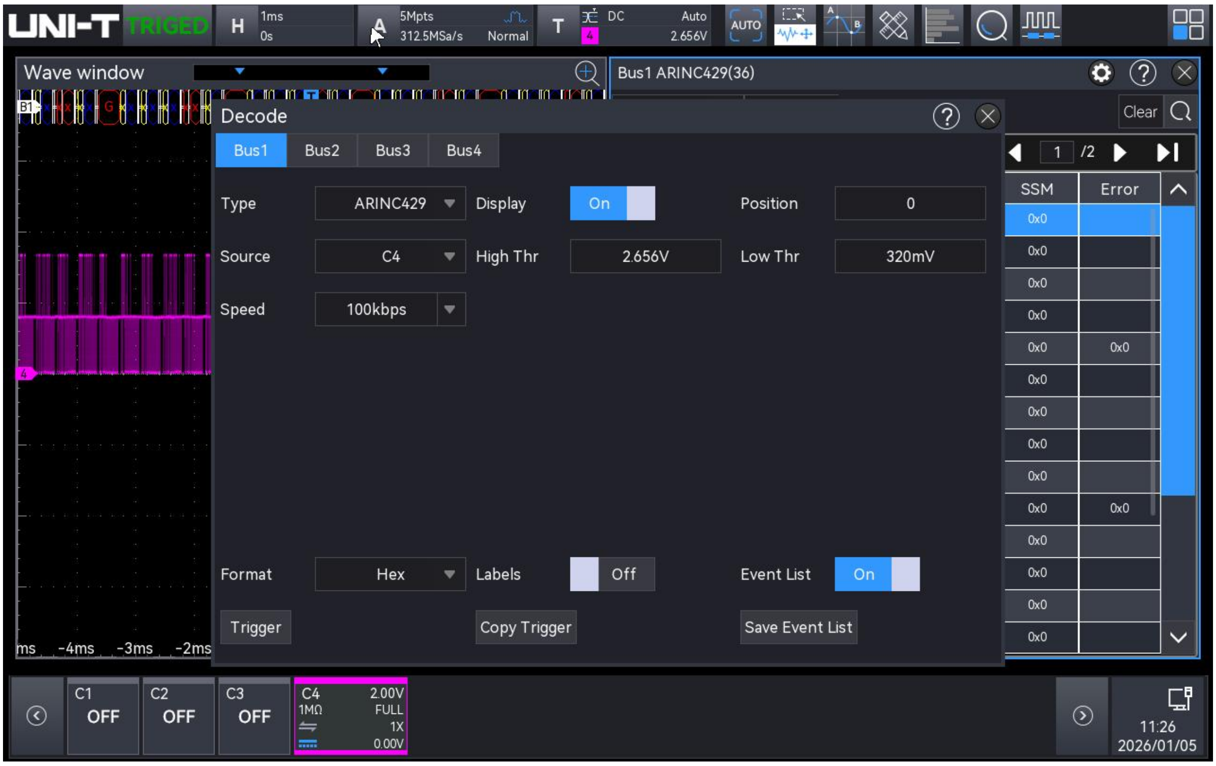
OFF: The decoding bus label is hidden.

(3) Event List

Click Event List to turn the event list on or off.

ON: The event list is displayed, as shown in the following figure.

OFF: Click close × icon in the upper-right corner of the screen to close the event list.



(4) Save Event List

When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

Click Save Event List in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB flash drive is detected). For the setting steps, refer to Save and Load section.

**Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

(5) Jump to Trigger Menu

Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(6) Copy Trigger Settings

When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

# 10. Automatic Measurement



The UPO6000LP series measurement menu provides access to all parameter measurement functions, including parameter snapshots, custom parameters, parameter statistics, frequency counters, digital voltmeters, and global settings for parameter measurements.

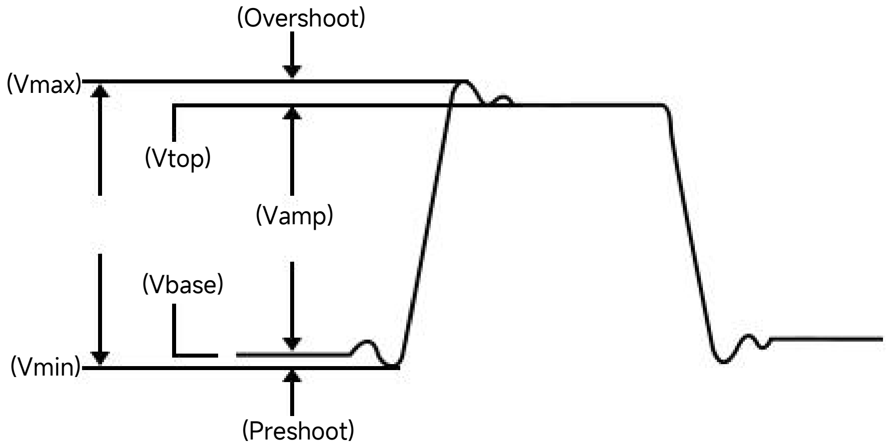
Access the Measure menu using the following method.

- Click Home icon in the top right-corner and select the measurement icon to enter the Measure menu.
- If the measurement is added to the toolbar, click measurement icon in the toolbar in the top right-corner to enter the Measure menu.

## 10.1. Parameter Measurement Overview

The UPO6000LP series oscilloscope can automatically measure 56 types of parameters, such as voltage, time, other parameters, and power analysis.

- **Voltage Parameter**




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

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

Top (Vtop): The voltage value from the flat top of the waveform to GND.

Bottom (Vbase): 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

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

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

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

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

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

AC RMS of cycle: Standard deviation of voltage value of waveform data in one cycle, which DC component has removed.

Area: The algebraic sum of the product of voltage and time for all points on the screen

Cycle area: The algebraic sum of the product of the voltage and the time at all the points in a cycle of the waveform.

Positive area: The algebraic sum of the product of all voltages and times on the screen greater than GND (ground).

Negative area: The algebraic sum of the product of all voltages and times on the screen less than GND (ground).

Positive cycle area: The algebraic sum of the product of all voltages and times greater than GND (ground) in a cycle.

Negative cycle area: The algebraic sum of the product of all voltages and times less than GND (ground) in a cycle.

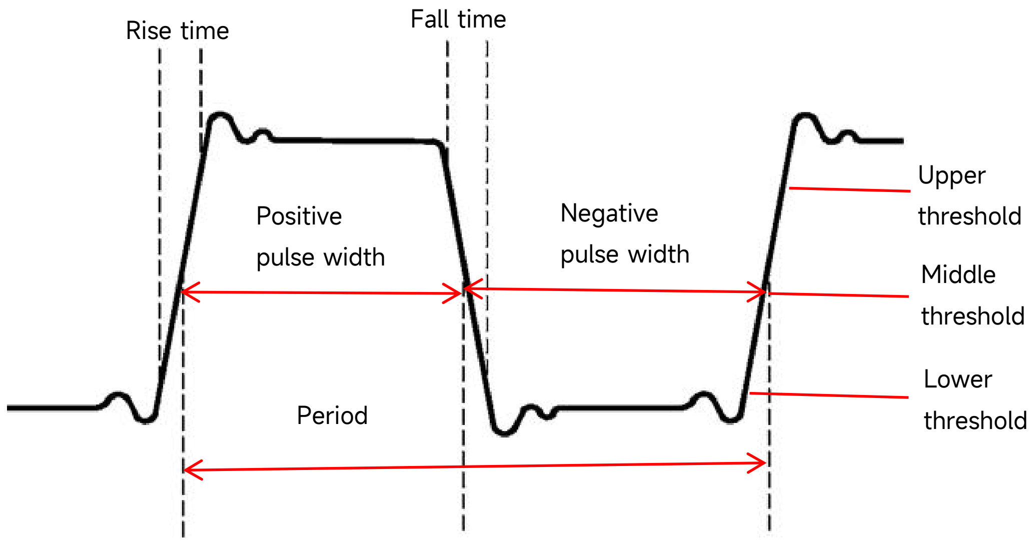
Positive overshoot: The nearest extreme point after the signal along the rising edge of the waveform crosses the upper threshold limit.

Negative overshoot: The nearest extreme point after the signal along the falling edge of the waveform crosses the lower threshold limit.

Positive pre-shoot: The nearest extreme point before the signal along the rising edge of the waveform crosses the upper threshold limit.

Negative pre-shoot: The nearest extreme point before the signal along the falling edge of the waveform crosses the lower threshold limit.

## ■ Time Parameter




Period: The time between two consecutive, homopolar edges of a repetitive waveform with the same threshold median crossing point.

Frequency: The reciprocal of the cycle

Rise time: Time needed for waveform amplitude rising from the lower threshold to the upper threshold.

Fall time: Time needed for waveform amplitude rising from the upper threshold to the lower threshold.

Positive pulse width: The time difference between the time at the middle threshold on the rising edge of the pulse and the time at the middle threshold on the falling edge of the pulse immediately following.

Negative pulse width: The time difference between the time at the middle threshold on the falling edge of the pulse and the time at the middle threshold on the rising edge of the pulse immediately following.

Positive duty ratio: The ratio of the positive pulse width to period.

Negative duty ratio: The ratio of the negative pulse width to period.

Positive pulse number: The number of positive pulses from the lower threshold to the upper threshold.

Negative pulse number: The number of negative pulses from the upper threshold to the lower threshold.

Rising edge count: The number of rising edges from the lower threshold to the upper threshold.

Falling edge count: The number of rising edges from the upper threshold to the lower threshold.

Burst width: Length of time that the intermediate reference level is exceeded more than once in a row.

Burst interval: The interval between two burst events.

Burst period: Burst period that satisfy burst width and burst interval.

Burst period count: The number that satisfies burst period.

- **Other Parameters**

Ratio: The ratio of the AC effective voltages of the master and slave sources, expressed in dB.

Period ratio: The ratio of the periodic AC RMS voltages of the master and slave sources, expressed in dB.

Setup time: The time from when the specified intermediate reference level on the data source was exceeded to the last time the specified intermediate reference level on the clock source was exceeded.

Hold time: The time from when the specified intermediate reference level on the clock source was exceeded to the last time the specified intermediate reference level on the data source was exceeded.

Setup and Hold ratio: The ratio of the total time of setup time and hold time

FRFR: Time at the middle threshold intersection from the first rising edge of source 1 to the first rising edge of source 2.

FRFF: Time at the middle threshold intersection from the first rising edge of source 1 to the first falling edge of source 2.

FFFR: Time at the middle threshold intersection from the first falling edge of source 1 to the first rising edge of source 2.

FFFF: Time at the middle threshold intersection from the first falling edge of source 1 to the first falling edge of source 2.

FRLF: Time at the middle threshold intersection from the first rising edge of source 1 to the last falling edge of source 2.

FRLR: Time at the middle threshold intersection from the first rising edge of source 1 to the last rising edge of source 2.

FFLR: Time at the middle threshold intersection from the first falling edge of source 1 to the last rising edge of source 2.

FFLF: Time at the middle threshold intersection from the first falling edge of source 1 to the last falling edge of source 2.

Phase (r-r): The phase offset between the rising edge of the master source and the rising edge of the slave source at the median waveform threshold, expressed in degrees.

Phase (f-f): The phase offset between the falling edge of the master source and the falling edge of the slave source at the median waveform threshold, expressed in degrees.

Delay (r - r): The delay time between the rising edge of the main signal source and the rising edge of the slave signal source, measured at the median threshold value of the waveform.

Delay (f - f): The delay time between the falling edge of the main signal source and the falling edge of the slave signal source, measured at the median threshold value of the waveform.

- **Power Analysis**

- Power quality parameters

Voltage peak: The maximum value of the voltage waveform.

Voltage RMS: The root mean square (RMS) value of the voltage waveform.

Peak voltage factor: The ratio of the peak value to the RMS value of the voltage waveform.

Current peak: The maximum value of the current waveform.

Current RMS: The root mean square (RMS) value of the current waveform.

Current peak factor: The ratio of the peak value of the current waveform to its RMS value.

Active power: The power actually consumed by the load, measured in watts (W).

Reactive power: The power due to inductance and capacitance in a circuit, measured in volt-amperes reactive (var).

Apparent power: The product of the RMS values of voltage and current, measured in volt-amperes (VA).

Power factor: The ratio of active power to apparent power.

Power phase angle: The angle between active power and apparent power, measured in degrees (°).

- Surge current parameters

Surge current: The peak current flows into the power supply equipment when the power supply is turned on.

- Rds (on) Parameters

Rds(on): The effective resistance of the power switching device during its conducting stage.

- Switching Loss Parameters

Number of switching cycles: Total count of switching cycles that meet predefined criteria.
Turn-on power loss: Power dissipated during the turn-on phase of the switching device, measured in Watts (W).
Conduction power loss: Power dissipated during the conduction phase, measured in Watts (W).
Turn-off power loss: Power dissipated during the turn-off phase of the switching device, measured in Watts (W).
Total power loss: Combined power loss across the entire switching cycle, measured in Watts (W).
Turn-on energy loss: Energy consumed during the turn-on phase, measured in Joule (J).
Conduction energy loss: Energy consumed during the conduction phase, measured in Joule (J).
Turn-off energy loss: Energy consumed during the turn-off phase, measured in Joule (J).
Total energy loss: Total energy consumption over the full switching cycle, measured in Joule (J).

- Slew Rate Parameters

dv/dt: The rate of change of voltage over time is analyzed during the switching process of a power switching device.
di/dt: The rate of change of current over time is analyzed during the switching process of a power switching device.

- Ripple Analysis Parameters

Peak-to-peak ripple value: The difference between the maximum and minimum values of the ripple waveform.
Root Mean Square (RMS) Ripple Value: The RMS value calculated from the ripple waveform.

- Startup/Shutdown Time Parameters

Starup time: The time it takes for the output voltage to rise to 90% of its steady-state value after the power supply is powered on.
Shutdown time: The time it takes for the output voltage to fall to 10% of its maximum value after the power supply is turned off.

- Transient Response

Transient response time: The time required for the output voltage to recover and stabilize within the allowable error range, starting from the moment it deviates from the steady-state value due to a sudden change in the load current at the power supply output.

- Efficiency

Input power: The actual power supplied to the power supply, measured in watts (W).

Output power: The actual power delivered by the power supply, measured in watts (W).

Efficiency: The ratio of output power to input power.

**Note**: Parameters in the power quality category can only be customized after the power quality Analysis is enabled.

## 10.2. Frequency Counter

The counter analysis function provides counting measurements of frequency, period or product on any analogue channel.

Access the Counter menu using the following method.

- Click Home icon in the top right-corner and select the counter icon to open the counter box to switch on the counter analysis function.
- Click counter icon in the toolbar in the top right-corner, to open the counter box to switch on the counter analysis function.

The results of count measurement are displayed above the volts/div box. Counter analysis can be used for multiple channels.


(1) Counter Setting

Once the counter is enabled, tap the counter parameter to open the counter box to set the display state, source, test type, refresh time, effective digit, and clear count.

a. Display State

Click Display to turn the counter display on or off.

ON: The result of count measurement will be displayed at the bottom of the screen.

b. Source

Click Source to select the test signal. C1-C4 and the trigger source can both serve as the source for the counter.

c. Test Type

Select the Frequency, Period, or Accumulation parameter to test. Accumulation counts the signal edge events.

d. Refresh Time

Set the refresh time for count measurement results. Double-click Refresh time input field to open the numeric keypad and set the time. The time range is from 200 ms to 10 s.

e. Effective Digit

The display resolution of the counter measurement results can be set when the parameter type is Frequency or Period. Double-click Effective Digit input field to open the numeric keypad and set the effective digit. The range of effective digits is from 3 to 7.

f. Clear Count

If the Accumulation is selected and the count of signal edge event is being measured, click Clear Count to delete the count results and restart the count measurement.

## 10.3. Digital Voltmeter

The built-in digital voltmeter (DVM) of this oscilloscope can measure 4 effective digits of voltage on any analogue channel. DVM measurements are asynchronous to the oscilloscope's acquisition system and are always acquired.

Access the DVM measurement menu using the following method.

- Click Home icon in the top right-corner and select the voltmeter icon  to open the voltmeter box to switch on the voltmeter measurement.
- Click voltmeter icon  in the toolbar in the top right-corner, to open the voltmeter box to switch on the voltmeter measurement.

The results of voltmeter measurement are displayed above the volts/div box. Voltmeter measurement can be used for multiple channels.


(1) Voltmeter Setting

Once the voltmeter is enabled, tap the voltmeter parameter to open the voltmeter window and set the display state, source, test type, refresh time, and beep.

a. Display State

Click Display to turn the voltmeter display on or off.

ON: The result of voltmeter measurement will be displayed at the bottom of the screen.

b. Source

Click Source to select the test signal. C1-C4 can be used as the trigger source. DVM measurement can be performed even if C1-C4 is not open.

c. Test Type

- DC: Displays the average of the collected data.
- AC RMS: Displays RMS of the collected data that the DC component has removed.
- DC+AC RMS: Displays RMS of the collected data.

d. Refresh Time

Set the refresh time for voltmeter measurement results. Double-click Refresh Time input field to open the numeric keypad and set the time. The time range is from 200 ms to 10 s.

(2) Output Setting

a. Aux Output

Click Aux Output and select either ON to enable or OFF to disable the DVM pulse output from the AUX connector.

- ON: The Auxiliary menu automatically sets the AUX output mode to DVM. When a DVM alarm is triggered, a pulse signal will be output from the [AUX OUT] connector on the rear panel.

- OFF: The Auxiliary menu automatically sets the AUX output mode to Trigger Output. The output from the [AUX OUT] connector is not linked to the DVM test.

b. Output Pulse Width

Double-click Output Pulse Width to open the numeric keypad and set the output pulse width. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The range of output pulse width can be set from 200 ns to 1.5 s.

c. Output Polarity

Click Output Polarity to set it to positive or negative.

(3) Beep Setting

Set the display state, limit condition, and the lower/upper limit for the beep.

a. Display State

Click Display to turn the beep on or off.

ON: The oscilloscope will have an alarm if the test result meets the condition, otherwise, the alarm will not be sound.

b. Limit Condition

- >: The oscilloscope will have an alarm if the DVM value is greater than the set lower limit, and the lower limit can be set.
- <: The oscilloscope will have an alarm if the DVM value is less than the set upper limit, and the upper limit can be set.
- < >: The oscilloscope will have an alarm if the DVM value is greater than the set lower limit and less than the set upper limit, and the upper/lower limit can be set.
- > <: The oscilloscope will have an alarm if the DVM value is less than the set lower limit and greater than the set upper limit, and the upper/lower limit can be set.

c. Upper/Lower Limit

The set voltage is compared to DVM value, and the range can be set from -500 V to 500 V.

- When the trigger condition is set to > or <, click Lower or Upper input field to open the numeric keypad and set the lower/upper limit.
- When the trigger condition is set to < > or > <, click Lower or Upper input field to open the numeric keypad and set the lower/upper limit. The lower limit should be less than the upper limit.

## 10.4. Parameter Snapshot

The parameter snapshot displays a single parameter that is automatically measured.

Select Parameter Snapshot in the Measure menu to open the parameter snapshot window.


In the parameter snapshot window, click Source to select the test signal (C1–C4, M1-M4).

The color of the measured results matches the color of the corresponding source.

## 10.5. Parameter Measurement

Select Parameter Measurement in the Measure menu to open the parameter measurement. The parameter measurement info window is not displayed if the parameter measurement is not enabled.


The parameter measurement is displayed above the volts/div info window, showing the measure parameter and the current value. During the measurement, the counter, voltmeter is displayed on the far left by default, and the custom parameter follow behind. The parameter measurement supports the setting of up to 27 parameters.

The custom parameter measurement in the parameter measurement info window can be cancelled by clicking "-" in the top right-corner.

## 10.6. Measurement Statistics

Select the Measurement Statistics in the Measure menu to open the measurement statistics. The results of all parameter measurements are displayed in the Parameter Measurement window at the bottom of the screen.

Statistics: current value, maximum, minimum, average value, standard deviation, count, line chart, and histogram.

Turn on the Chart switch under the measurement statistics and select the line chart or histogram.


## 10.7. Add Parameter

To add a parameter to the Parameter Measurement Info window, access the Add Parameter menu using one of the following methods.

- Select the Add Parameter in the Measure menu to open the Add Parameter menu.
- Click icon ⊕ in the parameter measurement info label to enter the add parameter menu.

In the Add Parameter menu, click Vertical, Horizontal, Other, or Power, or swipe left or right to browse the menu, then select a parameter to start the corresponding measurement. This oscilloscope supports up to 27 parameter measurements to be opened at the same time.

a. Source

Click Source 1 or Source 2 to select C1 - C4 or M1 - M4 as the measurement source. Only certain parameters can be added. Supported parameters are highlighted, while unsupported parameters are grayed out.

b. Addible parameters

- Vertical parameters: Maximum, minimum, peak-to-peak, top, bottom, amplitude, middle value, average value, period average, RMS (root mean square), RMS of cycle, AC RMS, AC RMS of cycle, area, cycle area, positive area, negative area, positive cycle area, negative cycle area, positive overshoot, negative overshoot, positive preshoot, and negative preshoot.


■ Horizontal parameters: Period, frequency, rise time, fall time, positive pulse width, negative pulse width, positive duty ratio, negative duty ratio, positive pulse width count, negative pulse width count, rising edge, falling edge, rising edge count, falling edge count, burst width, burst interval, burst period, and burst period count.


■ Other parameters: Ratio, period ratio, setup time, hold time, setup and hold ratio, FRFR, FRFF, FFFR, FFFF, FRLF, FRLR, FFLR, FFLF, phase (r-r), and phase (f-f), delay (r-r), and delay (f-f).


When power analysis is enabled, parameters can be added for measurement as follows.

- Power quality parameters: Voltage peak, voltage RMS, voltage peak factor, current peak, current RMS, current peak factor, active power, reactive power, apparent power, power factor, and power phase angle.
- Surge current parameter: Surge current
- Rds(on) parameter: Rds(on)
- Switching loss parameters: Switch period count, switching-on power loss, Rds (on) power loss, switching-off power loss, total power loss, switching-on energy loss, Rds (on) energy loss, switching-off energy loss, and total energy loss.
- Slew rate parameters: dv/dt and di/dt.
- Ripple analysis parameters: Peak-to-peak ripple analysis, RMS ripple analysis.
- Startup/shutdown time: Starup time and shutdown time.
- Transient response: Transient response time
- Efficiency: Input power, output power, and efficiency

## 10.8. Clear Added Measurement

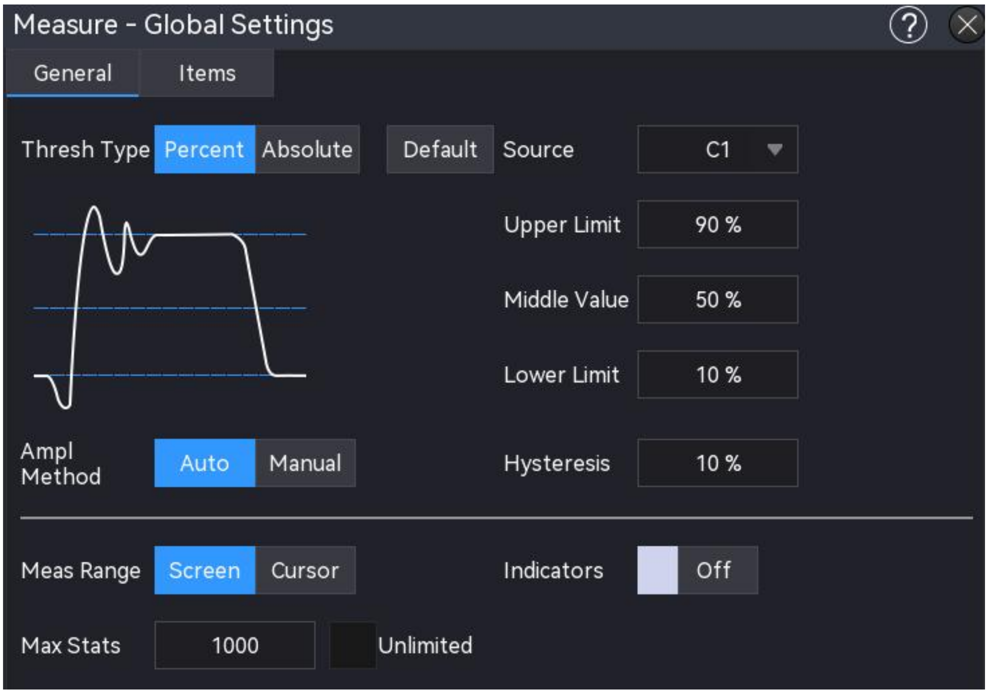
All added measurements can be deleted.

- In the Add Measure menu, click added parameter to select and to delete.
- In the results window at the bottom of screen, click "-" in the top right-corner of any measurement to delete the currently selected measurement.

■ In the Measure menu, click Clear to delete all added measurements.

## 10.9. Global Setting

In the Measure menu, click Global Settings to enter the Advanced setting menu.




The general and measurement settings can be set in the Advanced setting menu.

(1) General Setting

a. Threshold

■ Threshold type: Percentage, absolute value.

■ Default: Click Default to restore the upper limit, middle value, and lower limit to their default values. When the threshold type is set to absolute value, the upper and lower limits default to ±2.5 div.

■ Source: Click Source to select the measurement channel. C1-C4, M1-M4 are available.

■ Upper limit: Sets the upper reference level for waveform measurement. Click Upper Limit input field to open the numeric keypad and set the value. The default percentage is 90%, and the range can be set from 7% to 95%. The default absolute value is 500 mV, and the setting range changes with the vertical scale.

■ Middle value: Sets the middle reference level for waveform measurement. Click Middle Value input field to open the numeric keypad and set the value. The default percentage is 50%, and the range can be set from 6% to 94%. The default absolute value is 0 V, and the setting range changes with the vertical scale.

■ Lower limit: Sets the lower reference level for waveform measurement. Click Lower

Limit input field to open the numeric keypad and set the value. The default percentage is 10%, and the range can be set from 5% to 93%. The default absolute value is -500 mV, and the setting range changes with the vertical scale.

b. Measurement Range

The measurement window in the horizontal direction affects the results of all parameter measurements. The measure range can be set to the screen area or cursor area.

- Screen area: Full screen
- Cursor area: The horizontal time cursor area allows the user to set the cursor position as required and to measure the results directly within the cursor area.

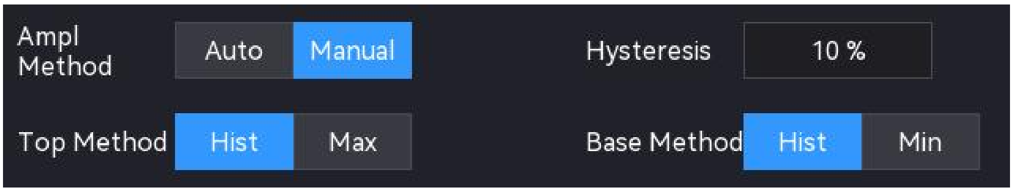
c. Maximum Count

This is a custom parameter. When measurement statistics are enabled, the number of statistical samples can be set from 10 to 10000, or set to unlimited times.

d. Amplitude Calculation

The amplitude measure mode can be set to auto or manual. The amplitude calculation affects the measurement of the top and bottom values.

- Auto: Based on the input signal, the amplitude calculation is automatically selected.
- Manual: Based on the manually selected top and bottom method, the corresponding amplitude values are calculated.




e. Top Calculation

- Histogram: Counts the value greater than the peak-to-peak 1/2, the highest probability is recognized as the top value.
- Maximum: The maximum of waveform is recognized as the top value.

f. Bottom Calculation

- Histogram: Counts the value that less than the peak-to-peak 1/2, the highest probability is recognized as the bottom value.
- Minimum: The minimum of waveform is recognized as the bottom value.

g. Indicator

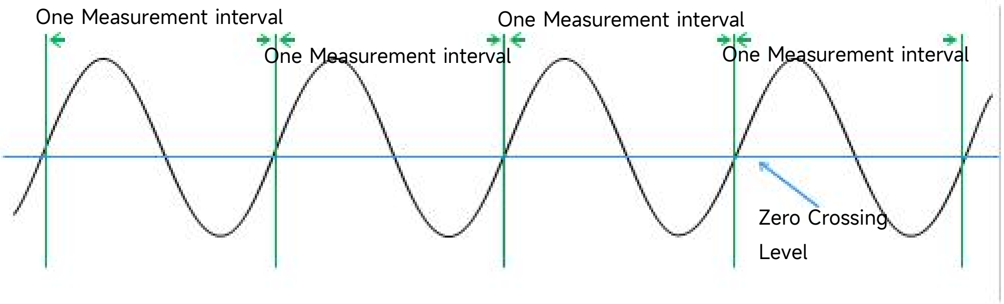
Click Indicator to turn the indicator on or off.

When the cursor indicator is switched on, one or more cursors appear on the screen. Before enabling the cursor indicator, at least one automatic measurement parameter must be active. The number of cursors will change according to the measurement parameter.

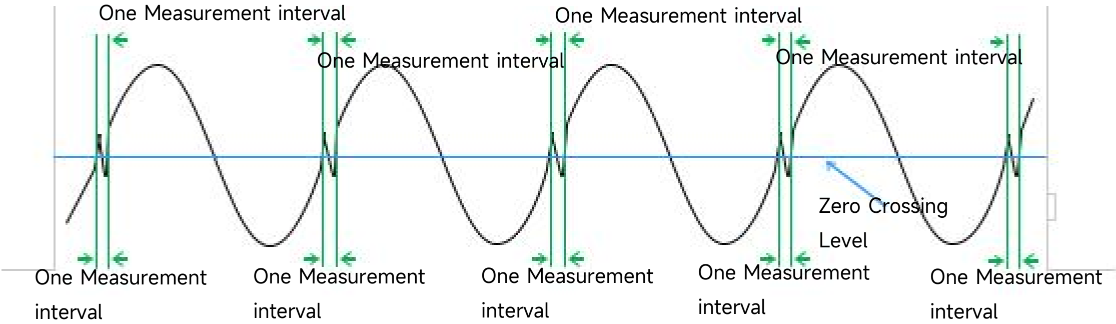
h. Hysteresis

Hysteresis defines a hysteresis band around the waveform amplitude threshold. This band is created by adding a hysteresis value to the threshold midpoint. Hysteresis is particularly useful when measuring noisy signals, as it allows the oscilloscope to ignore small, unwanted signal fluctuations and achieve more stable measurements.

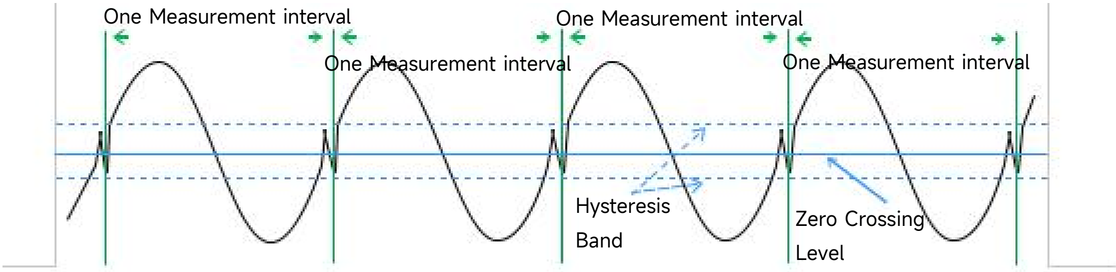
To illustrate the operating principle of hysteresis band settings, consider the perfect sine wave example shown in the figure below. In this case, the zero-crossing point or the 50% crossing point is clearly defined, allowing the oscilloscope to accurately detect the crossing and determine the measurement interval.




Now consider the example shown in the following figure, where non-monotonic behavior is present near the zero-crossing or 50% crossing level. During this region, multiple crossings may occur within a short time interval. Without hysteresis, this non-monotonic region may be incorrectly detected as the measurement interval, resulting in errors in period determination and, consequently, inaccurate calculation results.




By enabling hysteresis control, you can set a hysteresis band that is larger than the non-monotonic amplitude. This ensures that only valid signal crossings are detected, thereby preventing incorrect measurement interval calculations and improving overall measurement accuracy.




(2) Measurement Setting

a. Horizontal Parameter Setting

Current acquisition period averaging count: When measuring specific horizontal parameters, the instrument averages all measurement results obtained within the current acquisition and displays the averaged value as the current measurement result. This function takes effect only when it is enabled.

Applicable parameters: Period, frequency, rise time, fall time, positive pulse width, negative width, positive duty cycle, and negative duty cycle

b. RMS

- Unit: Sets the unit display for root-mean-square (RMS)-related parameters. The available options are RMS, dBm, and dB. When the unit is switched to dBm, ensure that the test load impedance is 50 Ω to maintain measurement accuracy.
- Reference value: When the unit is set to dB, the reference amplitude can be configured. The reference range can be set from 0.001 to 1000.

c. Burst

- Idle time: Sets the idle time for the measurement of burst width, burst interval, burst period, and burst period count.
- Idle level: High or low level.

d. Setup & Hold Setting

- Clock edge: Rising edge, falling edge, or rising/falling edge.
- Data edge: Rising edge, falling edge, or rising/falling edge.

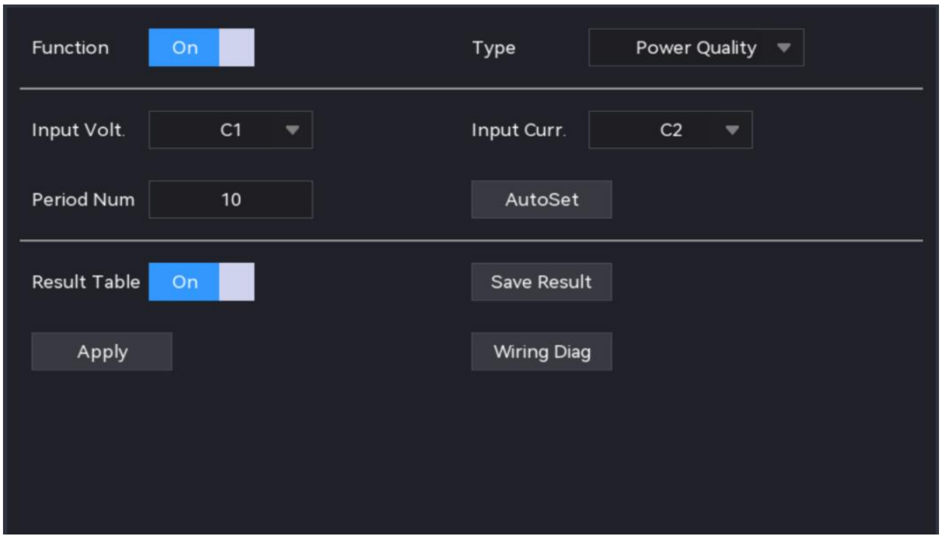
# 11. Power Analysis

- Power Quality
- Harmonics Analysis
- Surge Current
- Rds (on)
- Switching Loss
- Slew Rate
- Safe Operating Area
- Modulation Analysis
- Ripple Analysis
- Startup/Shutdown Time
- Transient Response
- Efficiency

This oscilloscope supports power analysis function (option), it can help the user to quickly analyze the efficiency and reliability of switch power. With this function, the user can analyze the power quality, harmonics analysis, surge current of the input power, Rds (on), switching loss, slew rate, safe operating area, and modulation analysis, ripple analysis, startup/shutdown time, transient response, and efficiency.

Access the power analysis menu using the following method.

- Click Home icon in the top right-corner and select the power analysis icon to enter the Power Analysis menu.
- If the power analysis is added to the toolbar, click power analysis icon in the toolbar in the top right-corner to enter the Power Analysis menu.




## 11.1. Power Quality

The power quality function measures the quality of the AC input line. Power quality analysis parameters include the voltage peak, RMS voltage, voltage peak factor, current peak, RMS current, current peak factor, active power, reactive power, apparent power, power factor, and power phase angle.

(1) Analysis Mode

Click Type to select Power Quality.

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

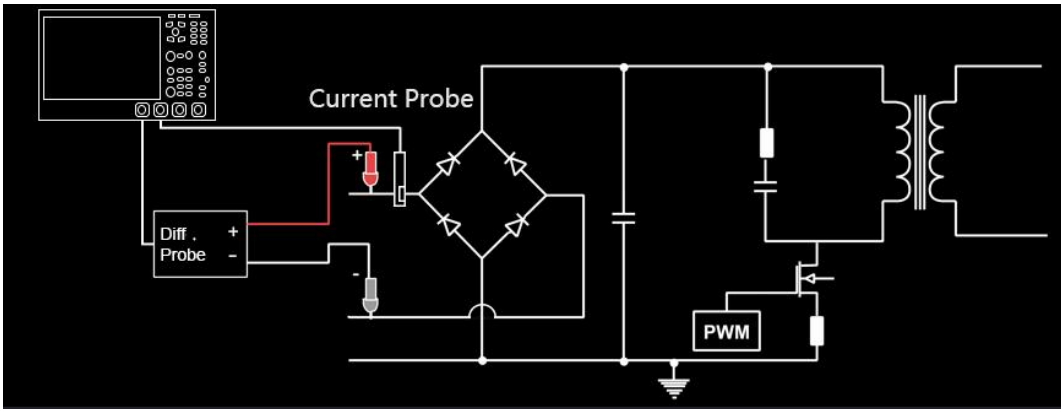
Click Wiring Diagram to display the wiring diagram of the power quality analysis. Follow the instructions in the figure below to make the wiring connections.

a. Voltage Probe

- Connect D+ to the live wire of the AC input.
- Connect D- to the neutral wire of the AC input.
- Select an appropriate probe attenuation ratio.

b. Current Probe

- Connect current probe to the live wire of the AC input. The arrow on the probe indicates the direction of the current.
- Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage channel (C1-C4). Set the unit and probe attenuation ratio of the voltage channel according to the connected voltage probe.

(5) Input Current

Click Input Current to select the input current channel (C1-C4). Set the unit and probe attenuation ratio of the current channel according to the connected current probe.

(6) Period Number

Double-click Period Number input field to open the numeric keypad and set the period number. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 40.

(7) Auto Setting

After configuring the input voltage and input current sources, the Autoset function automatically opens the voltage and current channels, sets the channel units to V and A, and closes the other channels.

(8) Apply

Click Apply to automatically open the Math1 channel, set the basic operation to voltage source x current source, and automatically add all power quality parameters while enabling the statistics. The measurement results are displayed in three formats: figure, result table, and parameter measurement.

- Graph result: Voltage waveforms, current waveforms, and power waveforms (power diagrams) are product of the current waveform and voltage waveform.
- Result table: Displays statistical results in a table.
- Parameter measurement: Displays power quality results in the parameter at the bottom of the screen.

(9) Result Table

Click Result Table key to open the measurement results table of power quality analysis.


(10) Power Quality Measurement Results

| | |
|---|---|
| Voltage peak | Maximum voltage waveform |
| RMS voltage | Effective voltage value $Vrms = \frac{1}{N} * \sqrt{\sum_{i=0}^{n-1} Vi^2}$ |
| Voltage peak factor | V_Crest = Vpeak / Vrms |
| Current peak | Maximum current waveform |
| RMS current | Root mean square of current $Irms = \frac{1}{n} * \sqrt{\sum_{i=0}^{n-1} Ii^2}$ |
| Current peak factor | I_Crest = Ipeak / Irms |
| Active power | The actual power consumed by the load, measured in watts (W). |
| Reactive power | The reactive power caused by inductors and capacitors in a circuit, measured in volt-amps reactive (VARs). |
| Apparent power | The product of the effective values of voltage and current, measured in volt - ampere (VA). |
| Power factor | The ratio of the actual power to the apparent power. |
| Power phase angle | In power triangle (apparent power $^2$ = active power $^2$ + reactive power $^2$), the phase angle is the angle between the apparent power and active power, it indicates the amount of reactive power. |

(11) Save Results Table

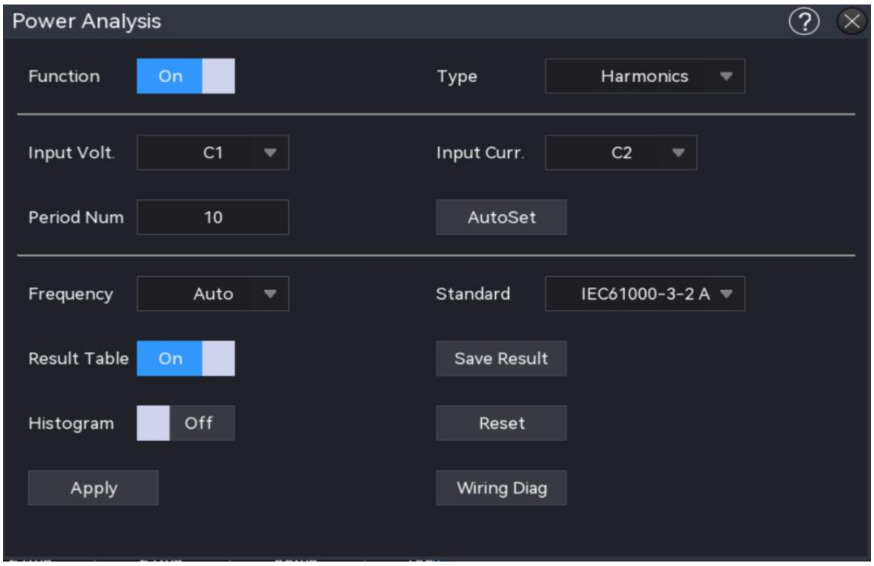
After opening the result table, click Save Result to open the export setting menu, the data can be saved as *.csv and *.pdf files to the internal storage or an external USB flash drive (when a

USB is detected). For the setting steps, refer to the section of Save and Load.

## 11.2. Harmonics Analysis

Current harmonics refer to the amplitudes of frequency components injected into AC lines, which may interfere with other devices on the same power grid or network.

The oscilloscope performs FFT (Fast Fourier Transform) on the current waveform and generates a measurement result table for up to 40 harmonics in IRMS, based on IEC 61000-3-2. It then compares the results with the limit values specified in the IEC standard to determine the pass/fail outcomes.




(1) Analysis Mode

Click Type to select Harmonics.

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

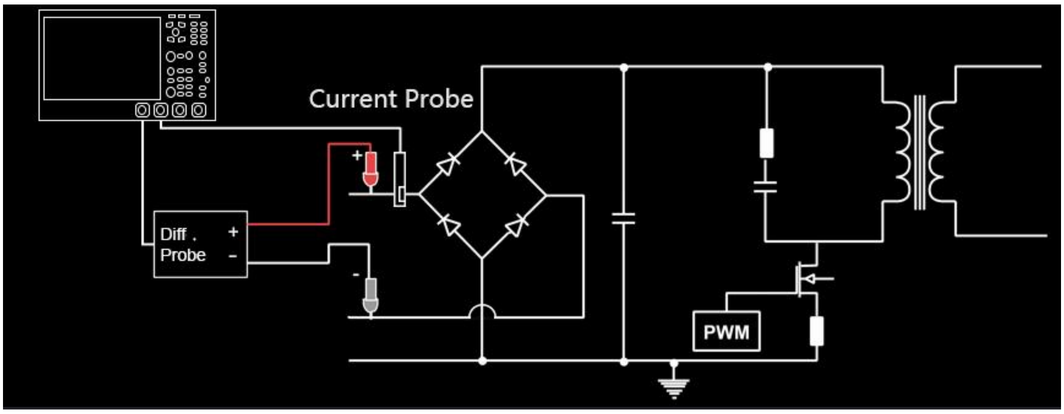
Click Wiring Diagram to display the wiring diagram of the current harmonics analysis. Follow the instructions in the figure below to make the wiring connections.

a. Voltage Probe

- Connect D+ to the live wire of the AC input.
- Connect D- to the neutral wire of the AC input.
- Select an appropriate probe attenuation ratio.

b. Current Probe

- Connect current probe to the live wire of the AC input. The arrow on the probe indicates the direction of the current.
- Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage channel (C1-C4). Set the unit and probe attenuation ratio of the voltage channel according to the connected voltage probe.

(5) Input Current

Click Input Current to select the input current channel (C1-C4). Set the unit and probe attenuation ratio of the current channel according to the connected current probe.

(6) Auto Setting

After configuring the input voltage and input current sources, the Autoset function automatically opens the voltage and current channels, sets the channel units to V and A, and closes the other channels.

(7) Line Frequency

Click Line Frequency to set the input line frequency, it can be set to auto acquisition, 50 Hz, 60 Hz, or 400 Hz.

(8) Harmonic Standard

Click Harmonic Standard to select the test standard for harmonic analysis (IEC61000-3-2 A/B/C/D).

- IEC61000-3-2 A: It is suitable for balanced three-phase equipment, household appliances (except D-type), tools (except portable tools), incandescent lamp, and audio frequency apparatus.
- IEC61000-3-2 B: It is suitable for portable tools.
- IEC61000-3-2 C: It is suitable for lighting equipment.
- IEC61000-3-2 D: It is suitable for the device that the rated power is less than or equal to 600W, the type is personal PC, personal computer monitors, and television receiver.

(9) Period Number

Double-click Period Number input field to open the numeric keypad and set the period number. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 40.

(10) Apply

Click Apply to enable the oscilloscope to automatically perform the harmonic analysis measurements based on the user-defined input voltage, input current, and period count. The measurement results are displayed in three formats, graph, result table, and histogram.

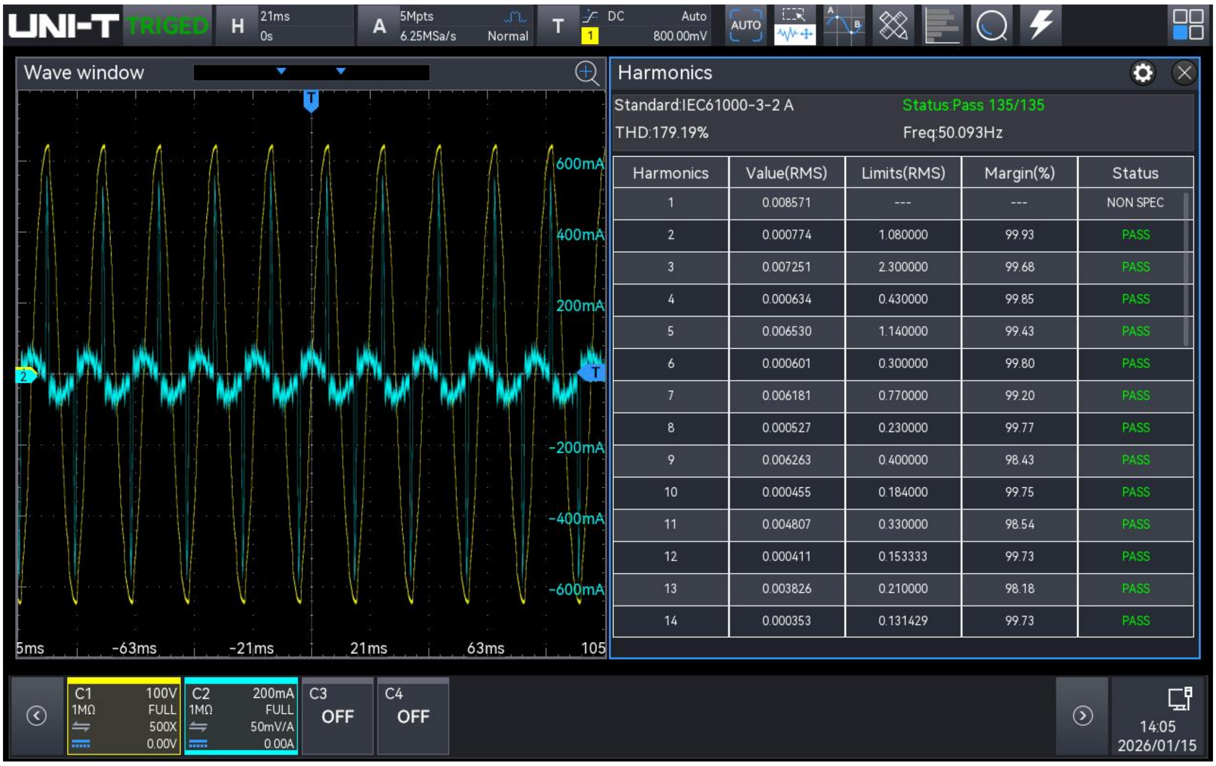
- Graph result: Displays voltage and current waveforms.
- Result table: Displays statistical results in a table.
- Histogram: Displays current harmonic results in a histogram.

(11) Histogram

Click Histogram key to open the histogram of harmonic analysis.


(12) Result Table

Click Result Table key to open the measurement results table of power quality analysis.




(13) Harmonic Analysis Measurement Results

| | |
|---|---|
| Harmonic, actual value (RMS), limit (RMS), margin, and state | The following values are displayed for the first 40 harmonics.<br>Actual value (RMS): The measured values displayed in the unit specified by the Harmonic Unit.<br>Limit (RMS): The limit defined by the selected harmonic analysis standard.<br>Margin: The margin defined by the selected harmonic analysis standard.<br>Pass/Fail: Indicates whether the value passes or fails according to the selected harmonic analysis standard.<br>Rows in the table or bars in the bar graph are colored based on the Pass/Fail status. A critical result is defined as a value greater than 85% of the limit but less than 100% of the limit. |
| THD (Total Harmonic Distortion) | $\mathrm{TDH} = 100 \times \frac{\sqrt{X_2{}^2+X_3{}^2+..+X_n{}^2.}}{X_1}$<br>$X_n$ = Voltage or current of each harmonic<br>$X_1$ = Fundamental voltage or current |

(14) Save Results Table

After opening the result table, click Save Result to open the export setting menu, the data can be saved as *.csv and *.pdf files to the internal storage or an external USB flash drive (when a USB is detected). For the setting steps, refer to the section of Save and Load.

# 11.3. Surge Current

The surge current refers to the initial current surge consumed when the power supply is first turned on. This current, which exceeds the steady-state current of the power converter, is primarily due to the charging of the input capacitor. A surge current test can verify whether the AC switch, bridge rectifier, fuse, and EMI filter operate within their allowable current limits. When the switching loop is repeatedly turned on and off, the AC input voltage should neither damage the power supply nor cause the fuse to blow. The maximum or minimum instantaneous value of the surge current can be captured using the oscilloscope single-shot acquisition function.

(1) Analysis Mode

Click Type to select Surge Current.

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

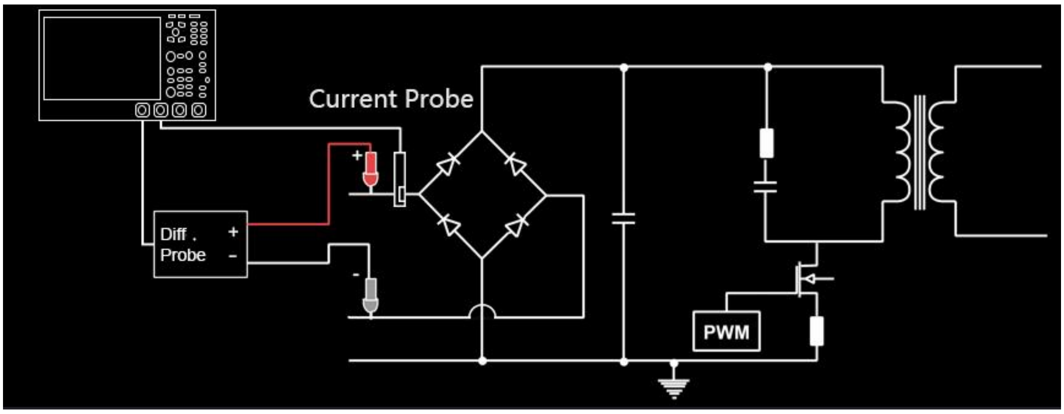
Click Wiring Diagram to display the wiring diagram of the power quality analysis. Follow the instructions in the figure below to make the wiring connections.

a. Voltage Probe

- Connect D+ to the live wire of the AC input.
- Connect D- to the neutral wire of the AC input.
- Select an appropriate probe attenuation ratio.

b. Current Probe

- Connect current probe to the live wire of the AC input. The arrow on the probe indicates the direction of the current.
- Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage channel (C1-C4). Set the unit and probe attenuation ratio of the voltage channel according to the connected voltage probe.

(5) Input Current

Click Input Current to select the input current channel (C1-C4). Set the unit and probe attenuation ratio of the current channel according to the connected current probe.

(6) Maximum Input Voltage (RMS)

Assign the maximum input voltage and set the vertical calibration of channel voltage. Double-click Maximum Input Voltage (RMS) input field to open the numeric keypad and set the maximum input voltage.

For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 V to 1000 V.

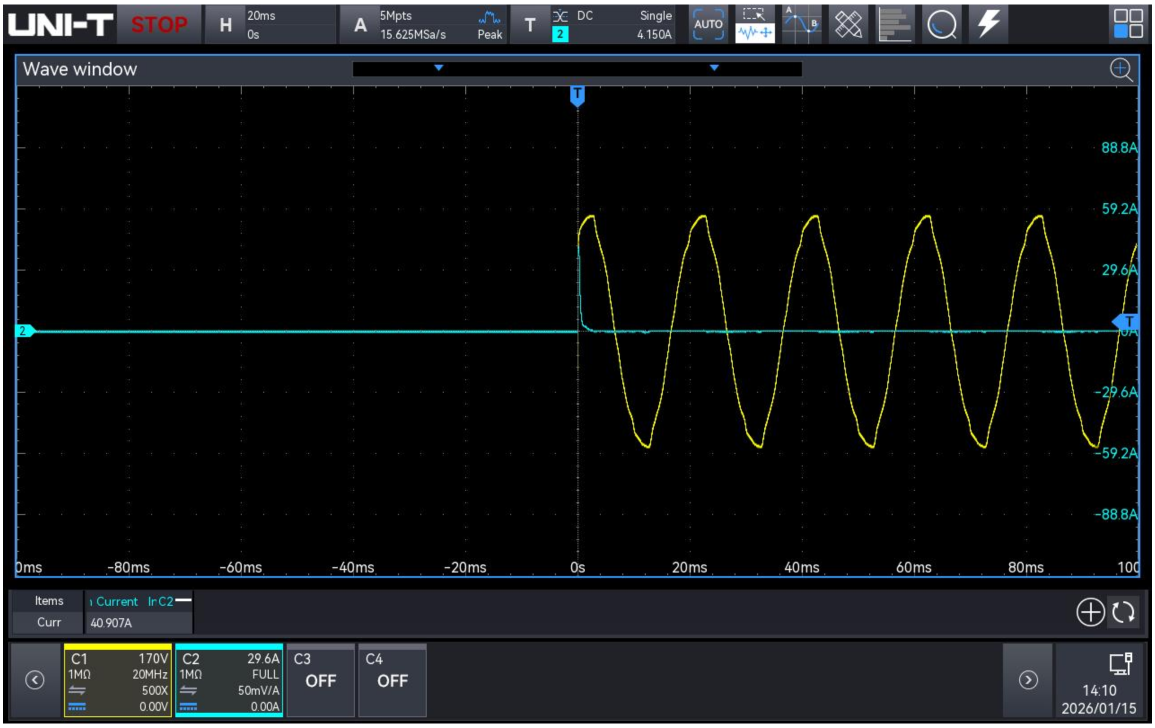
(7) Prospective Current

The prospective current is used to assign the expected surge current amplitude and set the vertical calibration of the current channel.

For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 100 mA to 500 A.

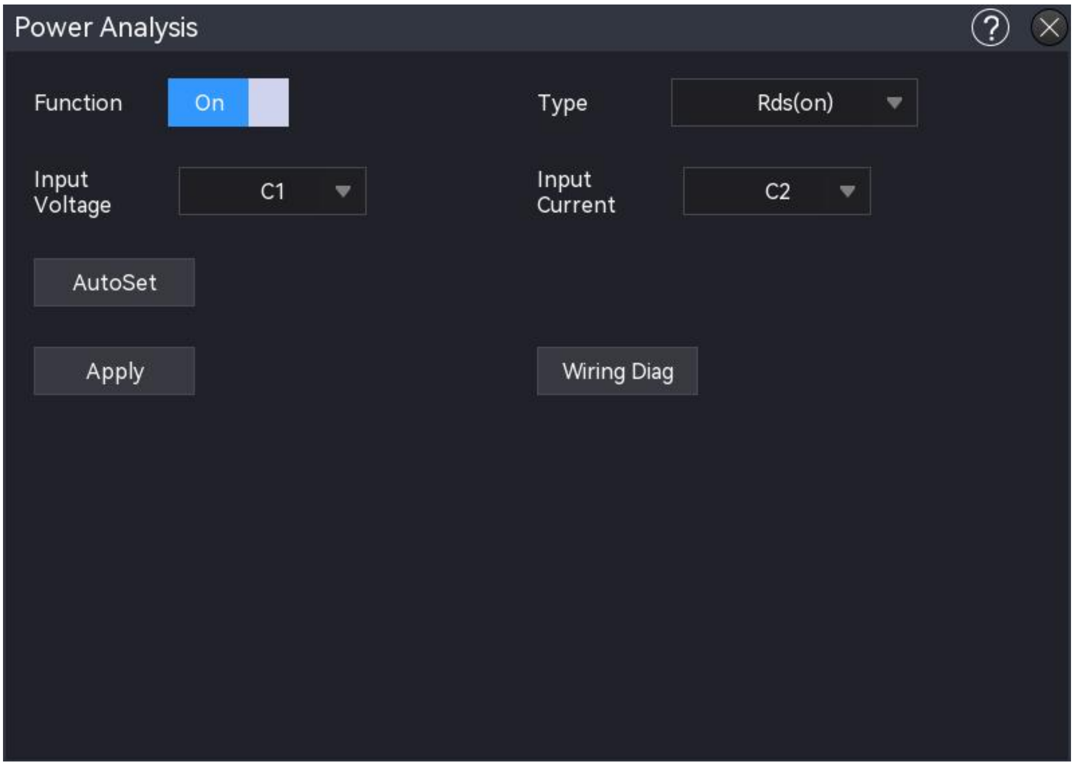
(8) Apply

Click Apply and follow the on-screen instructions. The result is displayed after the analysis is completed.




## 11.4. Rds(on) (Drain-Source On-State Resistance)

This measurement characterizes the on-state resistance, Rds(on), between the drain and source of the switching device during the conduction phase of the switching cycle.




(1) Analysis Type

Click Type to select Rds(on).

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

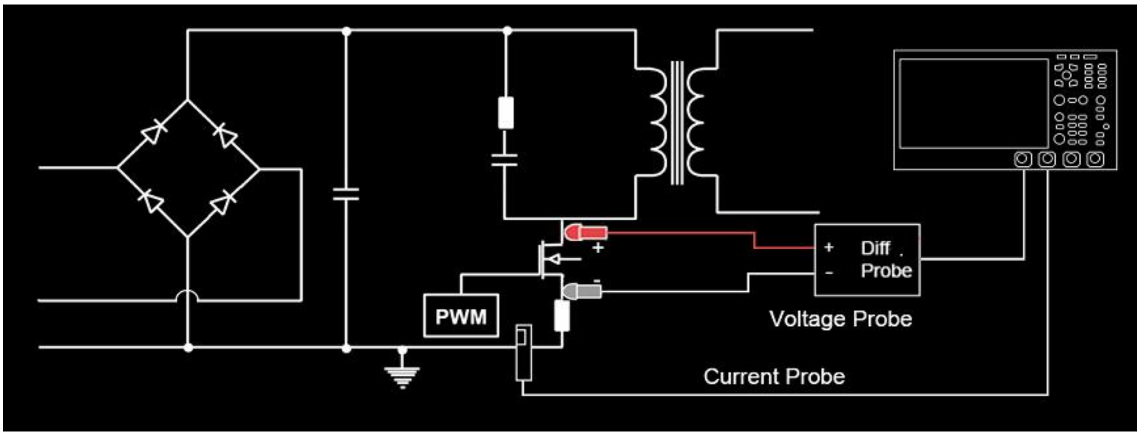
Click Wiring Diagram to display the wiring diagram of Rds (on). Follow the instructions in the figure below to make the wiring connections. Tap the icon in the top-right corner to close the wiring diagram.

a. Voltage Probe

- Connect D+ to the drain of the MOSFET.
- Connect D- to the source of the MOSFET.
- Select an appropriate probe attenuation ratio.

b. Current Probe

- Connect the current probe to the source of the MOSFET. The arrow on the probe indicates the direction of the current.
- Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage channel (C1-C4). Set the unit and probe attenuation ratio of the voltage channel according to the connected voltage probe.

(5) Input Current

Click Input Current to select the input current channel (C1-C4). Set the unit and probe attenuation ratio of the current channel according to the connected current probe.

(6) Auto Setting

After configuring the input voltage and input current sources, the Autoset function automatically opens the voltage and current channels, sets the channel units to V and A, and closes the other channels.

(7) Apply

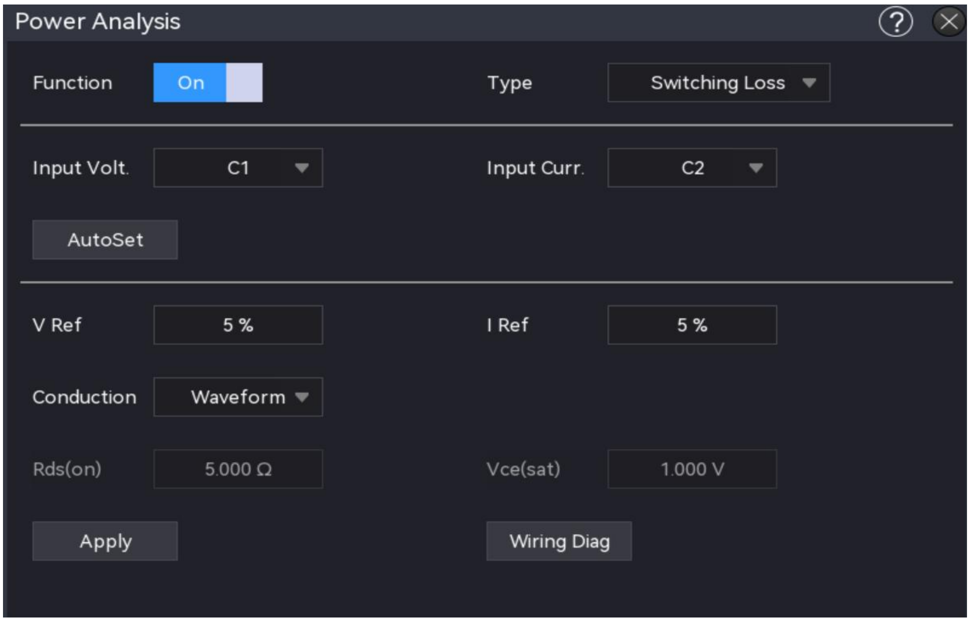
Click Apply to perform the power analysis function. This automatically adds the Rds(on) parameters and enables statistics.

(8) Slew Rate Measurement Results

| | |
|---|---|
| Rds(on) | The effective resistance of the power switch component during the conduction stage. |

## 11.5. Switching Loss

On-state losses occur when physical and parasitic capacitors are charged. During this process, the inductor generates a magnetic field, causing transient resistance losses. Similarly, when a switched-mode power supply is turned off, energy is discharged and interacts with various components, even if the mains power is unplugged, resulting in additional losses.




(1) Analysis Type

Click Type to select Switching Loss.

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

Click Wiring Diagram to display the wiring diagram of switching loss. Follow the instructions in the figure below to make the wiring connections. Tap the icon in the top-right corner to close the wiring diagram.

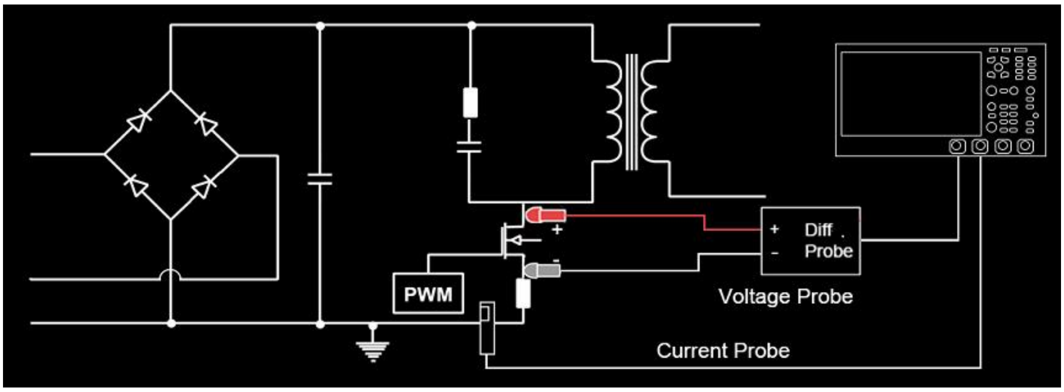
a. Voltage Probe

- Connect D+ to the drain of the MOSFET.
- Connect D- to the source of the MOSFET.
- Select an appropriate probe attenuation ratio.

b. Current Probe

- Connect the current probe to the source of the MOSFET. The arrow on the probe indicates the direction of the current.

- Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage channel (C1-C4). Set the unit and probe attenuation ratio of the voltage channel according to the connected voltage probe.

(5) Input Current

Click Input Current to select the input current channel (C1-C4). Set the unit and probe attenuation ratio of the current channel according to the connected current probe.

(6) Auto Setting

After configuring the input voltage and input current sources, the Autoset function automatically opens the voltage and current channels, sets the channel units to V and A, and closes the other channels.

(7) Voltage Reference

The voltage reference is used to identify the conduction state. When the voltage falls below the voltage reference percentage of the maximum waveform voltage, it is recognized as the on-state. The setting range is from 1% to 100%.

(8) Current Reference

The current reference is used to identify the conduction state. When the current falls below the current reference percentage of the maximum waveform current, it is recognized as the off-state. The setting range is from 0% to 100%.

(9) Conduction Type

Click Conduction to select the conduction type to voltage waveform, Rds(on), or Vce(sat).

- Voltage waveform: The power waveform uses the original data. The calculation formula is P = V * I, E = P * T.
- Rds(on): For on-state areas, where the voltage level is lower than the voltage reference (adjustable), the power calculation formula is P = I^2 * Rds(on).

  For off-state areas, where the current level is lower than the current reference (adjustable), the power calculation formula is P = 0 watt.

- Vce(sat): For on-state areas, where the voltage level is lower than the voltage reference (adjustable), the power calculation formula is P = Vce(sat) * I.

  For off-state areas, where the current level is lower than the current reference (adjustable), the power calculation formula is P = 0 watt.

(10) Rds(on)

The on-resistance can be configured when the conduction type is set to Rds(on), with a range from 1 mΩ to 200 Ω.

(11) Vce(sat)

The Vce voltage can be configured when the conduction type is set to Vce(sat), with a range from 50 mV to 2.5 V.

(12) Apply

Click Apply to enable the oscilloscope to automatically open the Math1 channel based on the user-defined input voltage, input current, voltage reference, and current reference. MATH1 is set to perform the basic operation of voltage × current. All switching loss parameters are automatically added, and the statistics function is enabled.

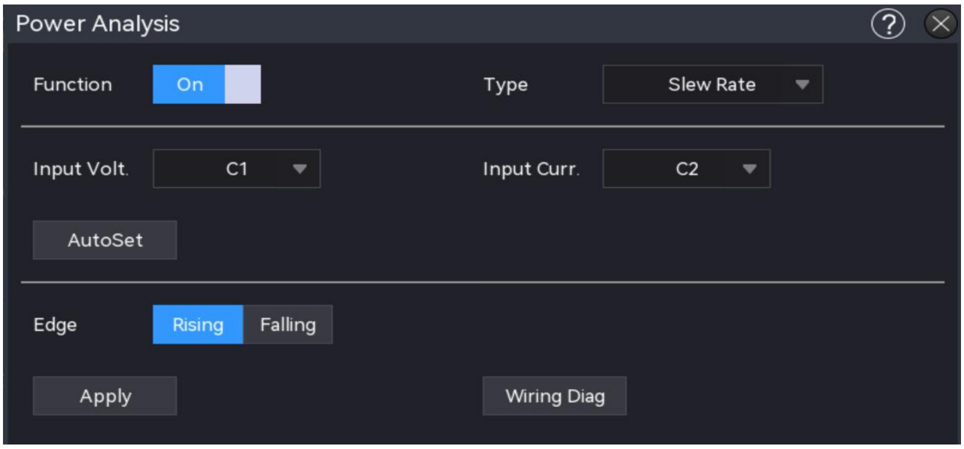

(13) Switching Loss Measurement Results

| | |
|---|---|
| Switching period number | Meets the required number of switching periods. |
| Switching-on power loss | The power loss of power switch device during the switching-on process, measured in watts (W). |

| | |
|---|---|
| Conduction power loss | The power loss of power switch device during the conducting process, measured in watts (W). |
| Switching-off power loss | The power loss of power switch device during the switching-off process, measured in watts (W). |
| Total power loss | The total power loss of power switch device during the whole switching period, measured in watts (W). |
| Switching-on energy loss | The energy loss of power switch device during the switching-on process, measured in joules (J). |
| Conduction energy loss | The energy loss of power switch device during the conducting process, measured in joules (J). |
| Switching-off energy loss | The energy loss of power switch device during the switching-off process, measured in joules (J). |
| Total energy loss | The total energy loss of power switch device during the whole switching period, measured in joules (J). |

## 11.6. Slew Rate

Slew rate analysis measures the rate of change in voltage or current during the switching period.




(1) Analysis Type

Click Type to select Slew Rate.

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

Click Wiring Diagram to display the wiring diagram of slew rate. Follow the instructions in the figure below to make the wiring connections. Tap the icon in the top-right corner to close the wiring diagram.

a. Voltage Probe

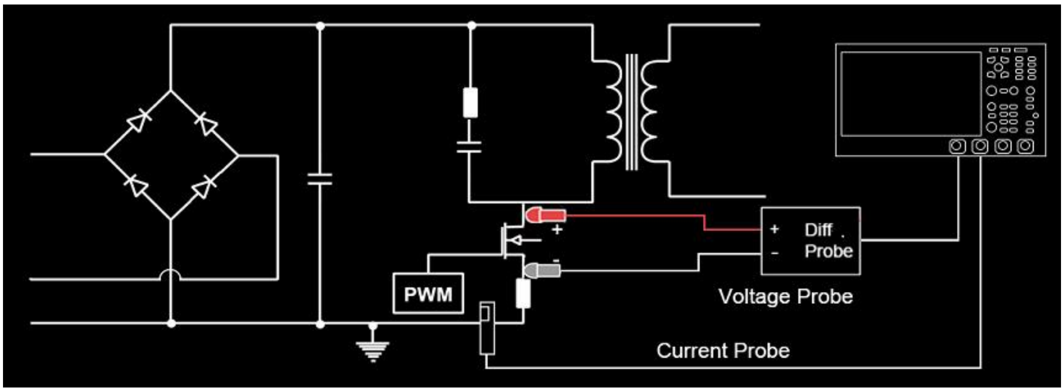
- Connect D+ to the drain of the MOSFET.
- Connect D- to the source of the MOSFET.

■ Select an appropriate probe attenuation ratio.

b. Current Probe

■ Connect the current probe to the source of the MOSFET. The arrow on the probe indicates the direction of the current.

■ Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage channel (C1-C4). Set the unit and probe attenuation ratio of the voltage channel according to the connected voltage probe.

(5) Input Current

Click Input Current to select the input current channel (C1-C4). Set the unit and probe attenuation ratio of the current channel according to the connected current probe.

(6) Auto Setting

After configuring the input voltage and input current sources, the Autoset function automatically opens the voltage and current channels, sets the channel units to V and A, and closes the other channels.

(7) Edge

Select the signal edge to test: rising edge or falling edge.

(8) Apply

Click Apply to enable the oscilloscope to automatically open the Math1 channel based on the user-defined input voltage, input current, and edge. The oscilloscope performs the slew rate analysis and automatically adds all slew rate parameters while enabling the statistics function.

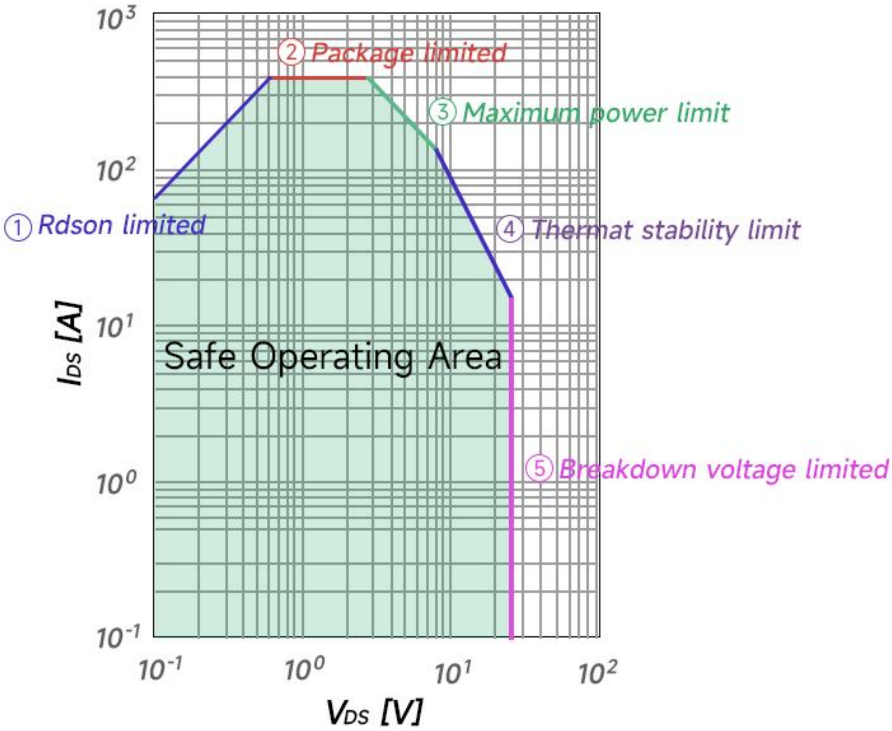

(9) Slew Rate Measurement Results

| | |
|---|---|
| dv/dt | The rate of change of voltage over time is analyzed during the switching process of a power switching device. |
| di/dt | The rate of change of current over time is analyzed during the switching process of a power switching device. |

## 11.7. Safe Operating Area

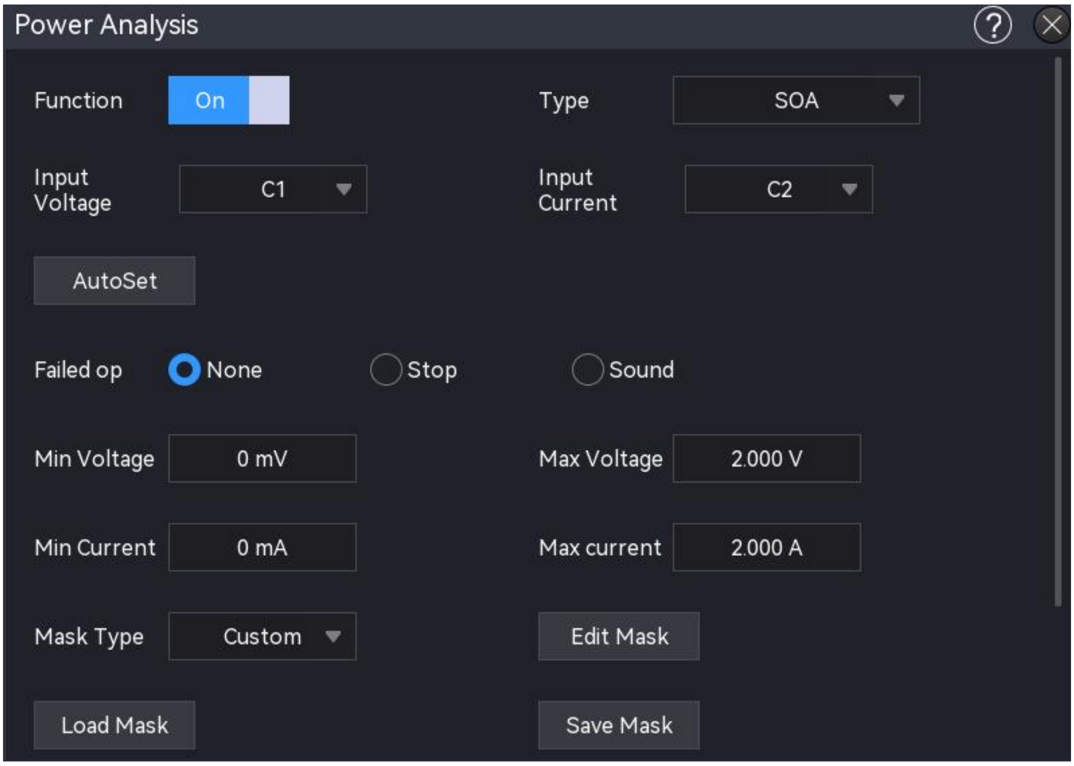
The safe operating area (SOA) of a MOSFET defines the limiting conditions for voltage, current, and power dissipation, under which the device can operate without sustaining damage.

The oscilloscope can automatically generate the SOA based on parameters such as voltage limits, current limits, and power limits configured in the menu. It evaluates whether the voltage and current of the switching device during normal operation exceed this area. This feature helps designers quickly identify potential problems or risks in the circuit.




(1) Analysis Type

Click Type to select SOA.




(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

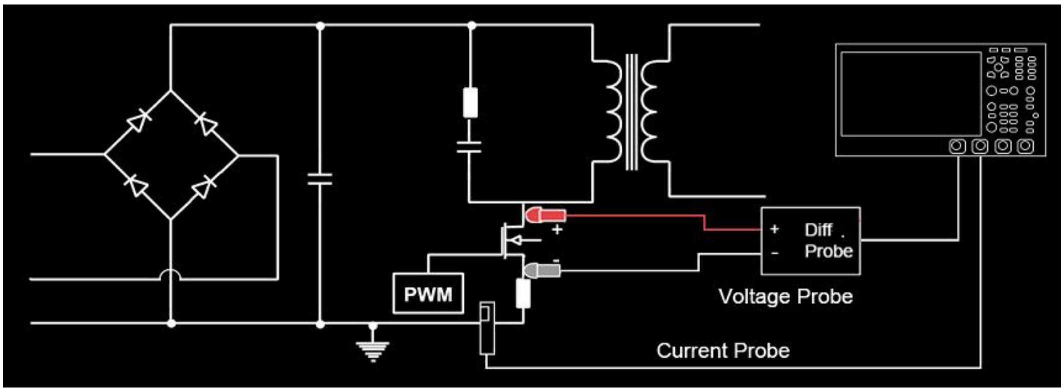
Click Wiring Diagram to display the wiring diagram of safe operating area. Follow the instructions in the figure below to make the wiring connections. Tap the icon in the top-right corner to close the wiring diagram.

a. Voltage Probe

- Connect D+ to the drain of the MOSFET.
- Connect D- to the source of the MOSFET.
- Select an appropriate probe attenuation ratio.

b. Current Probe

- Connect the current probe to the source of the MOSFET. The arrow on the probe indicates the direction of the current.
- Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage channel (C1-C4). Set the unit and probe attenuation ratio of the voltage channel according to the connected voltage probe.

(5) Input Current

Click Input Current to select the input current channel (C1-C4). Set the unit and probe attenuation ratio of the current channel according to the connected current probe.

(6) Auto Setting

After configuring the input voltage and input current sources, the Autoset function automatically opens the voltage and current channels, sets the channel units to V and A, and closes the other channels.

(7) Failure Operation

When the mask performs a failure operation, it can be configured to perform no operation, stop operation, and sound alarm.

- No operation: If the mask test fails, no operation will be performed, and the mask test will continue.
- Stop operation: If the mask test fails, the mask test will stop immediately.
- Sound alarm: If the mask test fails, the beeper will sound, but the mask test will continue.

(8) SOA Grid Setting

Set the minimum and maximum values for SOA voltage and current axes. This configures the minimum voltage, maximum voltage, minimum current, and maximum current.

- Minimum voltage: Sets the display minimum value for SOA voltage axis, with a default value of 0 mV.
- Maximum voltage: Sets the display maximum value for SOA voltage axis, with a default value of 10 V.
- Minimum current: Sets the display minimum current for SOA voltage axis, with a default value of 0 mA.
- Maximum current: Sets the display maximum current for SOA voltage axis, with a default value of 10 A.

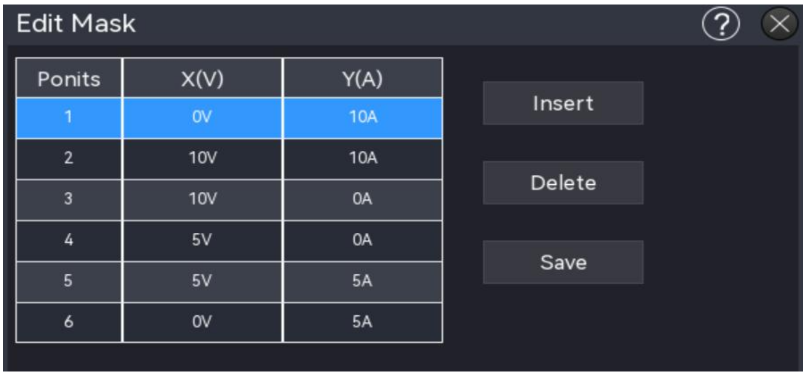
(9) Mask Type

The SOA mask can be configured using two generation types: Limit Value or Custom Value.

a. Limit value: Generates a mask using the voltage limit, current limit, and power limit.

- Voltage limit: Sets the maximum VDS (Drain-Source Voltage), with a default value of 5 V.
- Current limit: Sets the maximum IDM (Drain Current-Pulse), with a default value of 5 A.
- Power limit: Sets the power limit of thermal resistance, with a default value of 25 W.

b. Custom value: Generates a mask using user-defined settings.




- Edit mask: Click Edit Mask to open the mask editor. The default mask contains 6 points. The user can edit the mask by inserting, deleting, or editing points, and then save the mask.

  \- Insert point: Inserts a new point below the selected point.

  \- Delete point: Deletes the currently selected point.

  \- Edit point: Double-click selected point to set its axis by configuring the voltage value and current values.

  \- Save: Saves the user-defined mask and updates it in the SOA waveform display.

  **Note**: The mask point requires a block area; otherwise, the mask cannot be saved.

- Import mask: When the SOA mask type is set to user-defined, the user can load mask files stored in the instrument's internal memory or an external USB flash drive (only when the USB flash drive is detected) and apply them to the current SOA test. Click Import Mask to enter the file loading interface. Select the specified mask file (*.csv) from the loading path and apply it to the current SOA tes.

- Save mask: After editing the user-defined mask, the user can save it in *.csv format to the instrument's internal memory or an external USB flash drive (only when the USB flash drive is detected). Click Save Mask to enter the file saving interface. Enter the corresponding information in the File Name and Save Path fields to save the mask file to internal or external memory. For detailed saving instructions, refer to the Save and Load section.

(10) SOA Waveform

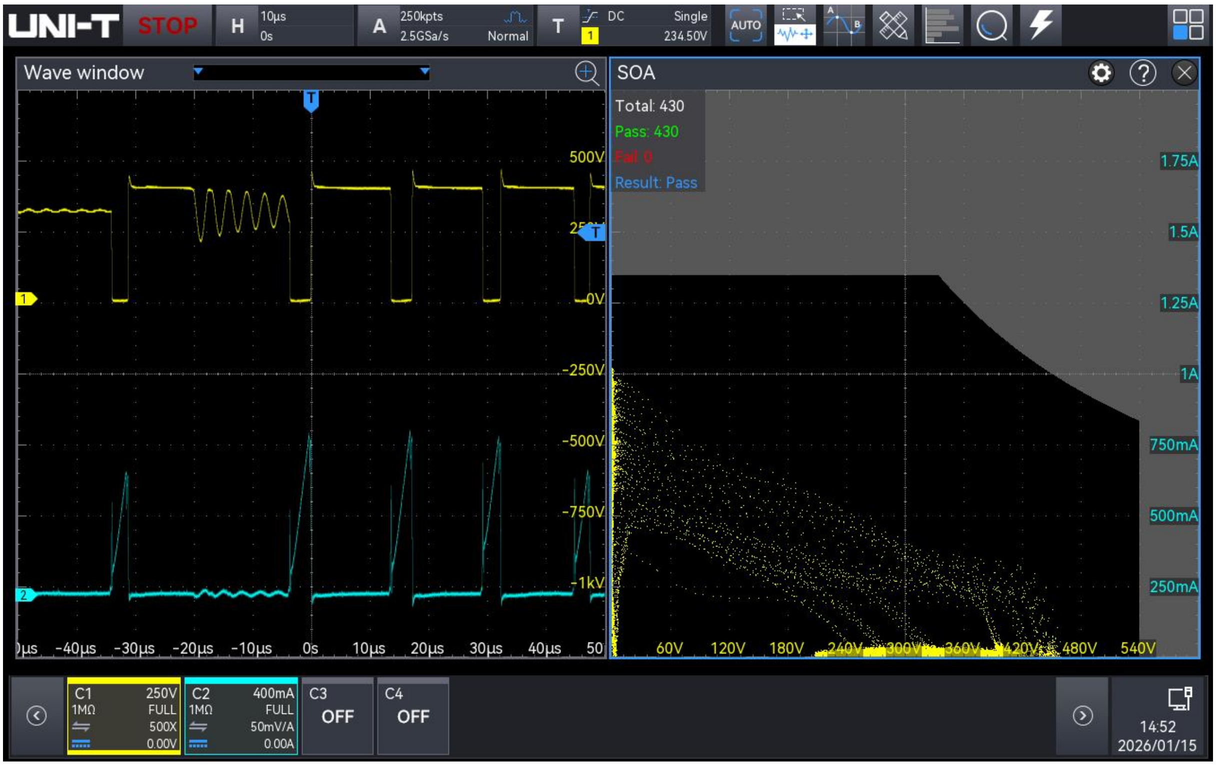
Click SOA Waveform to turn the waveform display on or off.

(11) Reset

Click Reset to clear the mask test results and restart the mask test.

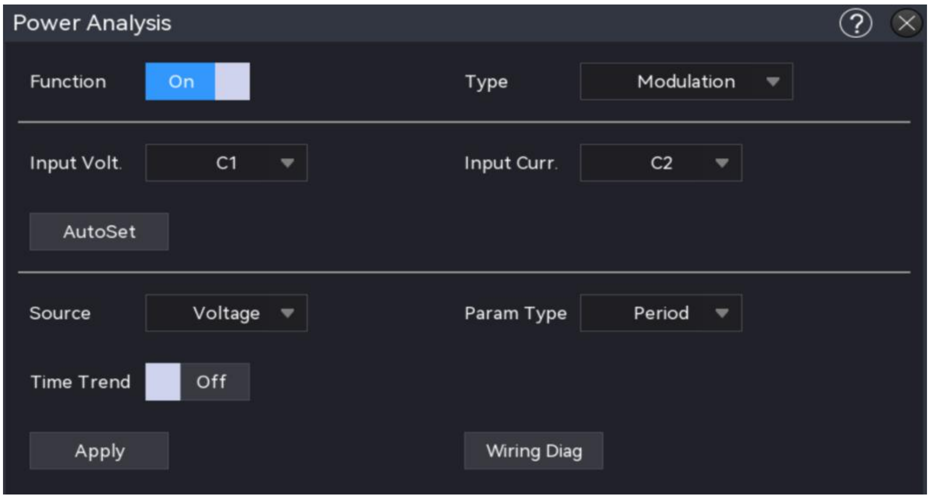
(12) Apply

Click Apply to enable the oscilloscope to perform the power analysis function based on the user-defined input voltage, input current, failure operation, minimum voltage, maximum voltage, minimum current, maximum current, and mask limit. The SOA waveform display is opened automatically.



## 11.8. Modulation Analysis

The feedback control loop stabilizes the output voltage, while modulation analysis specifically examines the characteristics of the control loop in the switching power supply. It measures the control pulse signal of the switching device (MOSFET) and observes the trends in pulse width, duty ratio, period, frequency, rise time, fall time, and other factors in response to various events.




(1) Analysis Type

Click Type to select Modulation.

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

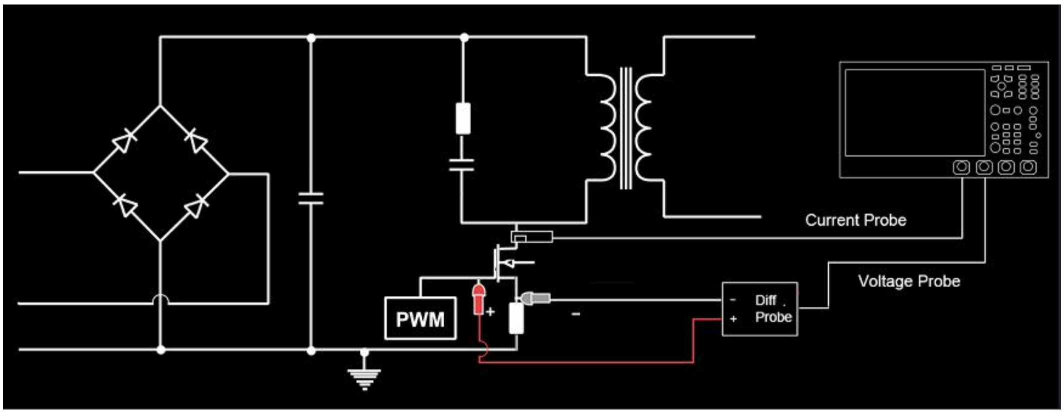
Click Wiring Diagram to display the wiring diagram of modulation analysis. Follow the instructions in the figure below to make the wiring connections. Tap the icon in the top-right corner to close the wiring diagram.

a. Voltage Probe

- Connect D+ to the drain of the MOSFET.
- Connect D- to the source of the MOSFET.
- Select an appropriate probe attenuation ratio.

b. Current Probe

- Connect the current probe to the source of the MOSFET. The arrow on the probe indicates the direction of the current.
- Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage channel (C1-C4). Set the unit and probe attenuation ratio of the voltage channel according to the connected voltage probe.

(5) Input Current

Click Input Current to select the input current channel (C1-C4). Set the unit and probe attenuation ratio of the current channel according to the connected current probe.

(6) Auto Setting

After configuring the input voltage and input current sources, the Autoset function automatically opens the voltage and current channels, sets the channel units to V and A, and closes the other channels.

(7) Source Reference

Click Source to configure the measurement source as voltage or current for the modulation analysis.

(8) Parameter Type

Click Parameter Type to configure the measurement type, including period, frequency, rise time, fall time, positive pulse width, negative pulse width, positive duty ratio, and negative duty ratio.

(9) Time Trend

Click Time Trend to turn the display on or off.

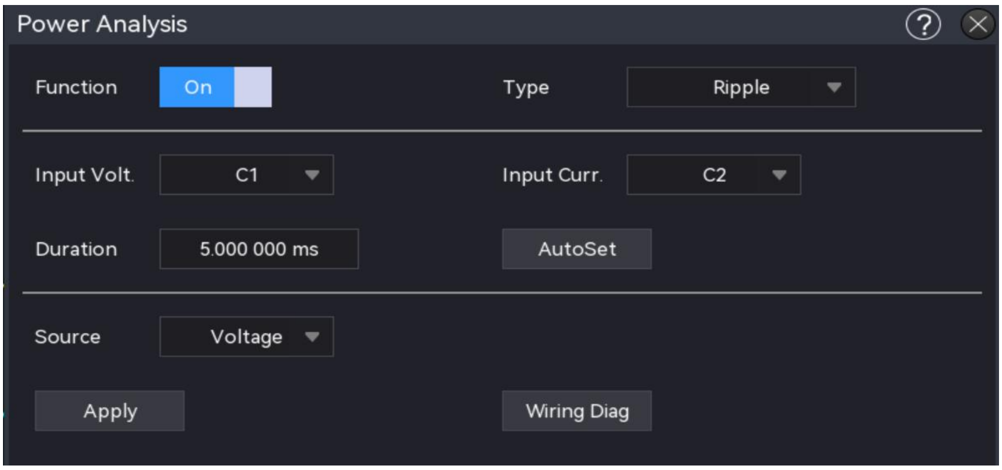
(10) Apply

Click Apply to enable the oscilloscope to perform the power analysis function based on the input voltage, input current, source reference, and parameter type. The time trend chart will automatically display after the analysis is complete.


The time trend graph visualizes the measured changes in each cycle of the modulation waveform using a mathematical waveform.

## 11.9. Ripple Analysis

Ripple refers to voltage fluctuations that occur in a DC regulated power supply. Because a DC regulated power supply is typically generated by rectifying and stabilizing an AC power source, it inevitably retains some AC components. These residual AC components, superimposed on the stable DC output, are known as ripple.




(1) Analysis Type

Click Type to select Ripple.

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

Click Wiring Diagram to display the wiring diagram of the ripple analysis. Please follow the instructions to make the wiring connection, as shown in the following figure. To close the diagram, click close icon in the upper-right corner.

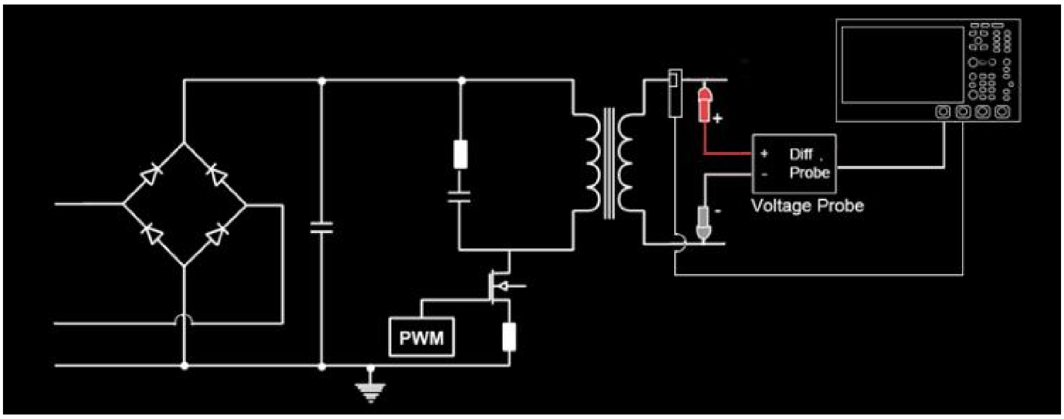
a. Voltage Probe

- Connect D+ to the power output terminal.
- Connect D- to the power ground.
- Select an appropriate probe attenuation ratio.

If the output voltage is low, a standard voltage probe can be used.

b. Current Probe

- Connect the current probe to the power output, ensuring that the arrow on the probe indicates the direction of current flow.
- Use Input Voltage and Input Current to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the channel to collect voltage (C1 - C4). The voltage channel should be configured with the appropriate unit and probe attenuation ratio according to the connected voltage probe.

(5) Input Current

Click Input Current to select the channel to collect current (C1 - C4). The current channel should be configured with the appropriate unit and probe attenuation ratio according to the connected current probe.

(6) Auto Setting

After configuring the input voltage and input current sources, the Autoset function automatically opens the voltage and current channels, sets the channel units to V and A, and closes the other channels.

(7) Duration

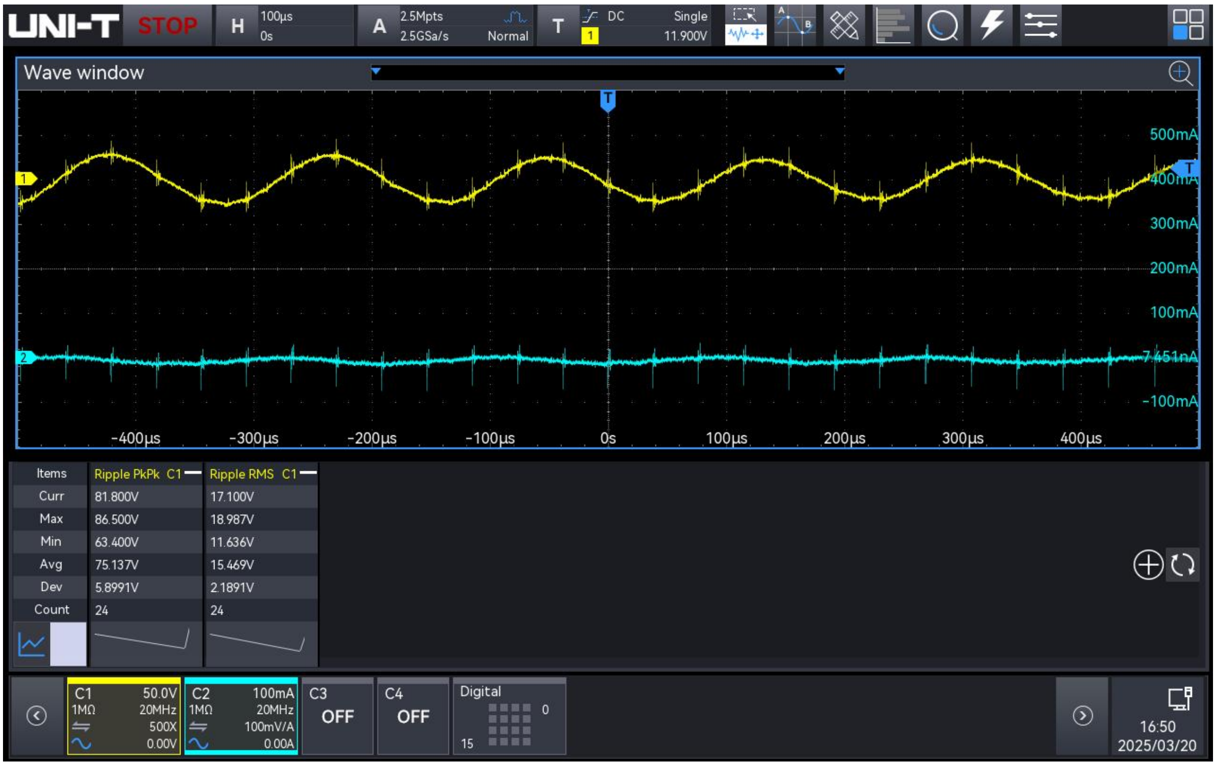
Set the time span for the entire waveform display. The valid range is from 10 ns to 20 s. The oscilloscope automatically adjusts the time base range using the formula: Time Base = Duration / 10

(8) Source Reference

Click Source to configure the measurement source as voltage or current for the ripple analysis.

(9) Apply

Click Apply to enable the oscilloscope to perform the ripple analysis function based on the input voltage, input current, duration, and source reference. This action will also automatically add all relevant ripple analysis parameters and enable statistics.




(10) Ripple Analysis Measurement Results

| | |
|---|---|
| Peak-to-Peak Ripple Value | The difference between the maximum and minimum values of the ripple waveform. |
| Root Mean Square (RMS) Ripple Value | The RMS value calculated from the ripple waveform. |

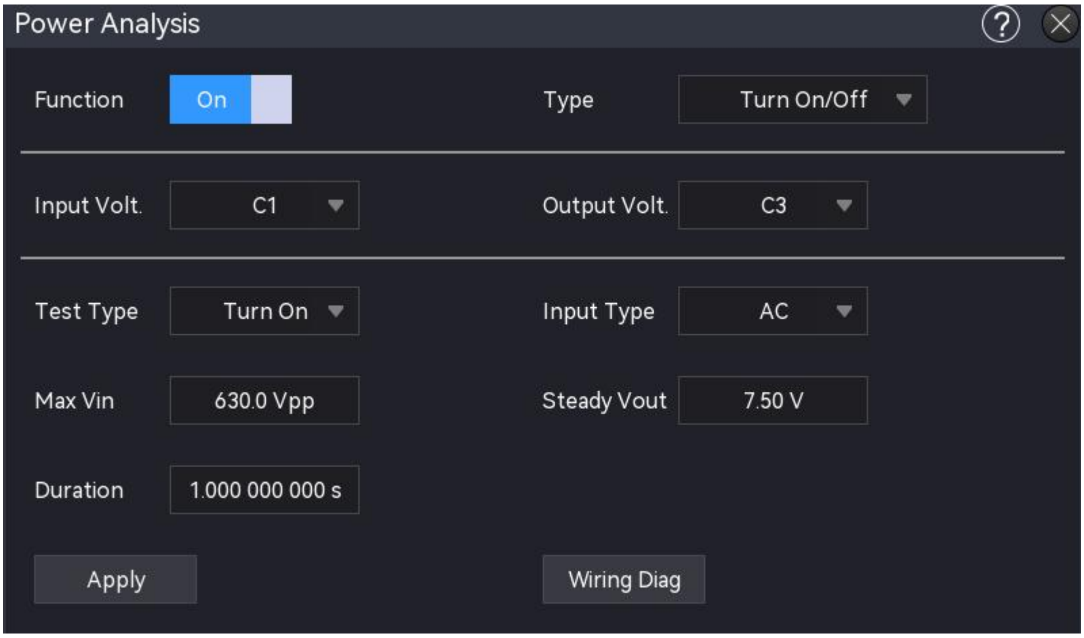
## 11.10. Startup/Shutdown Time

During startup analysis, the oscilloscope measures the time required for the power supply to rise to 90% of its steady-state output voltage after being powered on.

During shutdown analysis, it measures the time taken for the output voltage to drop to 10% of its maximum value after the power supply is turned off.

(1) Analysis Type

Click Type to select Turn On/Off.




(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

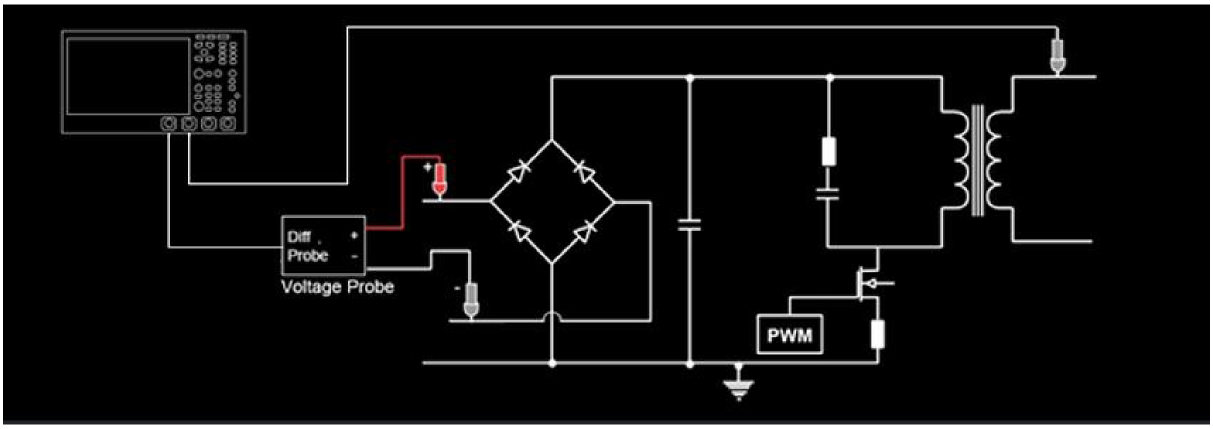
Click Wiring Diagram to display the wiring diagram of the startup/shutdown time. Follow the instructions in the figure below to make the wiring connections. To close the diagram, click close icon in the upper-right corner.

a. Voltage Probe

- Connect D+ to the live wire of the AC input.
- Connect D- to the neutral wire of the AC input.
- Select an appropriate probe attenuation ratio.

b. Voltage Probe (Passive or differential probe)

- Connect the probe to the power output terminal.
- Use Input Voltage and Output Voltage to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the input voltage source (C1 - C4). The voltage channel should be configured with the appropriate unit and probe attenuation ratio according to the connected voltage probe.

(5) Output Voltage

Click Output Voltage to select the output voltage source (C1 - C4). The voltage channel should be configured with the appropriate unit and probe attenuation ratio according to the connected voltage probe.

(6) Test Type

Click Test Type drop-down list and select the desired test type to turn on or turn off.

(7) Input Type

Click Input Type drop-down list and select the appropriate input type to AC or DC.

(8) Maximum Input Voltage

Set the maximum input voltage. The voltage range is from 1 V to 1 kV. The oscilloscope automatically adjusts the voltage channel range based on this value using the formula: Channel Range = Maximum Input Voltage / 6.

(9) Steady Output Voltage

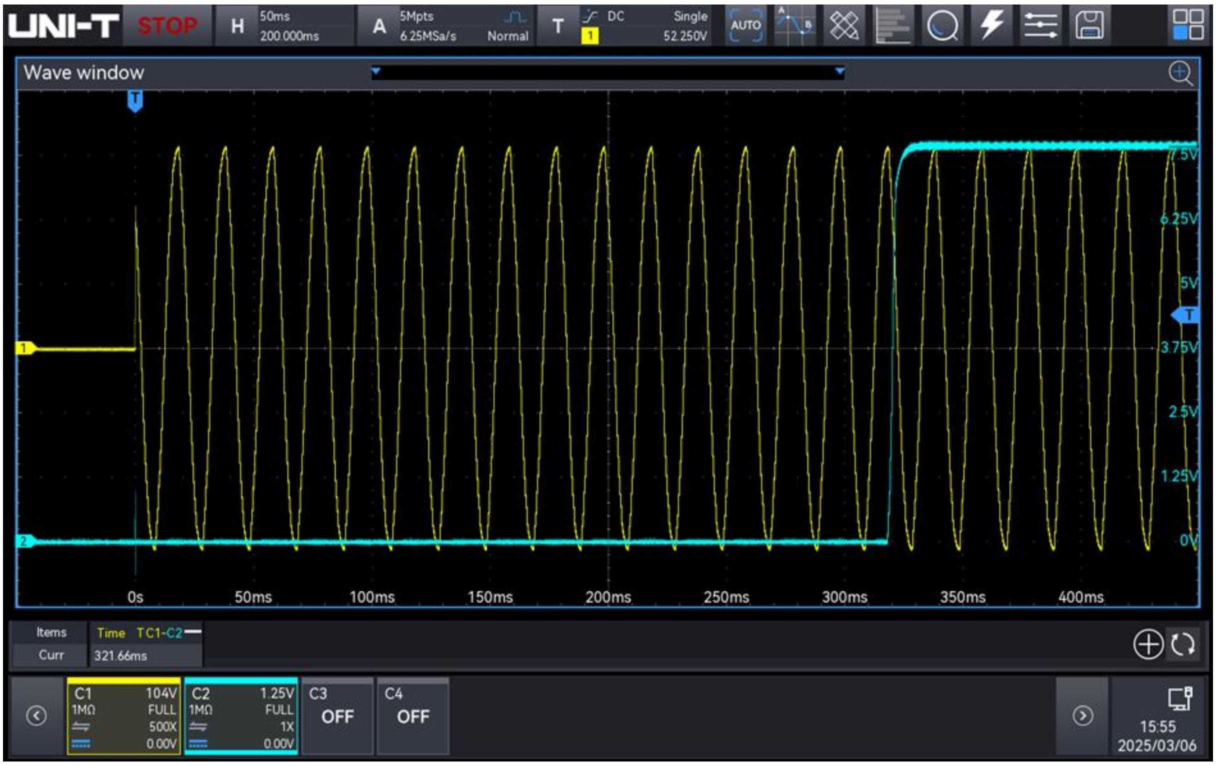
Set the steady-state output voltage. The voltage range is from -30 V to 30 V. The oscilloscope automatically adjusts the output voltage channel range based on this value using the formula: Channel Range = Steady Output Voltage / 6.

(10) Duration

Set the time span for the entire waveform display. The valid range is from 10 ns to 20 s. The oscilloscope automatically adjusts the time base range using the formula: Time Base = Duration / 10.

(11) Apply

Click Apply key and follow the on-screen instructions. Once the analysis is complete, the results are displayed.



(12) Startup/Shutdown Time Measurement Results

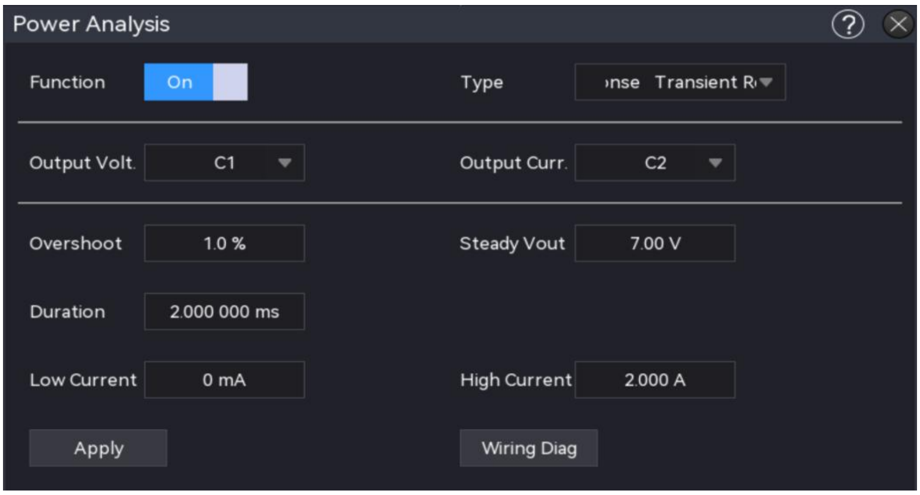
| | |
|---|---|
| Startup Time | The time it takes for the output voltage to rise to 90% of its steady-state value after the power supply is powered on. |
| Shutdown Time | The time it takes for the output voltage to fall to 10% of its maximum value after the power supply is turned off. |

# 11.11. Transient Response

Transient response analysis determines the response speed of a power supply's output voltage to changes in the output load. The response time is measured from the moment the output voltage first deviates from the stability band until it re-enters the stability band for the final time.

(1) Analysis Type

Click Type to select Transient Response.



(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

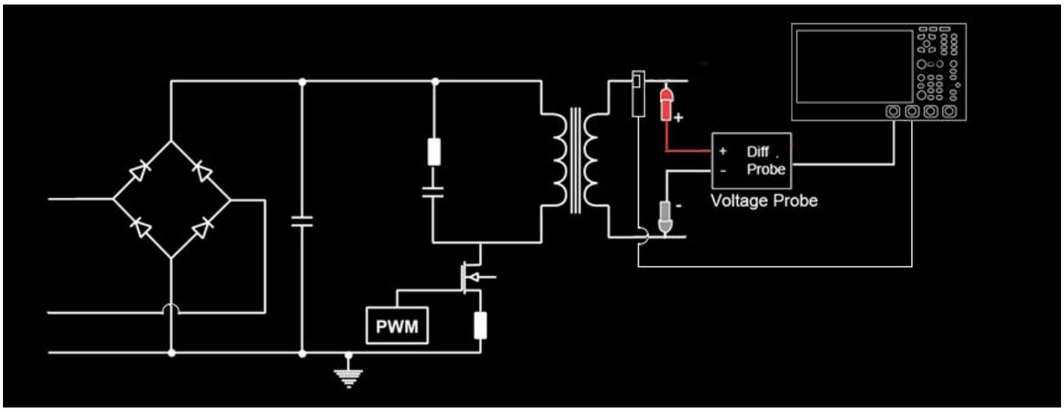
Click Wiring Diagram to display the wiring diagram of the transient response, please follow the instructions to make the wiring connection, as shown in the following figure. To close the diagram, click close icon in the upper-right corner.

a. Voltage Probe (Passive/Differential)

- Connect the voltage probe to the power output.

b. Current Probe

- Connect the current probe to the power output, the arrow indicates the current direction.
- Use Input Voltage and Output Voltage to specify the corresponding channels for the probes above.




(4) Output Voltage

Click Output Voltage to select the output voltage source (C1 - C4). The voltage channel should be configured with the appropriate unit and probe attenuation ratio according to the connected voltage probe.

(5) Output Current

Click Output Current to select the output current source (C1 - C4). The current channel should be configured with the appropriate unit and probe attenuation ratio according to the connected current probe.

(6) Overshoot

Set the overshoot percentage of the output voltage. This value is used to determine the stability band for transient response measurements and to adjust the oscilloscope's vertical scaling. The adjustable overshoot range is from 1% to 20%.

(7) Steady Output Voltage

Set the expected DC output voltage of the power supply in its stable state. The voltage range is from −30 V to 30 V.

(8) Duration

Set the time span for the entire waveform display. The valid range is from 10 ns to 20 s. The oscilloscope automatically adjusts the time base range using the formula: Time Base = Duration / 10.

(9) Low Current

Enter the low current value. This value represents the expected low current level either before or after the load change.

(10) High Current

Enter the high current value. This value represents the expected high current level either before or after a load change.

Note: The low and high current values are used to calculate the trigger level and to adjust the oscilloscope's vertical scaling. After a load change, when the current transitions from the low level to the high level (or from the high level to the low level), the oscilloscope triggers and performs the transient response settling time measurement.

(11) Apply

Click Apply key and follow the on-screen instructions. Once the analysis is complete, the results are displayed.


(12) Transient Response Measurement Results

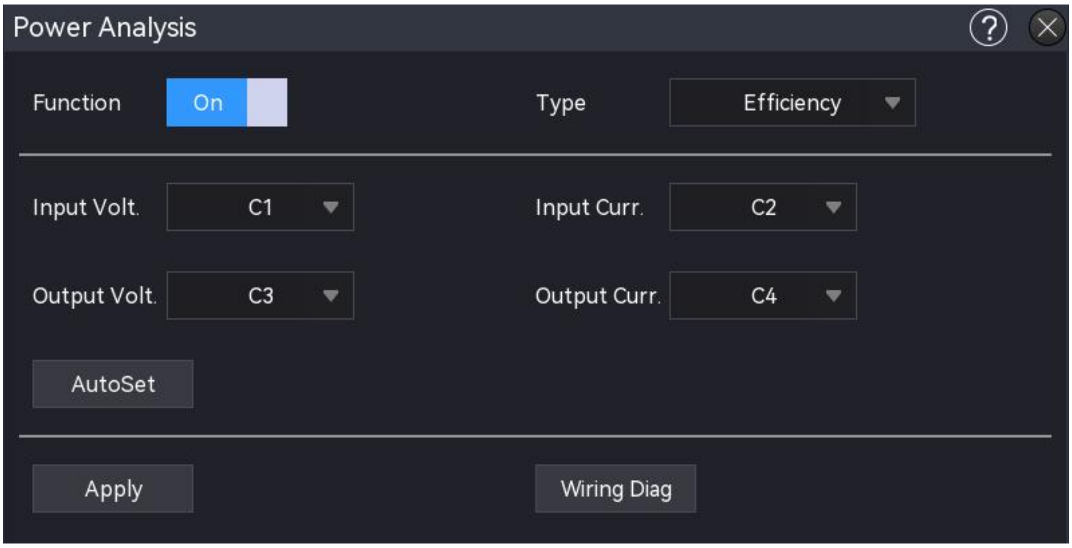
| | |
|---|---|
| Transient Response Time | The time required for the output voltage to recover and stabilize within the allowable error range, measured from the moment it deviates from the steady-state value due to a sudden change in the load current at the power supply output. |

## 11.12. Efficiency

Efficiency analysis evaluates the overall efficiency of the power supply by measuring both the output power and input power. This analysis requires a 4-channel oscilloscope, as it must measure the input voltage, input current, output voltage, and output current.

(1) Analysis Type

Click Type to select Efficiency.




(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

Click Wiring Diagram to display the wiring diagram of efficiency. Follow the instructions in the figure below to make the wiring connections. To close the diagram, click close icon in the upper-right corner.

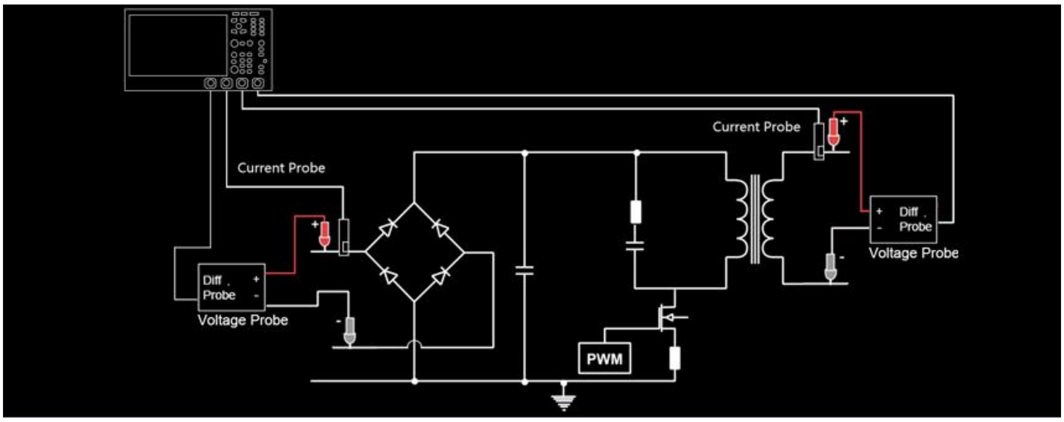
a. Voltage Probe 1
- Connect D+ to the live wire of the AC input.
- Connect D- to the neutral wire of the AC input.
- Select an appropriate probe attenuation ratio.

b. Current Probe 1
- Connect the current probe to the power output. The arrow indicates the direction of current flow.

c. Voltage Probe 2
- Connect D+ to the power output terminal.
- Connect D- to the power ground.
- Select an appropriate probe attenuation ratio.

d. Current Probe 2

- Connect the current probe to the power output. The arrow indicates the direction of current flow.
- Use Input Voltage and Output Voltage to specify the corresponding channels for the probes above.




(4) Input Voltage

Click Input Voltage to select the output voltage source (C1 - C4). The voltage channel should be configured with the appropriate unit and probe attenuation ratio according to the connected voltage probe.

(5) Input Current

Click Input Current to select the output current source (C1 - C4). The current channel should be configured with the appropriate unit and probe attenuation ratio according to the connected current probe.

(6) Output Voltage

Click Output Voltage to select the output current source (C1 - C4). The voltage channel should be configured with the appropriate unit and probe attenuation ratio according to the connected voltage probe.

(7) Output Current

Click Output Current to select the output current source (C1 - C4). The current channel should be configured with the appropriate unit and probe attenuation ratio according to the connected current probe.

(15) Auto Setting

After configuring the input voltage, input current, output voltage, and output current sources, the Autoset function will automatically open the voltage and current channels, set the channel units to V and A.

(16) Apply

Click Apply key and follow the on-screen instructions. Once the analysis is complete, the results are displayed.


(17) Efficiency Measurement Results

| | |
|---|---|
| Input Power | The actual power supplied to the power supply, measured in watts (W). |
| Output Power | The actual power delivered by the power supply, measured in watts (W). |
| Efficiency | The ratio of output power to input power. |

# 12. Cursor Measurement

- Time Measurement
- Voltage Measurement
- Screen Measurement

Use the cursor to measure X axis (time) and Y axis (voltage) of the waveform. The cursor measurement supports simultaneous measurement of multiple channels, as well as Math waves. The source, test type and mode can be configured in the cursor measurement menu.

(1) Source: Sets the source for the cursor measurement, C1-C4 and M1-M4 can be selected.

(2) Test type: Time, voltage, and screen measurements.

(3) Synchronized move: Configures the cursor time signature tracking method. It can be set to on or off.

- OFF: The two cursors can be adjusted independently.
- ON: The two cursors move together in a synchronized manner.

(4) Horizontal cursor: Sets the horizontal cursor position relative to the time base. Two modes can be selected: position fixed and delay fixed.

- Position fixed: The default display is at ±2 divisions. When the time base is changed, the positions of the AX and BX cursors on the screen remain unchanged.
- Delay fixed: The default display is at ±2 divisions. When the time base is modified, the AX and BX cursor positions are stretched or compressed accordingly.

(5) Cursor result window: Supports hover or fixed display modes.

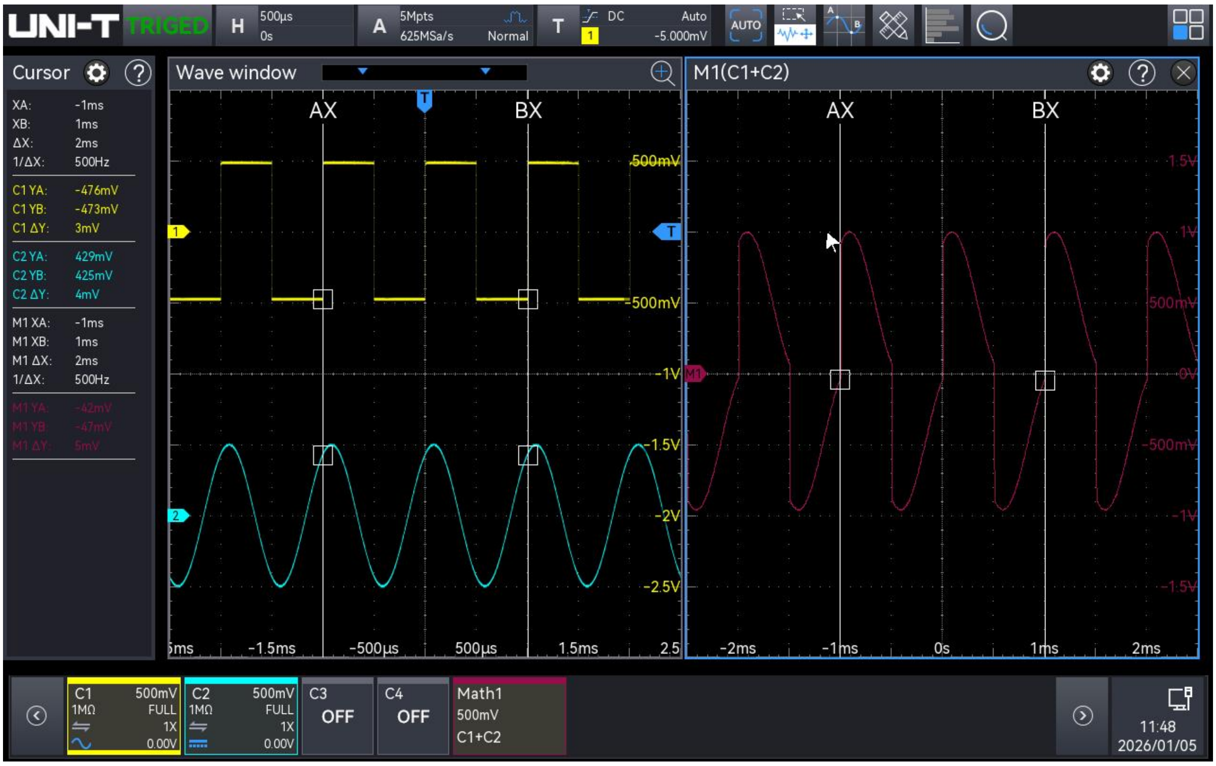
- Hover: Drag the window to display it anywhere on the screen.
- Fixed: Drag the window to the left side of the screen to fix it in place.

Access the cursor measurement menu using the following method.

- Click Home icon in the top right-corner and select the counter icon to enable the cursor measurement.
- If the cursor icon is added to the toolbar, click cursor icon in the toolbar in the top right-corner to enable the cursor measurement.
- If cursor measurement results appear, click icon to open the cursor measurement.

## 12.1. Time Measurement

In the Cursor menu, open the cursor and click Time, and then check the Source to perform the cursor measurement, as shown in the following figure.



Cursor results info window: "X" represents the results of channel time measurement. "Y" represents the voltage measurement result at the intersection of the open channel and the cursor.

The Math wave supports split-screen display. When measuring a Math wave with the cursor, it is also displayed in split-screen mode, allowing each Math channel's cursor to be adjusted independently without affecting the others.

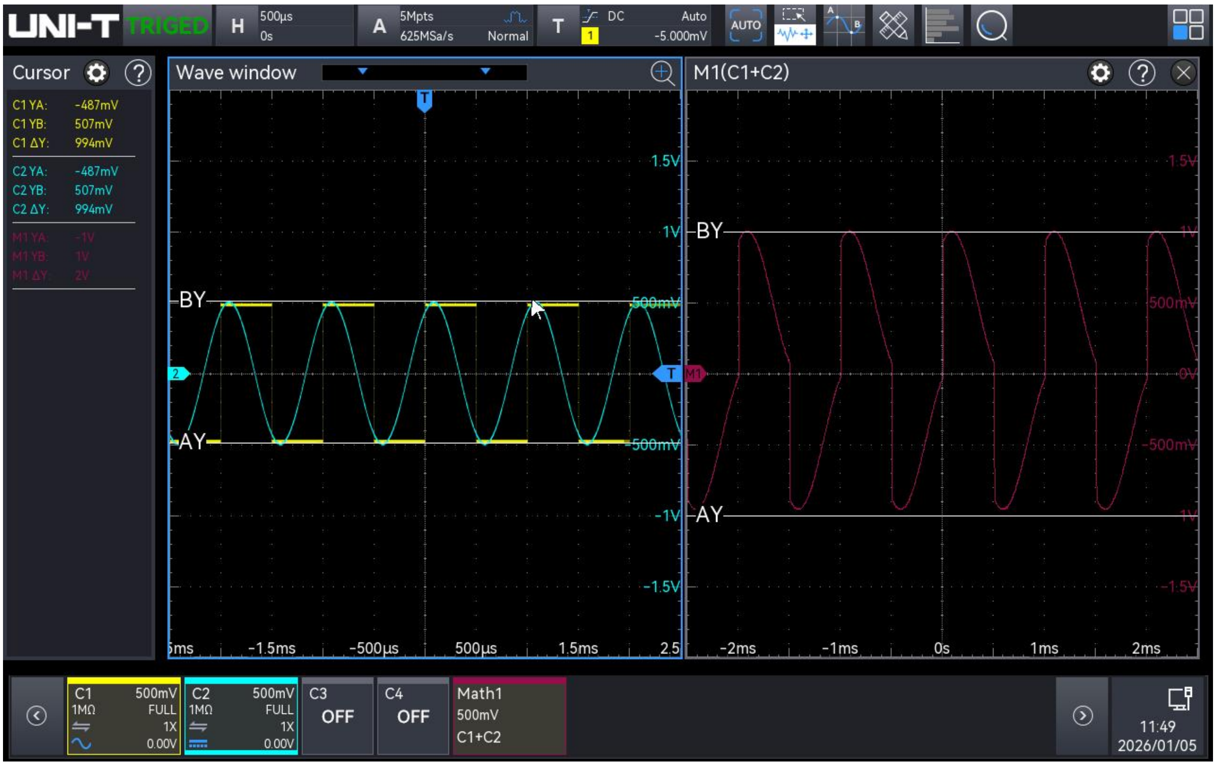
The position of AX and BX cursors can be set using the following methods.

- Use the cursor to select AX or BX, then drag it with the mouse to the correct position.

## 12.2. Voltage Measurement

The voltage measurement is the same as the time measurement, adjusting the vertical position of the cursor and measuring the voltage of each cursor.

In the Cursor menu, open the cursor and click Voltage, and then check the Source to perform the cursor measurement, as shown in the following figure.




Cursor results info window in top left corner: "Y" represents the results of channel voltage measurement.

The Math wave supports split-screen display. When measuring a Math wave with the cursor, it is also displayed in split-screen mode, allowing each Math channel's cursor to be adjusted independently without affecting the others.

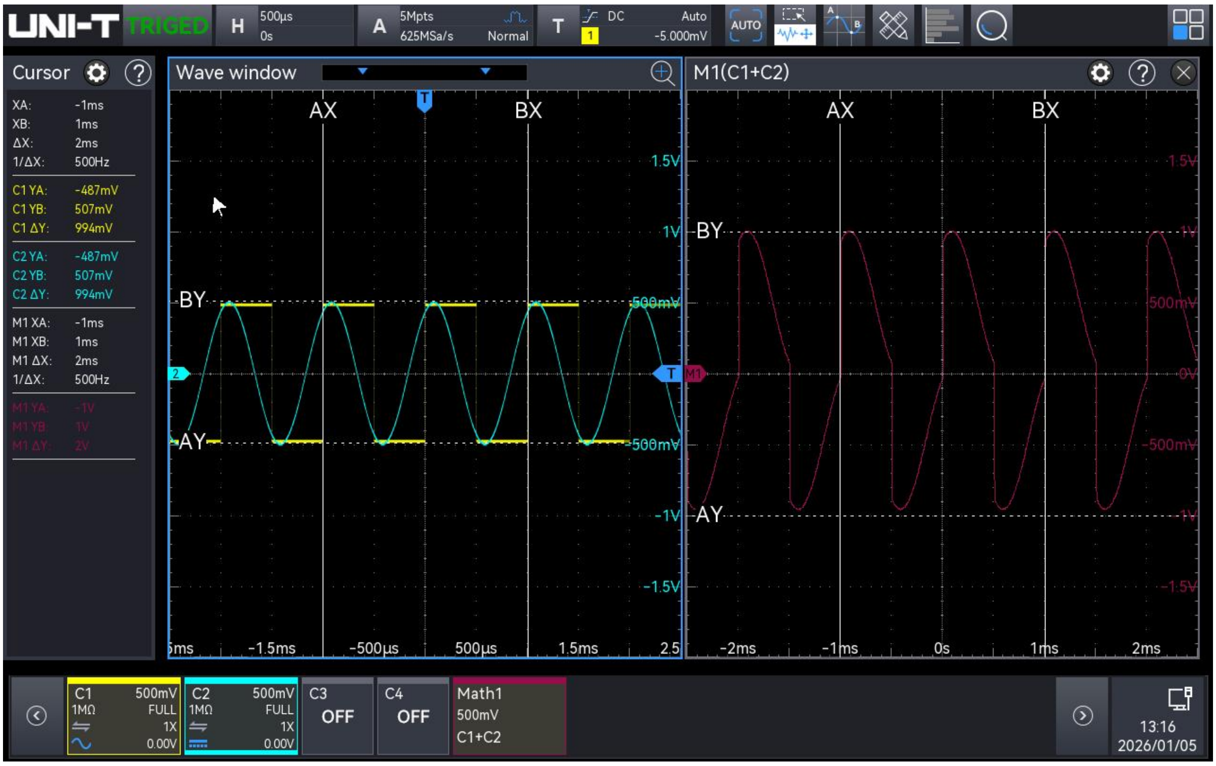
The position of AY and BY cursors can be set using the following methods.

- Use the cursor to select AY or BY, then drag it with the mouse to the correct position.

## 12.3. Screen Measurement

The screen measurement supports setting the time cursor and voltage cursor, i.e. the time and voltage measurement can be performed at the same time.

In the Cursor menu, open the cursor and click Screen, and then check the Source to perform the cursor measurement, as shown in the following figure.




Cursor results info window in top left corner: “X” represents the results of channel time measurement, “Y” represents the results of voltage measurement.

The Math wave supports split-screen display. When measuring a Math wave with the cursor, it is also displayed in split-screen mode, allowing each Math channel’s cursor to be adjusted independently without affecting the others.

The position of AX, BX, AY, and BY cursors can be set using the following methods.

- Use the cursor to select AX, BX, AY, or BY, then drag it with the mouse to the correct position.

# 13. Sampling System



Sampling is the process of converting a signal from an analog input channel into discrete data points through an analog-to-digital converter (ADC).

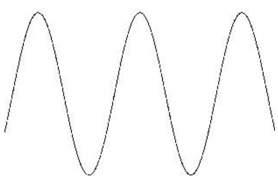
Access the sampling setting menu using the following method.

- Click Acquire sampling info label at the top of the screen to enter the Sampling setting menu, as shown in the following figure.

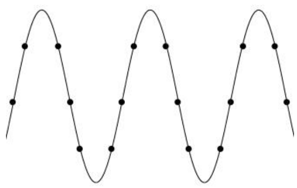

## 13.1. Sampling Rate

(1) Sampling and Sampling Rate

Sampling indicates that the oscilloscope is to take a sample from the input an analog signal and convert the sample to digital data, and then gather the digital data to waveform records. The waveform records will be saved in the storage memory.



Analog Input Signal



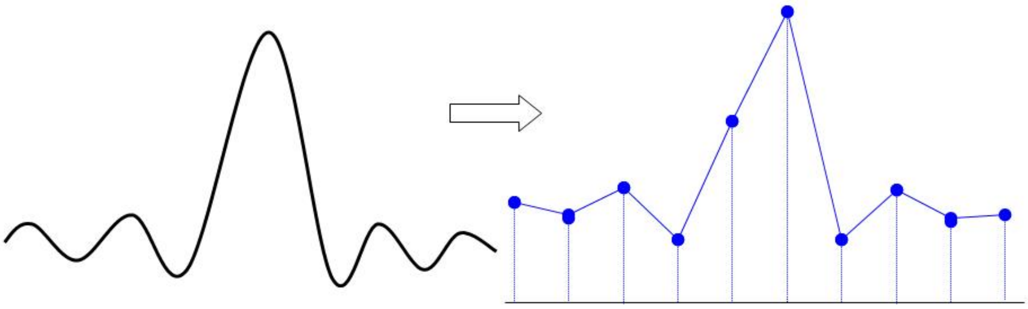
Sampling Point

Sampling rate indicates the time interval between two sampling points. The maximum sampling rate of the UPO6000LP series high-resolution oscilloscopes is 5 GSa/s.

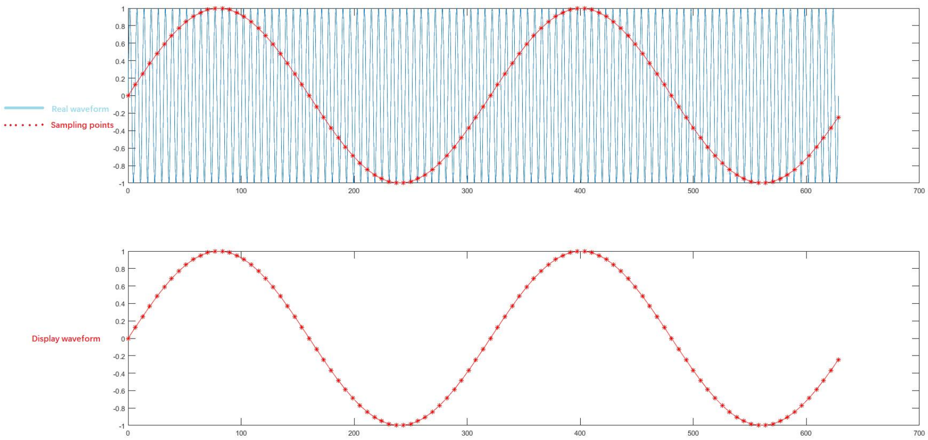
The sampling rate will change with the time base scale and memory depth. The real-time sampling rate is displayed in the Acquire sampling label at the top of the screen, the horizontal time base can be adjusted using the horizontal scale or by changing the memory depth.

(2) Effect of Low Sampling Rate

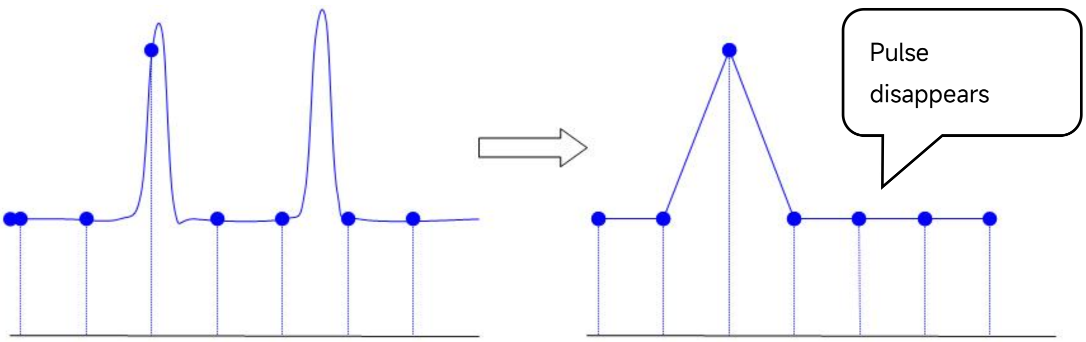
- Waveform distortion: Due to a low sampling rate, waveform details might be lost, and the sampled waveform may differ significantly from than the actual signal, as shown in the following figure.



- 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 a low sampling rate, the waveform does not reflect all the actual signals, as shown in the following figure.



## 13.2. Acquisition Mode

The acquisition mode controls how the oscilloscope uses the sampling rate to create a waveform. In the Sampling menu, click Acquisition Mode to select the mode.

(1) Normal Sampling

The oscilloscope samples the signal and reconstructs the waveform with equal time interval in normal mode. For most of waveform, this mode can produce the optimal display effect.

(2) Peak Sampling

The oscilloscope finds the maximum and minimum of the input signal from every sampling interval and using this value to display the waveform. Thus, the oscilloscope can get and display the narrow pulse, otherwise, this narrow pulse will be missed in normal sampling. In this mode, the noise will also look larger

(3) High Resolution

The oscilloscope averages the adjacent point of sampling waveform; it can reduce the random noise of input signal and generate a smoother waveform on the screen.

(4) Average

The oscilloscope obtains several waveforms and calculates its averaged value, and then displays the final waveform. This mode can reduce the random noise.

To observe the waveform by changing the acquisition method. If the signal contains large noise, the waveform does not average and the waveform adopts 32 times averaged as shown in the following figure. The higher the average value, the lower the noise and the vertical resolution will be much higher, but the change in the waveform will also be slower.

The “Average Number” is enabled when the acquisition mode is average, and the range can be set from 2 to 8192, each increment is a power function of 2. The default averaging number of the oscilloscope is 2.

By changing the acquisition mode setting to observe the waveform changes. If the signal contains large noise, when the average mode is not adopted and when the 32 times average mode is adopted, the sampled waveform is shown in the following figure.

Not Averaged Waveform

Waveform of 32 Times Averaged

**Note**: Average and high resolution uses different average methods. The former is multiple sampling averaged, the latter is single sampling averaged.

(5) Enhanced Resolution

If the signal-to-noise ratio and the effective resolution of the oscilloscope need improvement, select the enhanced resolution acquisition method, then set the desired number of enhanced resolution bits. For more details, refer to the section of Enhanced Resolution (ERES).

## 13.3. 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 capacity of the memorizer.

The relation of the memory depth, sampling rate and sampling time:

Memory depth = Sampling rate (Sa/s) × Sampling time (s/div×div)

In the Sampling menu, click Memory Depth to select the memory depth. The real-time memory depth is displayed in the Acquire sampling label at the top of the screen.

The UPO6000LP supports the memory depth as shown in the table below: Auto (5 Mpts), 25 kpts, 250 kpts, 500 kpts, 5 Mpts, 50 Mpts, 100 Mpts, and MAX (500 Mpts).

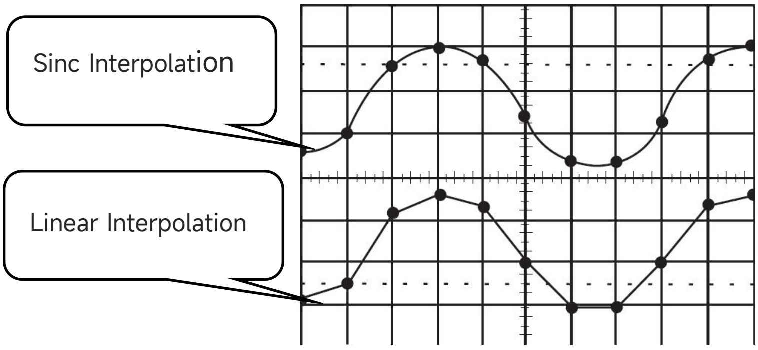
## 13.4. Interpolation Method

With real-time sampling, the oscilloscope acquires discrete samples of the displayed waveform. In general, waveforms displayed by dots are difficult to observe. The digital oscilloscope usually uses interpolation to improve the visualization of the signal. The interpolation is a method of "connect each sampling point" and using some points to calculate the waveform. With real-time sampling using interpolation, even if the oscilloscope only captures a small number of sampling points in a single pass, interpolation can be used to fill in the gaps between points and reconstruct an accurate waveform.

The interpolation is divided into Sinc interpolation (sinx/x) and linear interpolation.

Linear interpolation: Straight lines are directly connected at adjacent sampling points. This method is limited to the reconstruction of pure edge signals, such as square waves.

Sinc interpolation (sinx/x): Use a curve to connect the sampling points, this is more common. Sinc interpolation uses mathematical processing to calculate the result in the actual sampling point interval. This method bends the signal waveform to produce a more realistic common shape than pure square waves and pulses. When the sampling rate is 3 to 5 times the system bandwidth. Sinc interpolation is recommended. The figure below shows a very different display using the two interpolation methods.




Interpolation Method Comparison

## 13.5. Enhanced Resolution (ERES)

In the Acquire menu, select Enhanced Resolution and set the Eres bit count to off, 1 bit, 1.5 bits, 2 bits, 2.5 bits, 3 bits, or 4 bits.

In Enhanced Resolution mode, the oscilloscope digitally filters the sampling points, reducing broadband random noise on the input signal and improving the signal-to-noise ratio, thereby increasing the effective resolution (ENOB) of the oscilloscope. The UPO6000LP Eres processing is implemented by a hardware engine, allowing the oscilloscope to maintain a high waveform refresh rate even when Eres is enabled.

Unlike other acquisition methods, Eres does not require the signal to be periodic or triggered stably. However, because it relies on digital filtering, enabling Eres reduces the oscilloscope's system bandwidth. The higher the number of enhanced resolution bits, the lower the bandwidth.

The table below shows the relationship between the number of Eres bits and the resulting bandwidth.

| Eres Bit | - 3 dB Bandwidth |
|---|---|
| 1 bit | 0.6*Sampling rate |
| 1.5 bits | 0.2* Sampling rate |
| 2 bits | 0.1* Sampling rate |
| 2.5 bits | 0.032* Sampling rate |
| 3 bits | 0.012* Sampling rate |
| 4 bits | 0.005* Sampling rate |

# 14. Display System

In the Display menu, set the waveform type, persistence, waveform view scale, grid type, grid brightness, waveform brightness, window transparence, staff, and color temperature.

Access the Display menu using the following method.

- Click Home icon in the top right-corner and select the display icon to open the display menu.
- If the display function is added to the toolbar, click display icon in the toolbar in the top right-corner to enter the display menu.


## 14.1. Display Type

In the Display setting menu, the display type can be set to vector or point. The default setting is vector display.

- Vector display: The sampling points are shown in the connecting line. This mode provides the most realistic waveform in most cases and makes it easy to view the steep edges of the waveform (e.g. square wave).
- Point display: Directly display the sampling point.

## 14.2. Persistence

Set the persistence in the Display menu, once the persistence is enabled, the oscilloscope uses the new acquired waveform to refresh the display but will not immediately delete the old acquired waveform. The old acquired waveform is displayed with the low brightness color, and the new acquired waveform is displayed with the normal color and brightness.

The persistence can be set to auto, 50 ms, 100 ms, 200 ms, 500 ms, 1 s, 2s, 5 s, 10 s, 20 s, infinite, and off. The default is auto.

- Auto: When the trigger type is set to decoding type, the waveform is automatically displayed as a single amplitude.
- Adjustable persistence (50 ms…10 s, 20 s): In different persistence, the oscilloscope updates the display with the newly acquired waveforms, and the acquired waveforms are cleared after the appropriate time. Glitch with slower changes or lower probability of occurrence can be observed.
- Infinite: Once Infinite is selected, the oscilloscope never clears the acquired waveform. Use infinite persistence to measure noise and jitter, and to capture episodic events.
- OFF: Only a single amplitude waveform is displayed without persistence.
- Stop persistence: When the Stop Persistence is checked, the persistence effect remains on the screen after the oscilloscope is stopped. When Stop Persistence is unchecked, only a single waveform is displayed on the screen without the persistence effect.

## 14.3. Waveform View Scale

For stack and overlay modes, the vertical and horizontal scales can be set to removable or fixed. The default setting is removable.

Removable: The scale moves with the waveform, and the scale units are displayed.

Fixed: The grid scale remains stationary when the waveform's vertical or horizontal position is adjusted.

## 14.4. Grid Type

In the Display setting menu, four grid types can be set: grid display, full display, frame, and crosshair.

- Grid: Displays a grid with 8 rows and 14 columns.
- Full scale: Displays in crosshair and gird.
- Frame: No crosshair, no gird display.
- Crosshair: Divides the screen into 4 parts.

## 14.5. Grid Brightness

In the Display setting menu, drag the scroll bar to set the grid brightness.

The brightness range: 0%-100%. The default is 50%.

## 14.6. Waveform Brightness

In the Display setting menu, drag the scroll bar to set the waveform brightness.

The brightness range: 1%-100%. The default is 50%.

## 14.7. Window Transparence

Set the window transparence for all pop-up info window (e.g. Cursor menu, Waveform view menu, etc)., to an appropriate value for better visibility of the measured data. Drag the scroll bar to adjust the transparency.

The brightness range: 0%-100%, 0% represents non-transparent, 100% represents fully transparent, and the default is 50%.

**Note:** The auxiliary window is non-transparent by default.

## 14.8. Staff

The staff refers to the graduations displayed on the horizontal and vertical axes of the display screen. Its display can be set to display or hide.

Display: The graduations on the horizontal and vertical axes are displayed in all waveform windows.

Hide: The graduations on the horizontal and vertical axes are hidden in all waveform windows.

## 14.9. Color Temperature

In the Display setting menu, turn on/off color temperature. The default setting is OFF.

Once the color temperature is enabled, the different color represents the number of data acquired or the probability distribution on the screen.

# 15. Save and Load



Users can save the current oscilloscope settings, waveforms, screen images and parameters in various formats to internal memory or external USB storage devices and reload the saved settings or waveforms as required. Updated versions can also be loaded into the system to upgrade the instrument. In addition, the user can copy, delete and rename files of specified types in internal memory or external USB memory using the Disk Management menu.

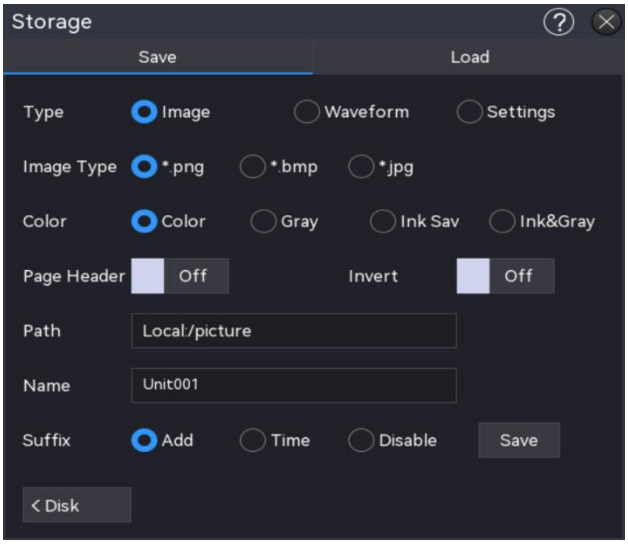
**Note**: This oscilloscope has four USB HOST ports (two on the front panel and two on the rear panel) for connecting USB flash drives for external storage. It supports only USB flash drives formatted with FAT32, NTFS, or EXFAT.

## 15.1. Storage Menu

Access the Storage menu using the following method.

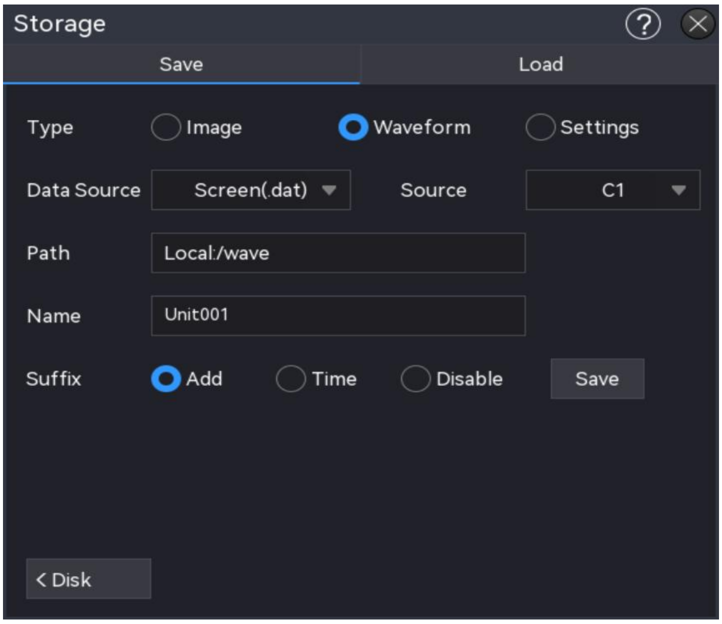
- Click Home icon in the top right-corner and select the storage icon to open the storage menu.
- If the display function is added to the toolbar, click storage icon in the toolbar in the top right-corner to enter the storage menu.

The storage setting menu has two sub-menu: storage and load. Select a submenu to adjust its settings.




## 15.2. Save Waveform

In the Save submenu, select Waveform to open the waveform save settings menu. The channel information of the selected signal source (such as vertical position and horizontal timing) will be saved to internal or external memory.




Waveform setting menu: Data source, source, save path, filename, suffix, time, disk management, and save.

(1) Data Source

Click Data Source to select the data type of the waveform to save, it has three types: screen (.dat), deep memory (.csv). Screen (.dat) and Deep Memory (.csv) files can be opened using the host software analyzer. Arbitrary Wave (.bsv) files can be loaded into either the signal source or the oscilloscope's built-in generator.

(2) Source

Click Source to select the source of waveform to be saved. For waveform data storage, only open sources are supported. When saving .dat and .bsv, the source can select C1 - C4 and M1 - M4. When saving .csv, the source can select C1 - C4, M1 - M4, or all open analog channels.

(3) Save Path

Double-click Save Path input field to open the file browser menu, and select the save directory in the file browser menu, then click Enter key to confirm the save path. For details on using the file browser, refer to the section of File Browser. When no USB flash drive is connected, the default save path is Local:/wave. When a USB flash drive is detected, UDISK: is selected as the default save path.

(4) Filename

Double-click Filename input field to open the numeric keypad and set the filename. For details on using the numeric keypad, refer to *Enter Character String* in the section of 5.7 Parameter Setting.

(5) Suffix

Tap to select the suffix to Forbid, Time, or Accumulate. The filename of the picture will be saved with the selected suffix to the internal or external storage.

- Forbid: Saves the file with the filename only, without any suffix.
- Time: Saves the file with the current system time added as a suffix to the filename.
- Accumulate: Saves the file with an incrementing number added as a suffix to the filename, starting from 0001.

(6) Time

Adds a time column to the saved CSV file. This option is available when CSV is selected as the data format. When the Time Save checkbox is enabled, the saved CSV file contains both time and voltage columns. When it is disabled, the CSV file contains only the voltage column.

(7) Disk Management

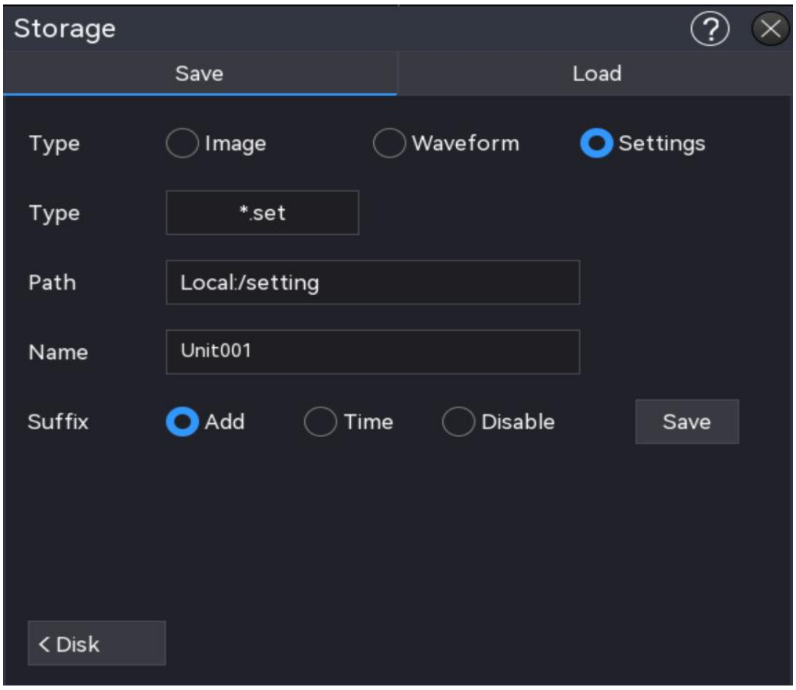
Click Disk to open the file browser. For details on using the file browser, refer to File Browser.

(8) Save

Click Save to store the waveform file using the current settings. A save result message is displayed.

## 15.3. Save Setting

In the Save submenu, select Settings. The oscilloscope saves the current settings in .set format to internal or external storage. The saved settings can be loaded as needed.




Save setting menu: File type, save path, filename, suffix, disk management, and save.

(1) File Type

Click Data Type to select the data type of waveform to be saved, *.set can be selected.

(2) Save Path

Double-click Save Path input field to open the file browser menu, and select the save directory in the file browser menu, then click Enter key to confirm the save path. For details on using the file browser, refer to the section of File Browser. When no USB flash drive is connected, the default save path is Local:/wave. When a USB flash drive is detected, "UDISK:" is selected as the default save path.

(3) Filename

Double-click Filename input field to open the numeric keypad and set the filename. For details on using the numeric keypad, refer to *Enter Character String* in the section of 5.7 Parameter Setting.

(4) Suffix

Tap to select the suffix as Forbid, Time, or Accumulate. The filename of the picture will be saved with the selected suffix to the internal or external storage.

- Forbid: Saves the file with the filename only, without any suffix.
- Time: Saves the file with the current system time added as a suffix to the filename.
- Accumulate: Saves the file with an incrementing number added as a suffix to the filename, starting from 0001.

(5) Disk Management

Click Disk to open the file browser. For details on using the file browser, refer to the section of

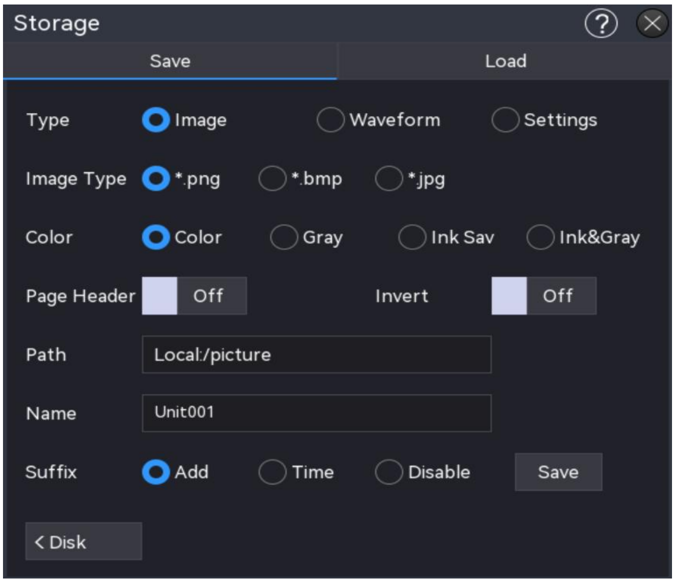
File Browser.

(6) Save

Click Save to store the waveform file using the current settings. A save result message is displayed.

## 15.4. Save Image

In the Save submenu, select Image to open the settings menu. The oscilloscope saves the current image settings to internal or external storage.




Picture save setting menu: Image type, color, page header, inverse color, save path, filename, disk management, and save.

(1) Image Type

Click Image Type to select the format: *.png, *.bmp , or *.jpg. The screen image will be saved with the selected format to internal or external storage.

(2) Color

Click Color to select the image color for saving.

| | |
|---|---|
| Color | The oscilloscope saves the screenshot in the currently displayed colors. |
| Save ink | The oscilloscope saves the screenshot with a light background instead of the default dark background to reduce ink usage. |
| Grey | The oscilloscope saves the screenshot in grayscale. |
| Grey & Save ink | The oscilloscope saves the screenshot in grayscale with a light background instead of the default dark background. |

(3) Page Header

Click Page Header to turn the page header on or off.

ON: The instrument model and image data are displayed in the page header.

OFF: No information is displayed in the page header.

(4) Inverse Color

Click Inverse Color to turn the inverse color function on or off.

(5) Save Path

Double-click Save Path input field to open the file browser menu and select the save directory in the file browser menu, then click Enter key to confirm the save path. For details on using the file browser, refer to the section of File Browser. When no USB flash drive is connected, the default save path is Local:/wave. When a USB flash drive is detected, UDISK: is selected as the default save path

(6) Filename

Double-click Filename input field to open the numeric keypad and set the filename. For details on using the numeric keypad, refer to *Enter Character String* in the section of 5.7 Parameter Setting.

(7) Suffix

Tap to select the suffix to Forbid, Time, or Accumulate. The filename of the picture will be saved with the selected suffix to the internal or external storage.

- Forbid: Saves the file with the filename only, without any suffix.
- Time: Saves the file with the current system time added as a suffix to the filename.
- Accumulate: Saves the file with an incrementing number added as a suffix to the filename, starting from 0001.

(8) Disk Management

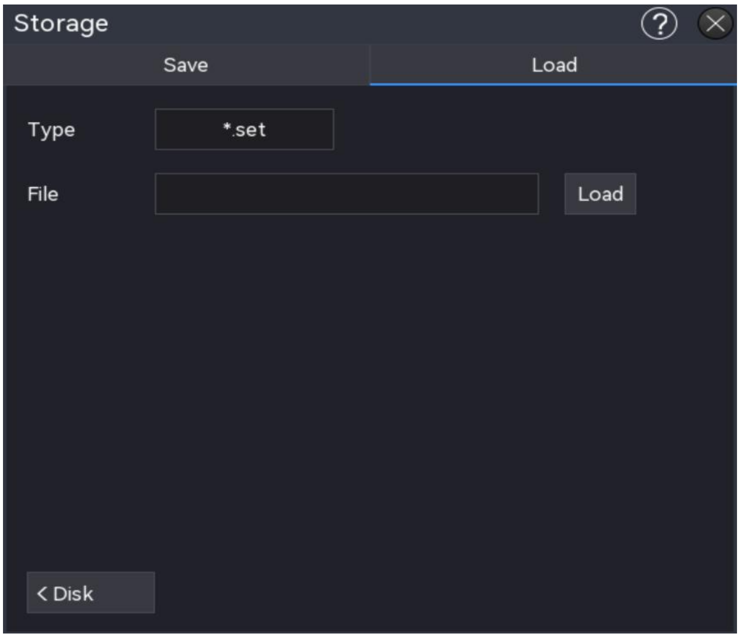
Click Disk to open the file browser. For details on using the file browser, refer to the section of File Browser.

(9) Save

Click Save to store the picture file using the current setting. A save result message is displayed.

## 15.5. Load Setting

In the Load submenu to set the load settings. To load the saved setting file to the oscilloscope.




Load setting menu: File type, file, load, and disk management.

(1) File Type

Select the type of loading setting file, the default type is .set, and cannot be changed.

(2) File

Click File to open the file browser and select the setting file to load. For details on using the file browser, refer to the section of File Browser.

(3) Load

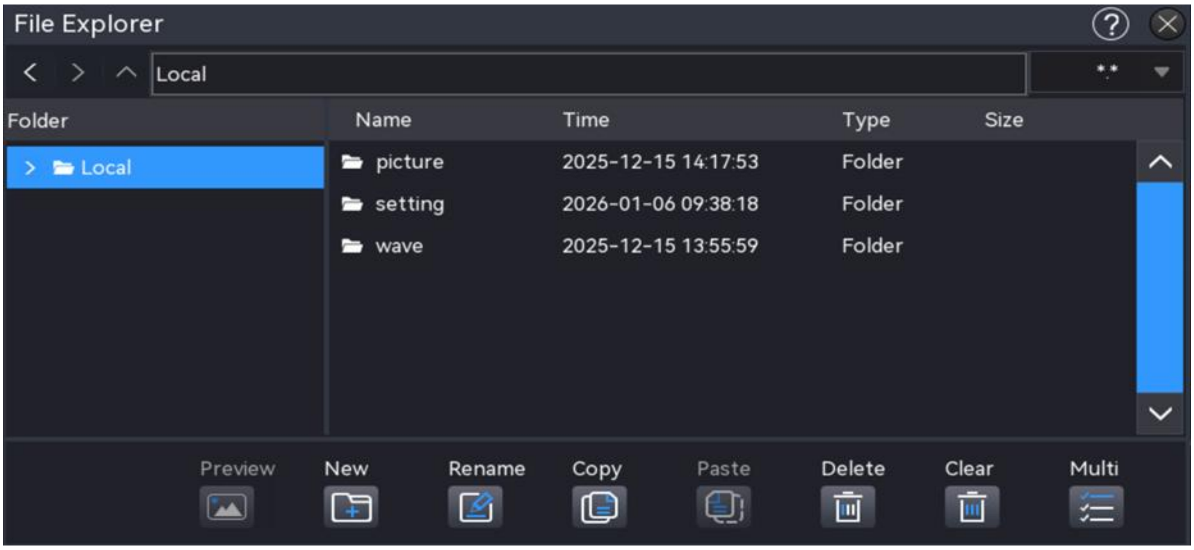
Click Load to load the selected setting file.

(4) Disk Management

Click Disk to open the file browser. For details on using the file browser, refer to the section of File Browser.

## 15.6. File Browser

In the Save menu, select Disk at the bottom left corner to open the Disk Management menu, as shown in the following figure




(1) Select Disk

Before connecting an external storage, please make sure that USB (FAT32, Flash) is correctly connected. The save interface displays the contents of Local by default. If an external storage is connected, click hardware icon in the top left in the Save menu to select Local or USB. If USB is selected, the save interface will display the contents of USB.

After selecting a file path, press the key < , > in the upper left corner of the popup box to undo the path selection, or press the key ^ to navigate to the previous directory.

**Note**: The USB name appears as the USB flash drive's name and cannot be modified.

(2) Preview

To preview locally saved images, select an image and click Preview. If multiple images are available, click < on the left side and > on the right side to switch between them.

(3) New File

Click New icon to open the numeric keypad, add a new file and enter the new filename. For details on using the numeric keypad, refer to *Enter Character String* in the section of 5.7 Parameter Setting.

(4) Rename File or Directory

Click Rename icon to open the numeric keypad, rename the file or directory.

(5) Copy File to the Specified Directory

Select the specified file or file folder, click Copy icon and enter the specified file folder and then Click paste to complete this setting.

(6) Paste

Select the specified file or file folder, click Paste icon to copy the file or file folder to the specified file folder.

(7) Delete

In the current directory, check the file or directory to delete, and Click Delete icon , and then Click Enter to complete this setting, click Cancel to cancel this setting.

(8) Clear All

Clear all files and file folders in the current directory. Click Clear icon and then Click Enter to complete this setting, click Cancel to cancel this setting.

(9) Multiple Choice

This oscilloscope supports selecting multiple files or file folders at the same time. Click multiple-choice icon , click check box to the right of the file, and it will be displayed in the selected state when the selection is complete, and then deselect it by clicking the check box again, and the check box will return to its original state . The user can also select all files and directories under the current disc by clicking the check box in the top right-corner of the menu. Clicking the check box again will cancel the Select All operation.

(10) Suffix Type

Click file browser in the top right-corner and select the suffix type,

*.*, .png, .bmp, .jpg, .csv, .bsv, .dat, or .set, *.* represents all file types.

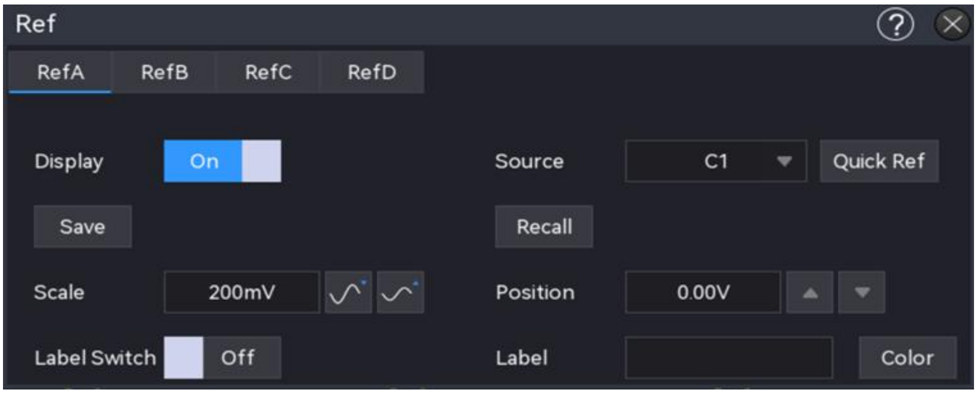
# 16. Reference Waveform

During the actual test, the user can compare the signal waveform with the reference waveform to determine the cause of the fault.

Access the reference waveform menu using the following method.

- Click Home icon  in the top right-corner and select the reference icon  to open the reference waveform.
- If the reference waveform is added to the toolbar, click reference icon  to enter the reference waveform.

This oscilloscope provides 4 reference waveforms (RefA-RefD), the setting of each reference waveform is the same. This chapter describes how to set the reference waveform using RefA as an example.




## 16.1. Display

Click Display to turn the reference waveform display on or off.

## 16.2. Source

Click Source to select the waveform source to be saved. Only the opened sources are supported to save waveform data, C1 - C4， M1 - M4 can be selected.

## 16.3. Save

(1) File Path

Double-click File Path input field to open the file browser menu, and select the save catalogue in the file browser menu, then click Enter key to confirm the save path. For details on using the file browser, refer to the section of File Browser. When no USB flash drive is connected, the default save path is Local:/wave. When a USB flash drive is detected, UDISK: is selected as the default save path.

(2) Filename

Double-click Filename input field to open the numeric keypad and set the filename. For details on using the numeric keypad, refer to *Enter Character String* in the section of 5.7 Parameter Setting.

(3) Waveform Type

The default waveform type is *.dat, and cannot be changed.

## 16.4. Load

Click Load to open the waveform load menu, then double-click import file filed to open the file browser. In the file browser, select the waveform file to load and click Recall to complete the setting.

## 16.5. Quick Reference

Click Quick Reference to load the currently selected source for reference. This action does not save the waveform as a .dat file. The Quick Reference function is only available for the current source, it should be re-selected after deletion.

## 16.6. Vertical Scale

Set the vertical scale of Ref wave in the display window, it can be set using the following steps.

- Tap the vertical scale icon , on the right to increase or decrease the vertical scale.
- Click number input field to open the numeric keypad to enter the specified numeric value.

## 16.7. Vertical Position

Set the vertical offset of Ref wave in the display window, it can be set using the following steps.

- Tap the arrow key , on the right of the vertical position to increase or decrease the vertical position.
- Click number input field to open the numeric keypad and set the specified numeric value.

## 16.8. Label

Click Label to turn the channel label on or off. The label can be customized, double-click Label input field to open the numeric keypad to enter the character string.


## 16.9. Channel Color

Set the color for the Ref waveform and label. Click Color to open the setting menu.

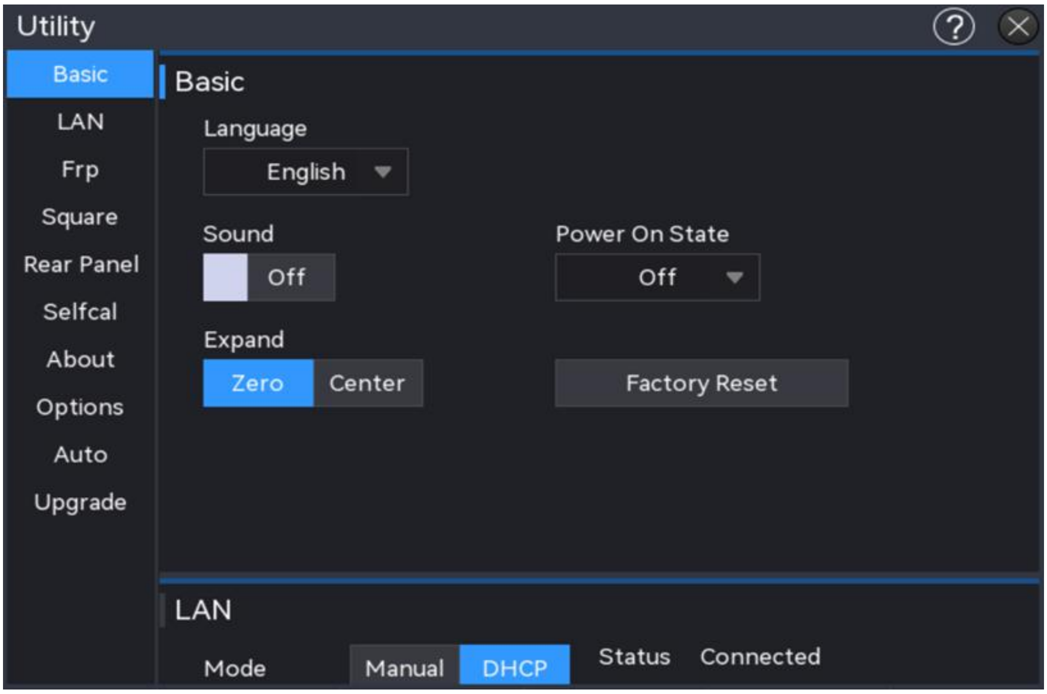
- Source: Click Source to select the source to set the color, the source can select M1, M2, M3, M4, R1, R2, R3, or R4.
- Color: Tap the color plate and drag to select the color.

# 17. Utility Function

Set the system function in the Utility menu.

Access the utility menu using the following method.

- Click Home icon in the top right-corner and select the utility icon to open the utility setting menu.
- If the utility function is added into the toolbar, click utility icon in the toolbar in the top right-corner to enter the utility setting menu.



## 17.1. Basic Information

The basic information includes language, restore setting, sound, power-on state, and vertical extension.

(1) Language

Click Language to set the system language to English, simplified Chinese, traditional Chinese, Italian, Spanish, French, German, or Polish.

(2) Sound

Click Sound to turn the beep on or off. When the sound is on, a buzzer sounds during the following operations:

- Click screen
- Remote control

(3) Power-on State

Set the oscilloscope power-on mode: Always Off, Always On, or Last Status.

- Always off: When the rear-panel power switch is turned on, the oscilloscope does not start automatically. It can only be powered on by pressing the Power softkey on the front panel.
- Always on: When the rear-panel power switch is turned on, the oscilloscope does not start automatically. It can only be powered on by pressing the Power softkey on the front panel.
- Last status: When the rear-panel power switch is turned on, the oscilloscope starts according to its last shutdown state. If the last shutdown was performed using the Power softkey, the oscilloscope must be powered on again using the Power softkey. If the last shutdown was due to direct power removal, the oscilloscope powers on automatically.

(4) Vertical Extension

Click vertical extension for the waveform.

- Screen center: When changing the vertical scale, the waveform will be extended or compressed around the screen center.
- Channel's zero position: When changing the vertical scale, the waveform will be extended or compressed around the channel's zero position.

## 17.2. Network Setting

When the device is connected to an available network, IP setting is used to set the IP, subnet mask, gateway, and DNS address of the oscilloscope.

(1) IP Mode

Set the IP acquire mode. It can be set to manual or auto (DHCP).

- Manual: Manually set IP address, subnet mask, gateway address, and DNS address.
- Auto (DHCP): Only for checking IP address, subnet mask, gateway address, and DNS address.

  a. IP Address: IP address format is *nnn.nnn.nnn.nnn*. The first nnn range is from 1 to 233, and the second nnn range is from 0 to 255. It is recommended that users can consult network administrators for an available IP address.

  b. Subnet mask: The format is *nnn.nnn.nnn.nnn*. The nnn range is from 0 to 255. It is recommended that users can consult network administrators for an available subnet mask.

  c. Gateway address: The format is *nnn.nnn.nnn.nnn*. The first nnn range is from 1 to 255, and the second nnn range is from 0 to 255. It is recommended that users can consult network administrators for an available gateway address.

  d. DNS address: The format is *nnn.nnn.nnn.nnn*. The first nnn range is from 1 to 255, and the second nnn range is from 0 to 255. It is recommended that users can consult network administrator for an available DNS address.

(2) Apply

After manually editing the IP address, subnet mask, gateway address and DNS address, click Apply to complete the configuration.

(3) LAN Reset

To clear the IP address, subnet mask, gateway address and DNS address, click LAN Reset to clear the edited IP address, subnet mask, gateway address, and DNS address.

## 17.3. Frp Setting

To configure external network access, click Apply after entering the settings. The user can then continue to access the network using the new FRP proxy IP address information that you modified (assuming the configuration is correct).

(1) Frp Setting

Frp mode: On or Off

When set to On, the oscilloscope can be accessed via an FRP connection.

(2) IP Setting

a. Frp_ip address: IP address format is *nnn.nnn.nnn.nnn*. The first nnn range is from 1 to 233, and the other three nnn range is from 0 to 255. It is recommended that users can consult network administrators for an available IP address.

b. Web port range: 1000 – 65535, default: 9005.

c. Control port range: 1000 – 65535, default: 9006.

d. Picture port range: 1000 – 65535, default: 9007.

(3) Apply

After manually editing the IP address, port, control port, and picture port information, click Apply to complete the settings.

**Note**: This oscilloscope uses frp intranet penetration to enable external network access. It oscilloscope operates as an FRP client based on FRP version 0.34.0. The client must connect to an FRP server with the FRP server service running. The client communicates with the FRP server through port 7000, so the server must be configured with bind_port = 7000. When multiple oscilloscopes are connected to the same FRP server, the web port, image port, and control port assigned to each oscilloscope must be unique. Otherwise, the FRP proxy will fail, and remote access will not be available. If the FRP proxy settings are modified, the oscilloscope cannot be accessed through the local network at IP:9000. To restore normal LAN access, press Default to reset the configuration. After the reset, access via port 9000 will be restored.

## 17.4. Square Selection

This oscilloscope provides square-wave output ports. Port 1 corresponds to the Probe Comp 1 terminal on the front panel.

These ports support selectable output frequencies, as listed in the table below. The default frequency is 1 kHz.

| Port | Frequency |
|---|---|
| Port 1 | 1 kHz, 3V_REF |

## 17.5. Rear Panel

In the auxiliary menu, click Rear panel or drag the content to enter the rear panel setting

(1) 10MHz Sync

- Idle: The [10 MHz REF In/Out] connector on the rear panel is not used as a reference clock input or output and is for display only.
- Input: The [10 MHz REF In/Out] connector on the rear panel is used as a reference clock input.

(2) AUX Output

AUX output is used to select the output signal for the [AUX Out] connector on the rear panel.

- Output: Each time the oscilloscope generates a trigger, a signal reflecting the current capture signal of the oscilloscope is output from the [AUX Out] connector on the rear panel. When this signal is connected to the waveform display and device and measure the frequency of this signal, the measured result is the same as the current capture signal.

- Pass/Fail: In a pass/fail test, a positive or negative pulse is output from the rear panel [AUX Out] connector when the oscilloscope detects a pass or fail event. \
- DVM: During DVM measurement, a positive or negative pulse is output from the [AUX Out] connector on the rear panel when a DVM alarm condition is detected.

## 17.6. Auto-Calibration

Auto Calibration optimizes the oscilloscope for peak performance and ensures accurate measurement results.

This function calibrates the analog channels and can be performed at any time, particularly when the ambient temperature changes by 5 °C or more. Ensure that the oscilloscope has been powered on and operating for at least 20 minutes before performing the auto calibration.

(1) Analog channel calibration: This calibration applies only to the analog channels and takes approximately 3–5 minutes to complete.

## 17.7. About Oscilloscope

Click About in the auxiliary menu to check the oscilloscope information.

- Model: Product model
- Serial number: Product serial number; the unique identifier of the product
- IPK Version: Product IPK version number
- Software Version: Product software version number
- Power Board Version: Product power board version number
- Logic Version: Product logic version number
- File System Version: Product file system version number
- Hardware Version: Product hardware version number
- Factory Reset: Click Reset to restore the oscilloscope to its factory default settings. This will delete all locally stored files.

## 17.8. Option

Click Option in the auxiliary menu to check all options.

All options support a 540-hour trial period. After the trial period expires, users must purchase the corresponding option and obtain an option license to continue using it.

The procedure for obtaining a software license for the option is as follows:

1. Purchase the option key from a distributor.
2. Go to the UNI-T official website and navigate to Test Instrument > Service and Support > License Activation to access the software license activation page.
3. On the software license activation page, enter the purchased key, device serial number, and verification code as prompted. The software license will be generated immediately.

The steps for installing the software license for the option are as follows:

1. Download the software license from the License Information list, save it to the USB root directory, and insert the USB drive into the device.
2. Go to Utility > Options, and click Activate or Activate All to activate the option using the software license.
3. To verify that the license has been installed, open Utility > Options. In the option trial table, the Activation Status field will display Activated, and the Trial Duration field will display Permanent.

**Note**: If you have registered and logged in to the UNI-T official website and provided your email address during registration, the generated software license will be sent to your email as an attachment. Th software license can also be viewed and downloaded again from the license activation page after logging in to the UNI-T official website.

UPO6000LP series supports the following optional functions, as shown in the table below:

| Model | Option |
|---|---|
| UPO6000LP Series | CAN, LIN, CAN-FD, FlexRay, SNET, Audio, 1553B, Manchester, ARINC429, PWR, Gen (Generator + Bode Plot) |

## 17.9. Auto Setting

Click Auto setting in the auxiliary menu to enter the auto setting menu.

(1) Channel Setting

- In hold mode, the bandwidth limit, inverse phase, impedance, unit, probe attenuation ratio, label state remains the same, and coupling (ground), Vertical scale, offset and fine-tuning are reset to the default.
- In auto mode, the impedance, unit, probe attenuation ratio, and label state remain the same; other settings are reset to the default.

(2) Acquisition Setting

- In current mode, all settings remain the same.
- In auto mode, the acquisition mode reset to the default; other settings remain the same.

(3) Trigger Source

- In current mode, the source, trigger coupling remains the same; other settings are reset to edge trigger, auto, rising edge.
- In auto mode, all parameters are reset to the default.

(4) Activate Channel

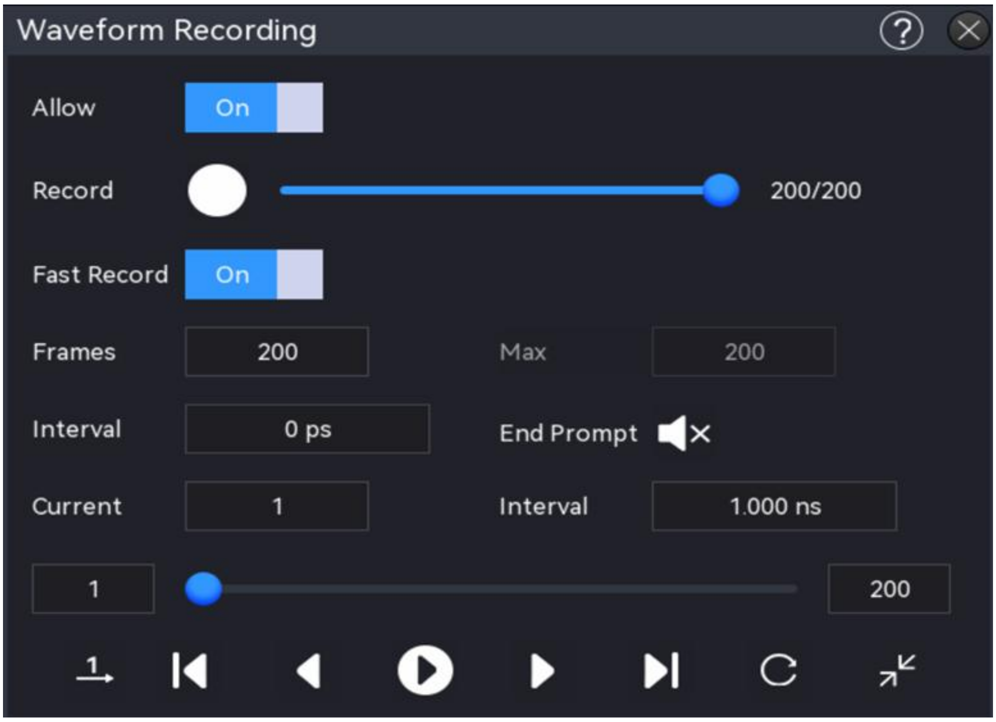
- In current mode, the switch state remains the same when the channel is automatically set.
- In auto mode, only channels with signal input will be activated.

# 18. Waveform Recording

The Waveform Record and Playback function allows recorded waveforms, to be played back for convenient analysis.

The Waveform Recording menu can be accessed using the following methods.

- Click Home icon in the top right-corner and select the waveform recording icon to open the waveform recording menu.
- If the waveform recording function is added into the toolbar, click waveform recording icon in the toolbar in the top right-corner to enter the waveform recording menu.




(1) Turn on/off Record

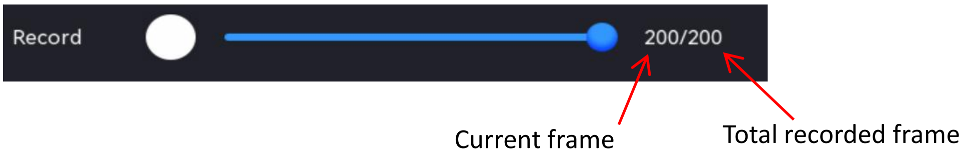
Click Record switch to turn the waveform recording on or off. Before using this function, refer to the Recording Setting section.

(2) Recording

Click Record to start recording.

- Click Record key to start recording. The recording icon will change to , indicating that recording is in progress.
- The data displayed on the right side of the recording progress bar represents the current number of frames out of the total number of recorded frames. During the recording process, the current recording information is shown on the screen in real time, with the current number of frames continuously updating.

- After recording, the recording icon will change to , indicating that recording has stopped.
- During recording, click icon can pause the recording,




(3) Playback

Click Playback icon to play back the recorded waveform. When playback starts, the icon changes to , indicating that the waveform is being played back. For more details about playback, refer to the Playback Setting section.

During playback, Current Frame will change in real time. The user can click pause key to stop the playback.

## 18.1. Recording Setting

During waveform recording, the oscilloscope records the waveforms of all currently opened channels at specified intervals until the user manually stops the recording operation or the number of recorded frames reaches the set limit.

Before waveform recording, the following recording option parameters can be configured.

(1) Sequential Acquisition

Sequential acquisition, which allows for continuous and uninterrupted recording, improves the waveform capture rate. During sequential acquisition, the waveform is not displayed on the screen and can only be played back after the recording is completed. It can be set to on or off.

(2) Recording Frame

The number of recording frames refers to the total number of frames that can be recorded. After starting the recording operation, the oscilloscope will automatically stop recording when the number of recorded frames reaches the specified limit. Double-click Frames input field to open the numeric keypad and set the recording frame. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to the maximum frame.

(3) Maximum Frame

Maximum frames input window displays the highest number of frames that can be recorded. Since the waveform memory size is fixed, a higher number of points per waveform frame results in fewer frames being recorded. Therefore, the maximum number of frames is directly related to the currently selected Memory Depth (see Memory Depth). The number of points per frame equals the current memory depth, which is calculated as ≥ sampling rate × horizontal time base × the number of frames displayed horizontally on the screen. Consequently, the maximum value for waveform recording is also influenced by the "Sample Rate" and "Horizontal Time Base".

(4) Recording Interval

The recording interval is the time interval between frames during the recording process. Double-click Interval input field to open the numeric keypad and set the recording interval. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 s to 10 s

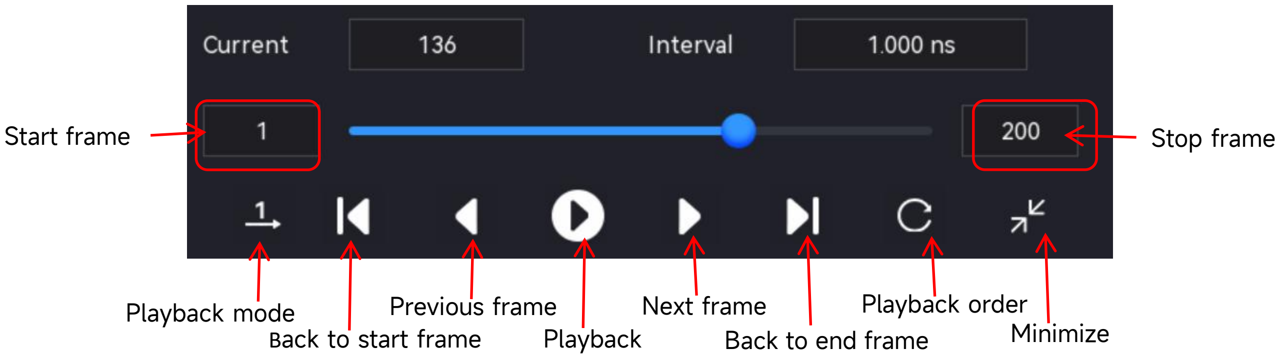
(5) End Sound

: The beeper emits a sound when recording ends.

: The beeper remains silent when recording ends.

## 18.2. Playback Setting

Before waveform recording, the following recording option parameters can be configured.




(1) Playback Mode

The playback mode is divided into two modes: single playback and cycle playback . Click icon in the bottom left corner of the screen to switch between modes.

: Playback starts from the start frame and ends at the end frame, stopping automatically.

: Playback starts from the start frame and ends at the end frame, repeating until manually stopped.

(2) Playback Order

The playback order consists of two types: sequential playback and reverse playback . Click icon in the bottom right corner of the screen to switch between them.

: Playback starts from the start frame to the end frame.

: Playback starts from the end frame to the start frame.

(3) Minimize

Minimize the playback window as needed. Click minimize icon  to reduce the size of the playback window, simplifying the interface for a more intuitive and user-friendly experience; Click restore icon  to revert the window to its original size and exit the minimized view.

(4) Current Frame

When the playback is stopped, double Click Current input field to open the numeric keypad and set the current frame. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The maximum value that can be set is equal to the number of recorded frames.

(5) Playback Interval

The playback interval refers to the time interval between frames during playback. Double-click Playback Interval input field to open the numeric keypad and set the playback interval. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 0 s to 5 s.

(6) Start Frame

The start frame is the frame from which playback begins. Double-click Start Frame input field to open the numeric keypad and set the start frame. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The default is 1. The maximum value that can be set is equal to the number of recorded frames.

(7) Stop Frame

The end frame is the frame at which playback stops automatically. Double-click Stop Frame input field to open the numeric keypad and set the stop frame. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The default is the total number of frames in the recorded waveform.

# 19. Pass/Fail Test

In product design and production, it is often necessary to monitor signal changes and determine product qualification. The standard Pass/Fail test function of this oscilloscope series effectively meets this need. Test rules can be established based on known "standard" waveforms, allowing the signal under test to be compared against these standards and statistical information about the results to be displayed.

When the oscilloscope detects a pass or fail, the system can be configured to stop monitoring immediately, activate a buzzer alarm, and save the current screenshot.

Access the Pass/Fail Test menu using the following method.

- Click Home icon in the top right-corner and select the pass/fail test icon to open the pass/fail test menu.
- If the pass/fail test function is added into the toolbar, click pass/fail test icon in the toolbar in the top right-corner to enter the pass/fail test menu.
- If pass/fail test results appear, click icon to open the pass/fail test menu.


## 19.1. Pass/Fail Test

Click Pass/Fail to turn the pass/fail test ON or OFF.

## 19.2. Source

Click Source to select C1-C4 as the trigger source.

## 19.3. Mask Setting

(1) Mask Source

Click Mask Source to select the source for creating a mask. The source can choose from C1-C4 and Ref. When Ref is selected, a reference waveform is required to generate the mask.

(2) Vertical Setting

Double-click Vertical input field to set the vertical tolerance. The setting range is from 1 to 100.

(3) Horizontal Setting

Double-click Horizontal input field to set the horizontal tolerance. The setting range is from 1 to 50.

(4) Test Rule

After setting the Mask Source, Vertical, and Horizontal, click Create to define the test rule. The blue area on the screen indicates regions not covered by the mask.

(5) Load Test Rule

When the pass/fail test function is enabled, the user can load and recall test rule files stored in the instrument's internal memory or an external USB flash drive (if the USB flash drive is detected) and apply them to the current pass/fail test. Click Load to access the file loading interface. Select the specified test rule file (*.tmp) from the File Path and apply it to the current pass/fail test.

(6) Save Test Rule

When the pass/fail test function is enabled, the current test rule range can be saved to the instrument's internal memory or an external USB flash drive in *.tmp format (if the USB flash drive is detected). Click Save to open the file-saving interface. Enter the required information in the File Name and Save Path fields to save the test rule file to either internal or external memory. For details on the saving process, refer to the Storage and Load section.

## 19.4. Output Setting

(1) Aux Output

Click Aux Output to turn the Aux Output function ON or OFF.

- ON: When the Aux Output function is enabled, the AUX Output menu automatically switches to Pass/Fail, indicating that during a pass or failure event, the Aux Output connector on the rear panel will output a pulse.
- OFF: When the Aux Output function is disabled, the AUX Output menu automatically switches to Trigger Output, indicating that the Aux Output connector on the rear panel is

not linked to the pass/fail test.

(2) Output Condition

Configure the signal output from the Aux Output connector. The output can be set to two signal conditions: Pass or Fail.

- Fail: When the AUX port on the rear panel detects a Fail signal, a pulse is output and the beeper sounds.
- Pass: When the AUX port on the rear panel detects a Pass signal, a pulse is output and the beeper sounds.

(3) Output Pulse Width

Set the pulse width for the pass/fail test. Double-click Output Pulse Width input field to open the numeric keypad and set the output pulse width. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 200 ns to 1.5 s.

(4) Output Polarity

Click Output Polarity to set the polarity of the output pulse to Positive or Negative.

## 19.5. Stop Setting

(1) Stop Type

Stop type refers to the condition under which the pass/fail test stops when a specific signal. It can be set to Fail or Pass.

- Fail: When a Fail signal is detected, the failed frame count increases by 1. The test automatically stops once the number of failures reaches the predefined threshold.
- Pass: When a Pass signal is detected, the success count increases by 1. The test stops once the number of successes reaches the specified threshold.

(2) Stop Condition

Click Stop Condition to configure the condition for ceasing the test. This can be set to ⩾ or ⩽.

- ⩾: The test will automatically stop when the number of frames of the stop type is greater than or equal to the specified condition.
- ⩽: The test will automatically stop when the number of frames of the stop type is less than or equal to the specified condition.

(3) Condition Time

Double-click Condition Time input field to open the numeric keypad and set the condition time. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 60000.

(4) Screenshot

Click Screenshot to turn the screenshot function ON or OFF.

- ON: Automatically captures and saves a screenshot to local or external USB storage when the pass test is stopped upon meeting the test conditions.
- OFF: No screenshot will be captured when the test is stopped.

## 19.6. Operation and Reset

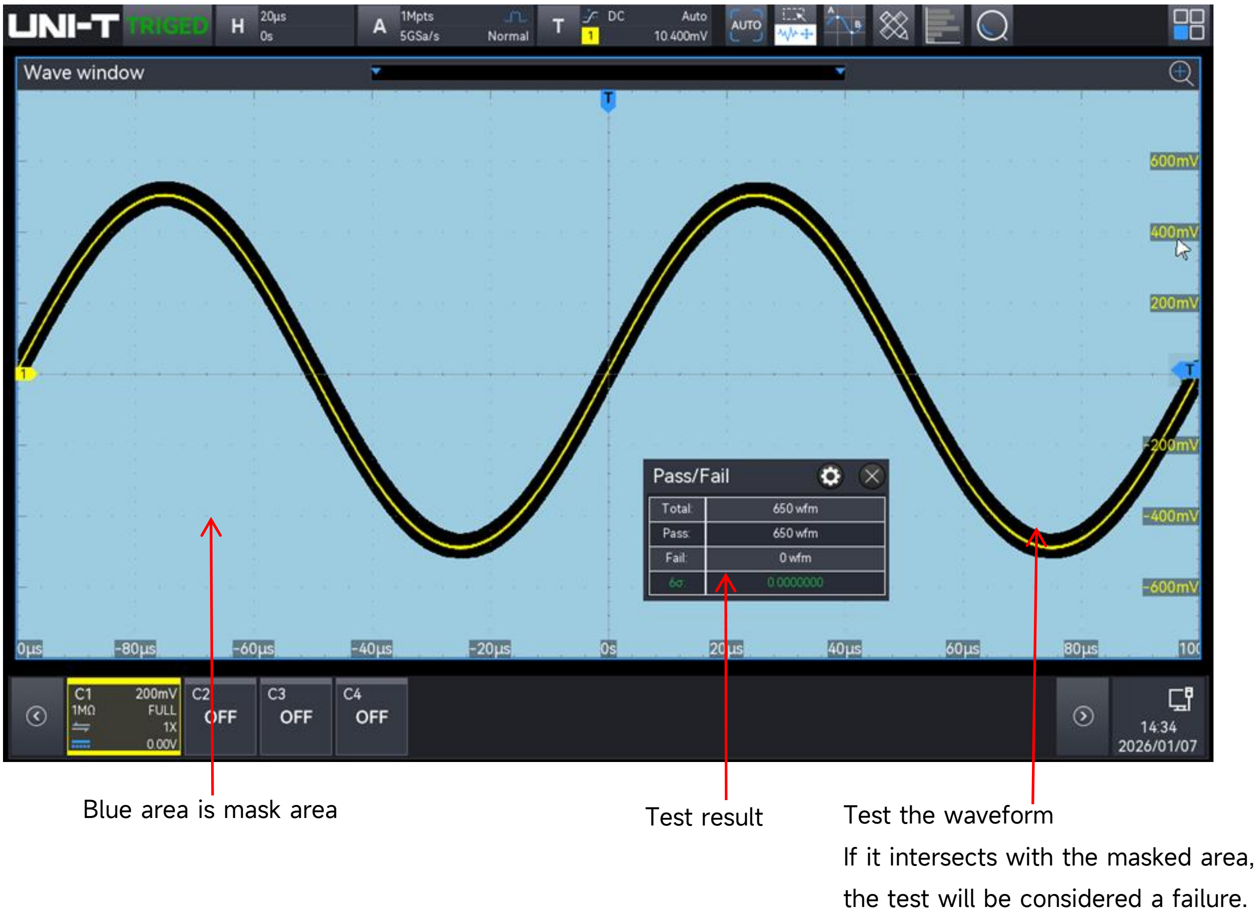
(1) Operation

After configuring all the options mentioned above, click Operation to configure the pass/fail test.

- Start: Initiates the pass/fail test.
- Stop: Terminates the pass/fail test.

(2) Reset

During the pass/fail test, click Reset can clear the test results and restart the pass/fail test.

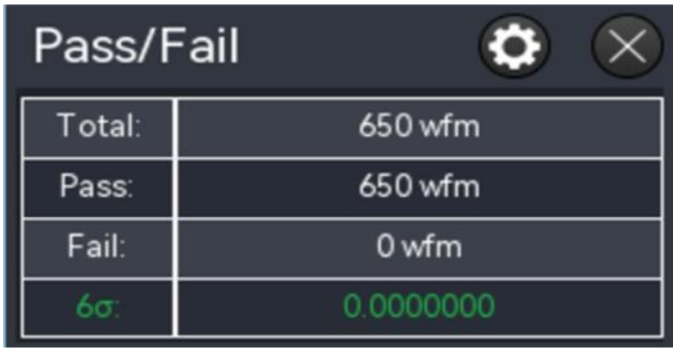



**Note**: Pass/fail tests can only be started or stopped when the pass/fail test function is enabled, and the test rule ranges are saved and loaded. Source channel changes and test rule adjustments are not permitted while the test is in progress.

## 19.7. Pass/Fail Test Result

When the Pass/Fail function is enabled, the test results window is displayed on the screen.

Test results window: Displays fail frames, pass frames, total frames, and 6σ, as shown in the following figure.




- Total: Total tested frames
- Pass: Pass frames
- Fail: Fail frames
- 6σ: The ratio of fail frames to total tested frames

The 6σ criterion indicates that, out of one million operations, only 3.4 failures are expected.

# 20. System Upgrade

## 20.1 Access Version

1. Navigate to the Auxiliary menu via Home > Auxiliary, and click About to view the system information: instrument model, software version, and hardware version.
2. Obtain the upgrade file by downloading it from the official website or through an authorized distributor.
   Ensure that the upgrade file matches the instrument model and hardware version, and that its software version is higher than the version currently installed.
   Save the upgrade file to the root directory of a USB flash drive.

## 20.2 Firmware Upgrade

This series supports both USB flash drive upgrades and online upgrades, providing a convenient and flexible way to update the firmware.

USB Flash Drive Upgrade

There are three methods to upgrade via a USB flash drive.

1. Power-on USB upgrade detection.
2. Navigate to Home > Auxiliary > Upgrade, detect the upgrade file on the USB flash drive.
3. Online upgrade.

(1) Power-on Upgrade

- Power off the oscilloscope and insert the USB flash drive containing the upgrade file. Press the power button. The oscilloscope automatically detects the upgrade file and enters the upgrade interface.
- Select the upgrade version and press Upgrades to start the process.
- The upgrade process takes approximately 5 minutes. After completion, the oscilloscope restarts automatically. Remove the USB flash drive during the restart process.
- Power on the oscilloscope and check the system information. If the displayed software version matches the upgrade version, the upgrade was successful.

(2) Local Upgrade

- Copy the upgrade file to the USB flash drive and insert it into the oscilloscope. The upgrade function becomes available only after the USB flash drive is detected.
- Click the Upgrade File drop-down menu. All available upgrade files in the root directory

of the USB flash drive are displayed. Select the required file directly. Alternatively, click Browse to open the file browser, select the target file, and click OK.

- Press Upgrade. A confirmation window appears with OK and Cancel options:
  OK: Start the upgrade using the selected file.
  Cancel: Abort the upgrade. The operation can also be canceled by clicking the close icon (×) in the upper-right corner of the window.
- The upgrade process takes approximately 5 minutes. After completion, the oscilloscope restarts automatically. Remove the USB flash drive during the restart process.
- Power on the oscilloscope and check the system information. If the displayed software version matches the upgrade version, the upgrade was successful.

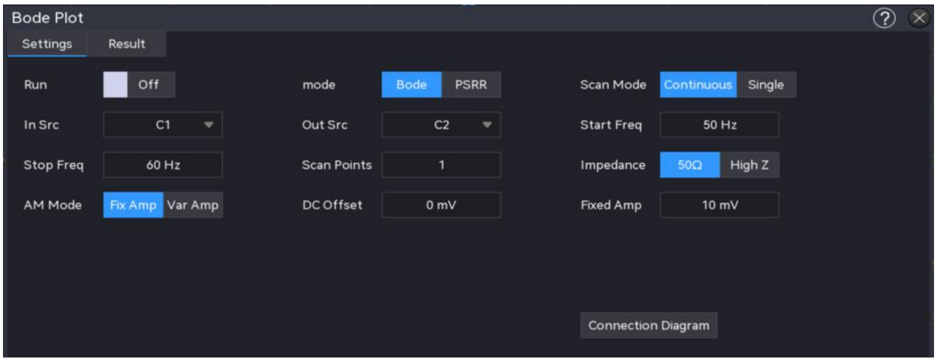
(3) Online Upgrade

- Ensure the oscilloscope's LAN interface is connected to a network. Apply for network permissions if necessary.
- Press Online Upgrade. A confirmation window will appear with OK and Cancel options:
  OK: Start the upgrade using the selected online file.
  Cancel: Abort the upgrade. The operation can also be canceled by clicking the close icon (×) in the upper-right corner of the window.

**Note**: Do not power off the oscilloscope during the upgrade process. Interrupting power may result in an incomplete firmware update and cause the instrument to fail to restart.

# 21. Bode Plot

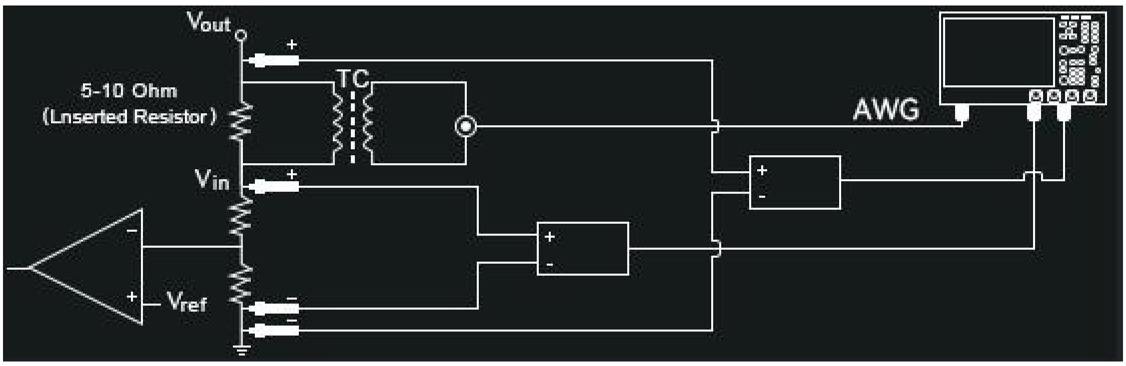
The Bode plot function provides a frequency response curve for the Device Under Test (DUT). During the sweep, the oscilloscope configures the function/arbitrary waveform generator to output a signal to the DUT. It then compares the DUT input and output signals, measuring the gain (G) and phase (P) at each frequency, which are plotted on the frequency response Bode plot. When the loop response analysis is complete, the user can move the cursor on the chart to view the measured gain and phase values at various frequency points. The scale and offset settings of both the amplitude and phase plots can also be adjusted.




## 21.1. Sweep Configuration

(1) Wiring Diagram

Before using the Bode plot (Power Supply Rejection Ratio) function, set up the loop connections as shown in the diagram below. Click the wiring instructions to view the circuit connection diagram for the Bode plot (Power Supply Rejection Ratio) function in the pop-uo window.




(2) Operation

Click Operation to set the Bode plot to on or off.

(3) Mode

Click Mode to set the operating mode to Bode plot or power supply rejection ratio (PSRR).

- Bode Plot: Conducts a Bode plot sweep, displaying both gain and phase curves simultaneously.

- Power Supply Rejection Ratio (PSRR): Performs a PSRR sweep and displays only the gain curve. The PSRR test determines how effectively a voltage regulator suppresses ripple noise across different frequency ranges. This analysis uses a waveform generator from the oscilloscope to provide a signal with varying frequencies, introducing ripple into the DC voltage supplied to the regulator. The AC RMS ratio of input to output is measured and plotted against the frequency range. Multiple methods can be used to measure PSRR. Due to higher background noise and lower sensitivity compared to a network analyzer, the oscilloscope PSRR measurement typically does not exceed -60 dB. However, PSRR testing with an oscilloscope is generally acceptable for a quick assessment of the overall PSRR performance of the power supply under test.

(4) Sweep Mode

Click Sweep Mode menu to select the sweep mode: single sweep and continuous sweep.

- Single sweep: The sweep proceeds from the start frequency to the end frequency and then stops automatically.
- Continuous sweep: The sweep continuously cycles from the start frequency to the end frequency without automatically stopping.

(5) Sweep Settings

Set the sweep setting: Start frequency, stop frequency, sweep points, amplitude mode, amplitude setting, DC offset, and source impedance.

a. Start frequency: Set the start frequency for the sweep. Range: 50 Hz - 50 MHz.

b. Stop frequency: Set the stop frequency for the sweep. Range: 60 Hz - 50 MHz.

c. Sweep points: Set the number of sweep points; a higher number provides better sweep resolution. Range: 1 - 1000.

d. Amplitude mode: Set the amplitude of the sweep signal, allowing for either fixed amplitude or variable amplitude.

  - Fixed amplitude: When the amplitude mode is set to fixed amplitude, the signal amplitude remains at a constant value. The adjustable range is 10 mV to 3 V (for a 50 Ω impedance) and 20 mV - 6 V (for a high-impedance setting).
  - Variable amplitude: When the amplitude mode is set to variable amplitude, the input signal can be set to different amplitudes at various frequency stages. The adjustable range is 10 mV - 3 V (for a 50 Ω impedance) and 20 mV - 6 V (for a high-impedance setting).

e. DC Offset: Set the offset of the sweeping signal. The adjustable range is -1 V to 1 V.

f. Source impedance: Set the source load to 50 Ω or high impedance.

(6) Channel Settings

Set the input and output signal channels for the device under test.

a. Input Source

Click Input Source drop-down menu to select the input signal channel for the Bode plot (PSRR). C1-C4 can be set.

b. Output Source

Click Output Source drop-down menu to select the output signal channel for the Bode plot (PSRR). C1-C4 can be set.

**Note**: The input source and output source cannot be the same channel.

## 21.2. Display Setting

(1) Bode Plot Display Settings

Set the display parameters: Automatic settings, gain range, gain offset, phase range, phase offset, start frequency, stop frequency, gain display, and phase display.

a. Automatic settings: Based on the amplitude and phase curves of the output signal, the oscilloscope automatically configures parameters including gain range, gain offset, phase range, phase offset, start frequency, and stop frequency.
b. Gain display: Sets whether gain data and related information are displayed in the waveform window. The gain display can be turned on or off.
c. Phase display: Sets whether phase data and related information are displayed in the waveform window. The phase display can be turned on or off.
d. Gain range: Sets the gain range displayed in the waveform window. Range: 1 dB - 500 dB.
e. Gain offset: Sets the gain offset displayed in the waveform window. Range: -250 dB to 250 dB.
f. Phase range: Sets the phase range displayed in the waveform window. Range: 1° - 180°.
g. Phase offset: Sets the phase offset displayed in the waveform window. Range: -180° to 180°.
h. Start frequency: Sets the start frequency displayed on the horizontal axis of the waveform window. Range: 50 Hz - 50 MHz.
i. Stop frequency: Sets the stop frequency displayed on the horizontal axis of the waveform window. Range: 60 Hz - 50 MHz.

# 21.3. Result Analysis and Export

(1) Result Analysis

The sweep results are shown in the figure below. Using the data list and cursor measurement functions, the user can conduct a detailed analysis of the Bode plot curves. The data list provides measurement information for each sweep point, and the cursor line allows for flexible measurement of changes at various positions on the curve.


1. Gain scale: Displays the gain based on the configured gain range and offset.
2. GM (Gain Margin): The difference between the gain measurement at the frequency point where the gain is 0 dB and the phase is 0°, calculated as GM = 0 dB - Gain Measurement.
3. PM (Phase Margin): The difference between the phase measurement at the frequency point corresponding to 0 dB gain and 0°.
4. Cursor: A movable cursor that measures the gain, phase, and frequency values at the intersection points of the gain result curve and phase result curve.
5. Two Cursor Lines: Displays the gain values, phase values, and frequency values at the intersection points with the gain and phase curves.
6. Swept phase result curve
7. Swept gain result curve
8. Frequency scale: Displays the configured start frequency and stop frequency for the sweep.
9. Phase scale: Displays the phase based on the configured phase range and offset.
10. Intersection of the phase margin curve with the zero axis.
11. Intersection of the gain margin curve with the zero axis.

(2) Table

The table displays detailed information for each sweep point. Check the Table to show the sweep point data in tabular form. The displayed content includes the sweep point index, frequency, amplitude, gain, and phase.


# 21.4. Result Analysis and Load

(1) Save Results Table

After opening the results table, click Save Result to open the export setting menu, the data can be saved as *.csv file to the internal storage or an external USB flash drive (when a USB is detected). For the setting steps, refer to the section of Save and Load.

(2) Load Results Table

When Bode plot data is saved in internal storage or on an external USB drive (only when the USB flash drive is detected), the results table can be loaded into the oscilloscope and display the corresponding Bode plot or PSRR graph. Select the Table, and click Load to open the load menu. Double-click File input window to open the file browser interface. In the file browser interface, select the file to load the Bode plot or PSRR data.

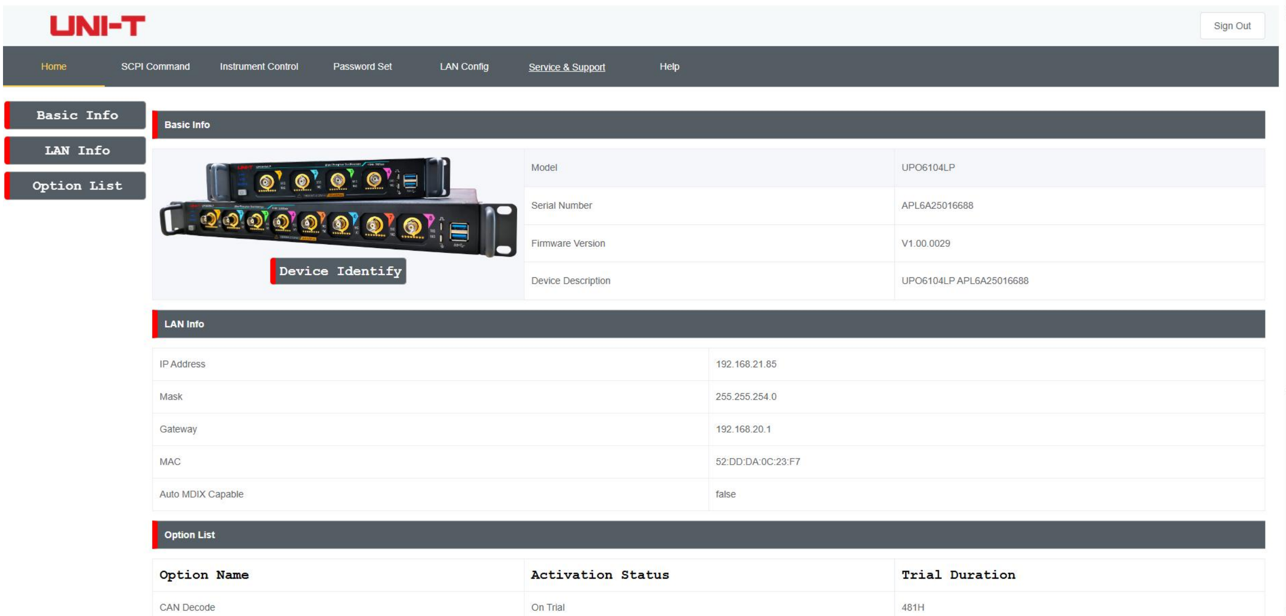
# 22. Web Access

## 22.1. Access Local Area Network

The computer and the oscilloscope must be connected to the same LAN. Check the oscilloscope's local IP address in the Utility menu, then enter http://<IP>:9000 in a web browser on the computer to access the oscilloscope.

Example:

- Computer IP: 192.168.42.3
- Oscilloscope IP: 192.168.42.12
- PC browser using 192.168.42.12 to access the oscilloscope can check the basic information, as shown in the following figure.




When the SCPI Command, Instrument Control, Network Settings, or Password Settings tab is selected, login is required. Refer to the *Login Web* section for detailed instructions.

## 22.2. Login Web

Many operations require logging into the web system. The login credentials are as follows:

- **Username:** admin
- **Password:** The initial password is the oscilloscope's serial number encoded using Base64.

For security reasons, users are strongly recommended to change the default password after the first login. After a new password is set, it will be required for all subsequent logins.

## 22.3. Access Outer Network

To access the oscilloscope from an external network, when access terminal and the oscilloscope are not on the same LAN segment, follow the steps below:

a. Ensure the network cable is plugged into the oscilloscope and that internet access is available.

b. Turn on the frp proxy service on the server.

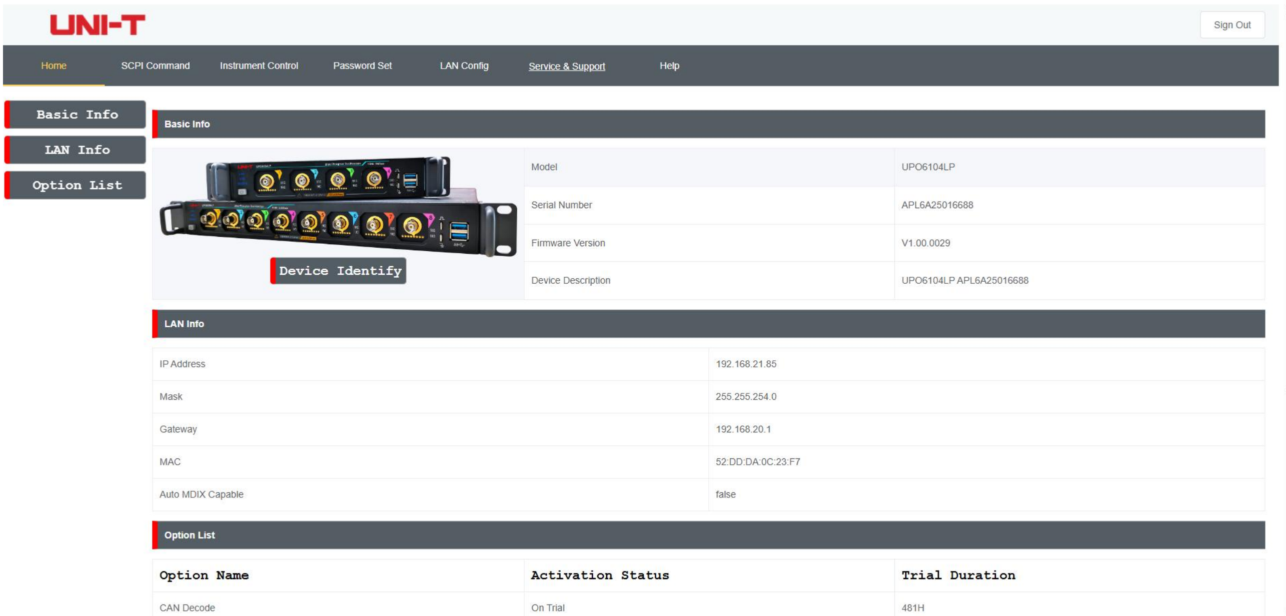
c. Configure the frp proxy IP and port, image port, and control port of the oscilloscope.

d. Access the oscilloscope through a web browser using the FRP proxy IP address.

**Note**: This instrument uses a way of frp (Fast Reverse Proxy) intranet penetration to access the outer network. The frp version is 0.34.0. The instrument carries an FRP-0.34.0 client port, which requires an FRP server to be running with the FRP service enabled. The client connects to the FRP server port 7000, so the server configuration must include “bind_port = 7000”.

## 22.4. Home Page

The home page of the web system displays the basic information of the currently connected instruments, along with network information and an option list.



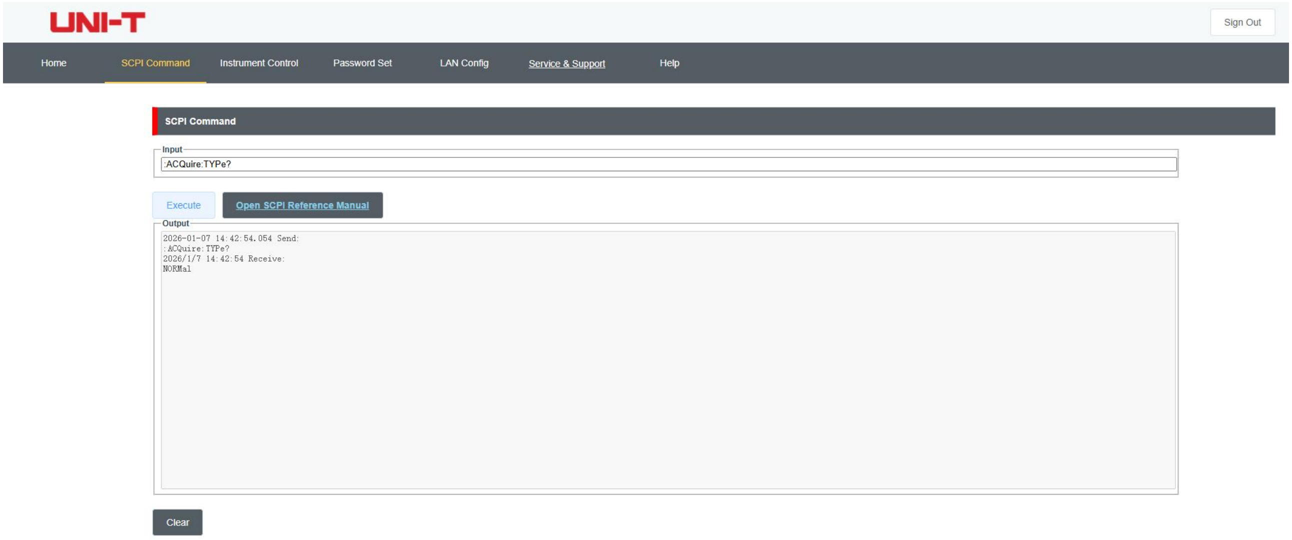

## 22.5. SCPI

On the Web System SCPI Command page, the user can send SCPI commands to the currently connected oscilloscope.

- Input command: Click Input Command to open the drop-down list of SCPI commands. This list displays all the SCPI commands supported by the oscilloscope. Select the desired command

and configure the relevant parameters (such as channel settings) as needed. Click Perform to send the SCPI command.

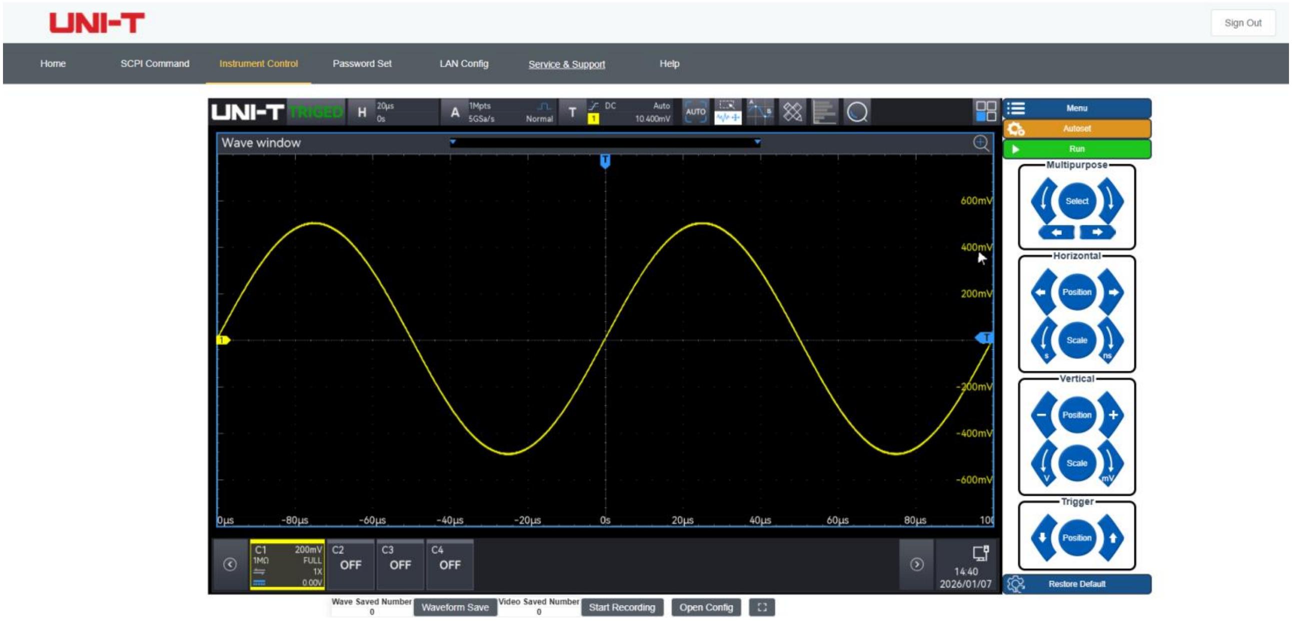
- SCPI output: Displays the execution results of the SCPI commands. When a command is performed, the window shows both the transmitted command and the corresponding response returned by the oscilloscope.
- Open SCPI programming manual: Click SCPI to access the SCPI programming manual for this oscilloscope.
- Clear: Clears the contents of the SCPI output window.




## 22.6. Instrument Control

On the Web System Instrument Control page, users can operate oscilloscope functions using a mouse. All operations are performed directly within the web, interface, including the following:

- Waveform manipulation: Click and drag the waveform to move it vertically or horizontally. The user can also adjust the trigger level position and reposition pop-up windows by dragging them with the mouse.
- Webpage: On the web page, the mouse wheel can be used to operate virtual knobs, or users can click the knobs to adjust waveform horizontal and vertical positions, modify the time base scale, change the volts-per-division setting, perform coarse or fine adjustments, and set the trigger level.
- Menu interactions: Use mouse clicks to open or close drop-down menus and pop-up dialogs, and to switch between different menu options.
- Input keyboard: Double-click screen to open the numeric keypad, alphabetical keypad, or other input options.




(1) Key Area

- Menu: Click the Menu key to open the auxiliary menu window box.
- Auto: Click the Auto key to perform the Autoset operation.
- Run/Stop: Click the Run/Stop key to change the oscilloscope's operating state.
- Reset: Click Restore Default to restore the settings to factory defaults.

(2) Rotary Knob

- Multi-function Knob: Supports multi-functional operations through a window box. Click , on either side of the knob to modify the numeric values. Click , at the bottom switches the numeric digits. When certain input fields have a default reset option available, clicking the knob can restore them to their default settings, such as the decoder line position.
- Horizontal-Position: Click , on either side of the knob or scrolling the rotary knob moves the waveform’s horizontal position. Click knob to set the horizontal position to center.
- Horizontal-Scale: Click , on either side of the knob or scrolling the rotary knob moves the waveform’s time base. Click knob to set the fine tuning or coarse tuning.
- Vertical-Position: Click , on either side of the knob or scrolling the rotary knob moves the waveform’s vertical position. Click knob to set the fine tuning or coarse tuning.
- Vertical-Scale: Click , on either side of the knob or scrolling the rotary knob moves the waveform’s volts/div scale. Click knob to set the fine tuning or coarse tuning.

- Trigger-Position: Click , on either side of the knob or scrolling the rotary knob moves the waveform's trigger position. Click knob to set the level to center.

(3) Configuration

Click Configuration to open the save configuration window, where the user can save waveform settings, set up video recording, and print screenshots.


① Save Waveform

Save the waveform data displayed on the screen in .dat or .csv format. After saving, the files can be downloaded to a local PC. The downloaded .dat and .csv files are compatible with the host computer's waveform analysis software.

a. Save mode: Set the save mode for waveform saving and video recording. The available modes are Screen Waveform and Deep Memory Waveform.

- Screen waveform: Saves or records only the waveforms displayed in the oscilloscope screen area. The saved file format is .dat.
- Deep memory waveform: Saves or records all data based on the oscilloscope's configured memory depth, including data outside the visible screen area. The saved file format is .csv.

b. Channel selection: Select the channel for waveform saving. The available options are C1, C2, C3, C4, and ALL (ALL is available only for screen waveform saving). If the channel is not enabled, its data cannot be saved.

c. Save method: Select the method to single save or continuous trigger for saving waveforms. Both methods can be selected.

- Single save: Saves only the .dat data from a single trigger of the selected channel.
- Continuous trigger: Builds upon the single save by performing a Single operation

and then saving the waveform data after detecting a STOP state. This process continues in a loop.

d. Recording operation: Click Waveform Save key to start saving the screen as .dat file. During the saving process, click Stop Save to cease the operation, which will automatically download the .dat file to local storage.

**Note**: When saving deep memory waveform data, if Stop Save is clicked before the operation is complete, the incomplete file will not be saved or downloaded.

e. Number of saved files: Displays the total number of waveform files that have been successfully saved and downloaded to local storage during the current operation.

② Video Recording

Click Screen Recording to record the current instrument interface on the web. During the recording, click Stop Recording will end the recording process. Once the recording stops, a video in .mp4 format will be generated, displaying the recording duration. After completion, the video will be automatically downloaded to local storage.

a. Save mode: Select the mode to normal recording or trigger recording for recording the video.

- Normal recording: This records the operations on the oscilloscope screen, directly generating a video in .mp4 format without changing the oscilloscope's state.
- Trigger recording: Records video data based on trigger events. In this mode, the system automatically sends the :KEY: Single command and checks whether the oscilloscope enters the STOP state. When the STOP state is detected, the currently triggered screen is captured as an image, downloaded to local storage, and added to the video data. This process repeats until the accumulated video data reaches or exceeds the specified segment size. The current video segment is then saved, and the system automatically continues recording the next segment.

b. Segment size: Choose the size of the recorded video. When the recorded data reaches the specified size, the current video is automatically saved and downloaded, and recording continues with a new video file. The available segment sizes are 5 MB, 20 MB, 100 MB, and 500 MB.

c. Recording operation: Click Start Recording to begin video recording according to the selected save mode. During recording, click Stop Recording to immediately stop the process. The video data currently stored in memory is saved as a video file and automatically downloaded to local storage.

d. Number of files saved: Displays the number of files that have been saved and downloaded to local storage during the current operation.

③ Screen Print

Click Screen Print key to save the oscilloscope screen image according to the configured settings. Click Download Image to download the image to local storage.

a. Image format: Select the format for the saved image. Available options: *.png, *.bmp, *.jpg.

b. Color mode: Select the color mode for saving. Available options: Color, Grayscale, Ink-Saving, Ink-Saving & Grayscale. For the setting steps, refer to the section of Save and Load.

c. Header: Enable or disable the image header.

ON: The instrument model and image creation date are displayed in the image header when the file is saved.

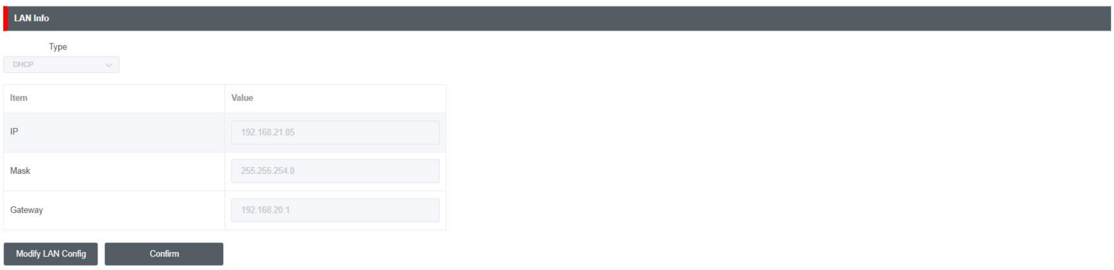
OFF: No information is displayed in the image header.

d. Inverse Color: Enable or disable color inversion. When enabled, the image colors are inverted according to the setting during saving.

## 22.7. Network Setting

Network settings allow users to configure the oscilloscope's LAN parameters and external network (FRP proxy) access.

a. Oscilloscope Network Information Settings

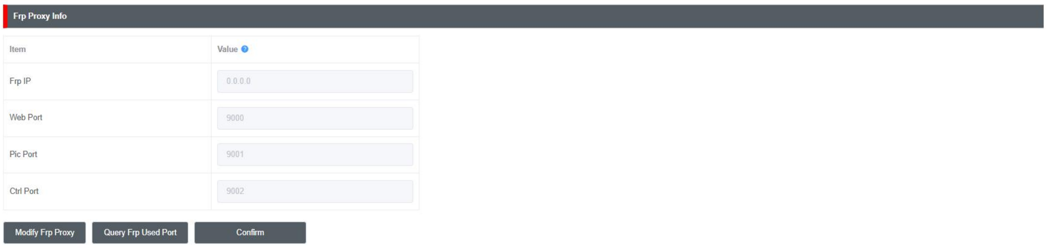



Click Modify Oscilloscope Configuration to configure the oscilloscope's local network settings, including IP acquisition mode, IP address, subnet mask, and gateway.

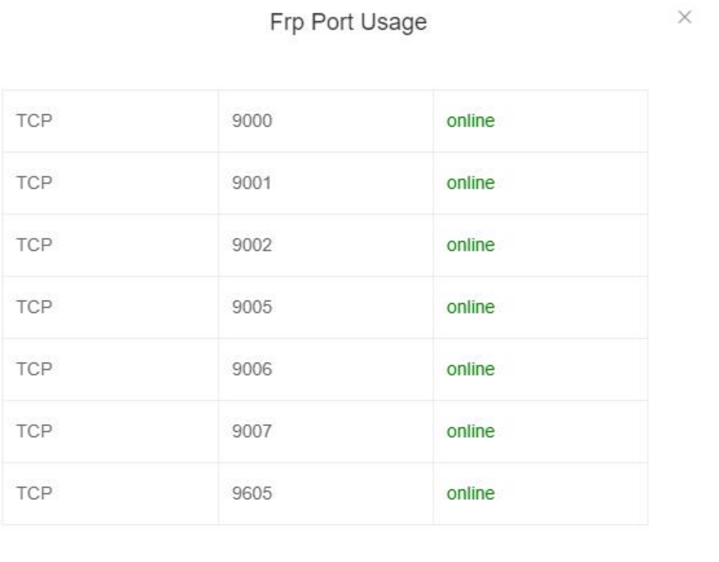
- DHCP: When DHCP is selected as the IP configuration method, there is no need to enter the IP address, subnet mask, or gateway. Click Confirm, and the oscilloscope will automatically obtain an IP address.
- STATIC: When STATIC is selected as the IP configuration method, enter the correct IP address, subnet mask, and gateway, then click Confirm.

After modifying the oscilloscope's network configuration, the user can access it using the new IP address information (provided the configuration is correct).

b. FRP Proxy Network Settings




- Modify FRP Proxy Configuration: Sets the current oscilloscope's FRP proxy information, including the FRP proxy server IP address, port, image port, and control port.
- Get FRP Used Ports: Displays the ports currently in use by the specified proxy IP. When configuring proxy ports, ensure that these occupied ports are not used.


Frp Port Usage

| | | |
|---|---|---|
| TCP | 9000 | online |
| TCP | 9001 | online |
| TCP | 9002 | online |
| TCP | 9005 | online |
| TCP | 9006 | online |
| TCP | 9007 | online |
| TCP | 9605 | online |

After entering the required information, click OK. The oscilloscope can then be accessed using the modified FRP proxy address, assuming the configuration is correct.

**Note**: When multiple oscilloscopes connect to the same FRP server, the web_port, pic_port, and ctrl_port must be unique for each instrument. Otherwise, the FRP connection will fail.
After modifying FRP proxy settings, access via LAN (IP:9000) may become unavailable.
To restore LAN access, press the Default key on the oscilloscope panel. After reset, LAN access via port 9000 will be restored.

## 22.8. Password Setting

The password setting allows users to configure their login credentials.
After logging in to the web system for the first time, users may create a custom password.
The default password is the oscilloscope serial number encoded in Base64.
Once the password is changed, the new password must be used for all subsequent logins.
If the password is forgotten, press the Default key on the oscilloscope panel to reset the

configuration.

**Note**: After resetting the oscilloscope to its default settings, you will need to log in to the web system again using the serial number encoded in Base64.


## 22.9. Help

Click Help page to open the Web usage help page, which provides basic instructions for each functional tab of the oscilloscope Web interface.

## 22.10. Service and Support

Click Service and Support page to be directed to the UNI-T official website of https://www.uni-trend.com/

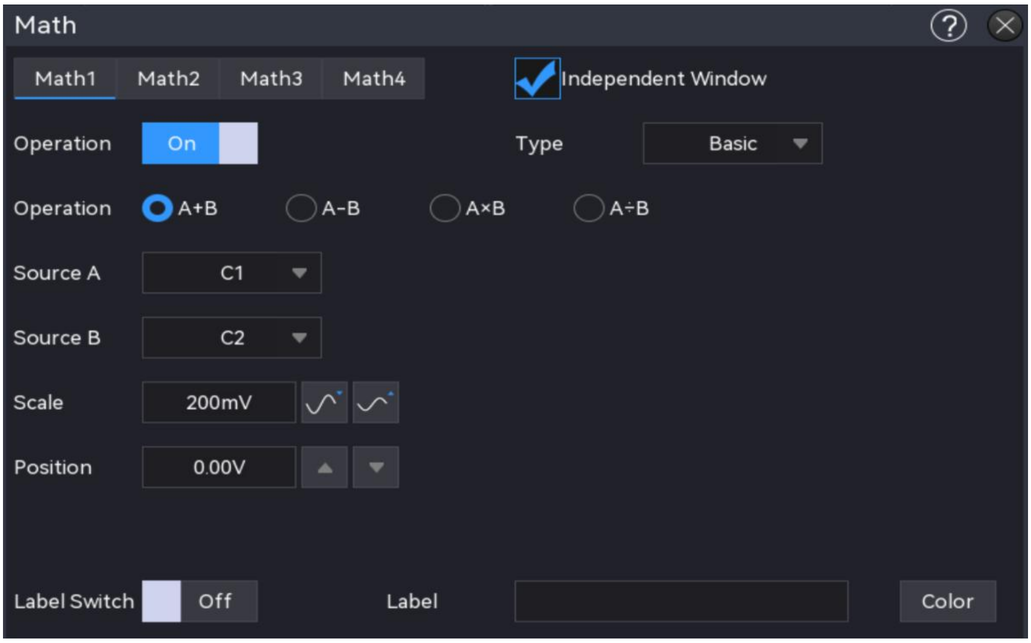
# 23. Mathematical Operation



The UPO6000LP series high-resolution oscilloscope carries a variety of mathematical operations, including Math, digital filter, and advanced operation.

Access the Math menu using the following method.

- Click Home icon in the top right-corner and select the save icon to open the math setting menu.
- If the math function is added into the toolbar, click counter icon in the toolbar in the top right-corner to enter the math setting menu.
- When M1-M4 is opened, click M1-M4 label at the bottom of screen, or click icon in the top right-corner to enter the math setting menu.



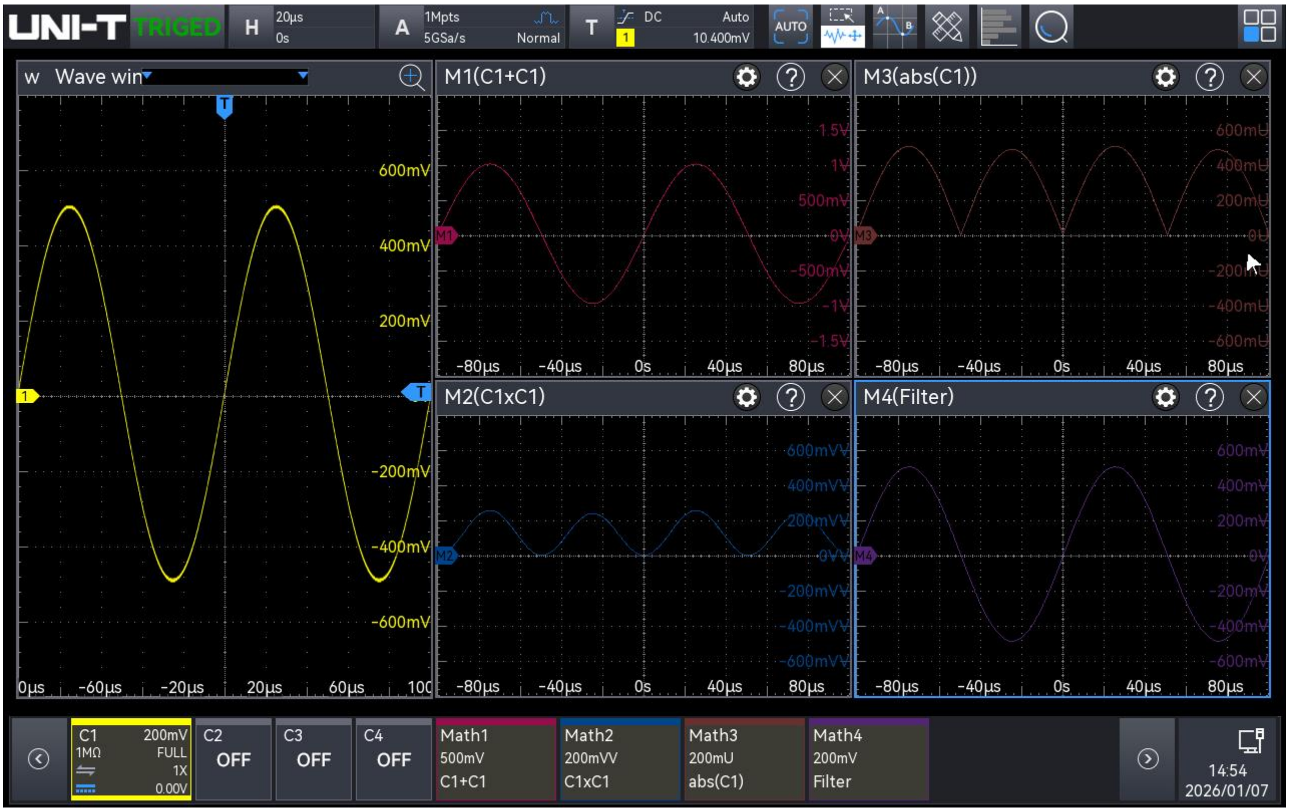

This oscilloscope supports 4 mathematical operations: Math1, Math2, Math3, and Math4. Math waveforms can be displayed in a shared window or an independent window, and both the label and channel color can be customized.

In the Math menu, tap M1–M4 to select and configure the desired math channel.

This chapter uses Math1 as an example to describe the Math function.

(1) Operation

In the Math menu, click Operation to turn the Math wave calculation on or off. The default is OFF. When M1–M4 is set to ON, the Math waveform result is displayed on the screen, as shown in the following figure.




(2) Independent Window

The operation result of Math wave can be displayed in a separate window.

Checked: 4 Math waves and the channel wave are displayed in a separate window.

Uncheck: 4 Math waves and the channel wave are displayed in the same window.

When the operation result of Math wave is displayed in a separate window, drag the label bar above the window to change the window position, or click close icon × in the upper-right corner to close the window.

(3) Label

Set the wave label for the Math wave operation result display window. For details, refer to Label for setting.

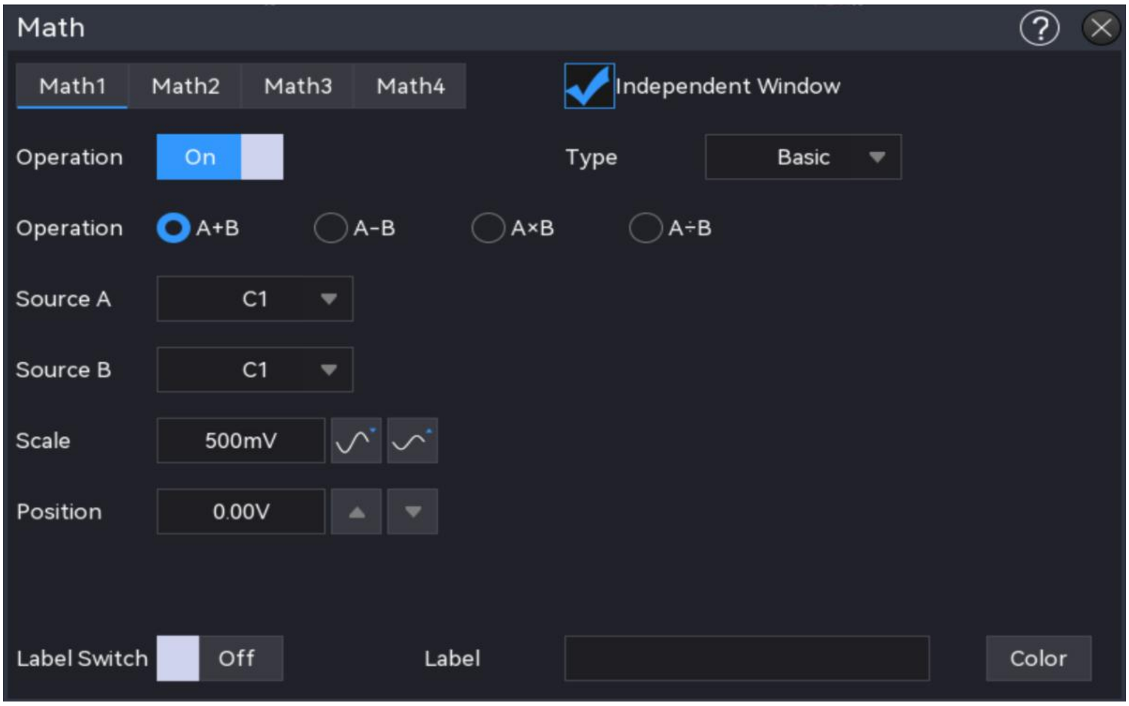
(4) Channel Color

Set the color for Math channel and Ref waveform and its label.

- Source: Click Source to select the source to set the color, the source can select M1, M2, M3, M4, R1, R2, R3, or R4.
- Color: Tap the color plate and drag to select the color.

## 23.1. Basic Operation

In the Math menu, click Type and select Basic operation to open the setting.




(1) Operation

- A+B: The waveform of source A and waveform of source B are added point by point and the results are displayed.
- A-B: The waveform of source A and waveform of source B are subtracted point by point and the results are displayed.
- A×B: The waveform of source A and waveform of source B are multiplied point by point and the results are displayed.
- A÷B: The waveform of source A and waveform of source B are divided point by point and the results are displayed. It is used to analyze the multiple relation between two channel waveforms.

**Note**: When the voltage of source B is 0, the divide result is 0.

(2) Source

Click Source A or Source B to select C1-C4 separately.

(3) Vertical Scale

Set the vertical scale of Math wave in the display window, it can be set using the following steps.

- Tap the vertical scale icon ,  on the right to increase or decrease the vertical scale.
- Double-click Vertical Scale input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter

Setting.

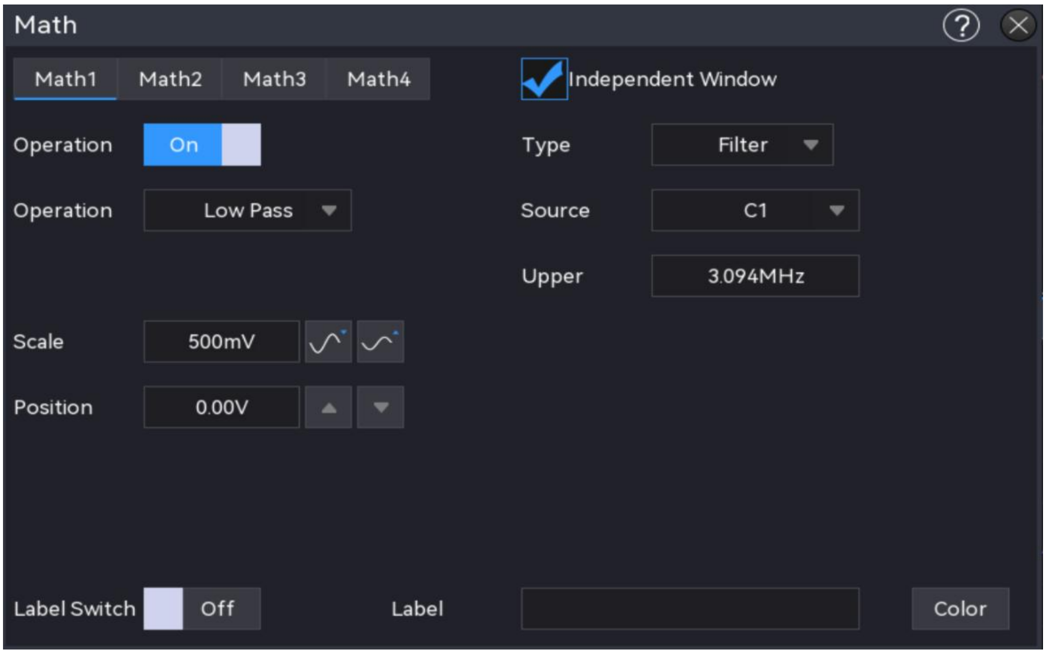
(4) Vertical Position

Set the vertical offset of Math wave in the display window, it can be set using the following steps.

- Tap the vertical position icon , on the right to increase or decrease the vertical scale.
- Double-click Vertical Position input field to open the numeric keypad and set the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

## 23.2. Digital Filter

In the Math menu, click Filter to open the setting.




(1) Source

Click Source to select C1-C4 as the trigger source.

(2) Filter Type

- Low pass: Only signals with the source frequency lower than the upper limit of the current frequency are allowed to pass.
- High pass: Only signals with the frequency higher than the lower limit of the current frequency are allowed to pass.
- Band pass: Only signal with the frequency higher than the lower limit of current frequency and lower than upper limit of the current frequency is allowed to pass.

- Band Limited: Only signal with the frequency lower than the lower limit of the current frequency or higher than the upper limit of the current frequency is allowed to pass.

(3) Lower Limit of Frequency

Double-click Lower Limit of Frequency input field to open the numeric keypad and enter the lower frequency limit directly. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. In low-pass mode, the lower frequency limit setting is invalid, and the menu is hidden.

(4) Upper Limit of Frequency

Double-click Upper Limit of Frequency input field to open the numeric keypad and enter the upper frequency limit directly. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. In high-pass mode, the upper frequency limit setting is invalid, and the menu is hidden.

**Note**: The range of upper/lower limit of frequency is related to the current horizontal time base.

(5) Vertical Scale

Set the vertical scale of Math wave in the operation result display window, it can be set using the following steps.

- Tap the vertical scale icon ,  on the right to increase or decrease the vertical scale.
- Double-click Vertical Scale input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

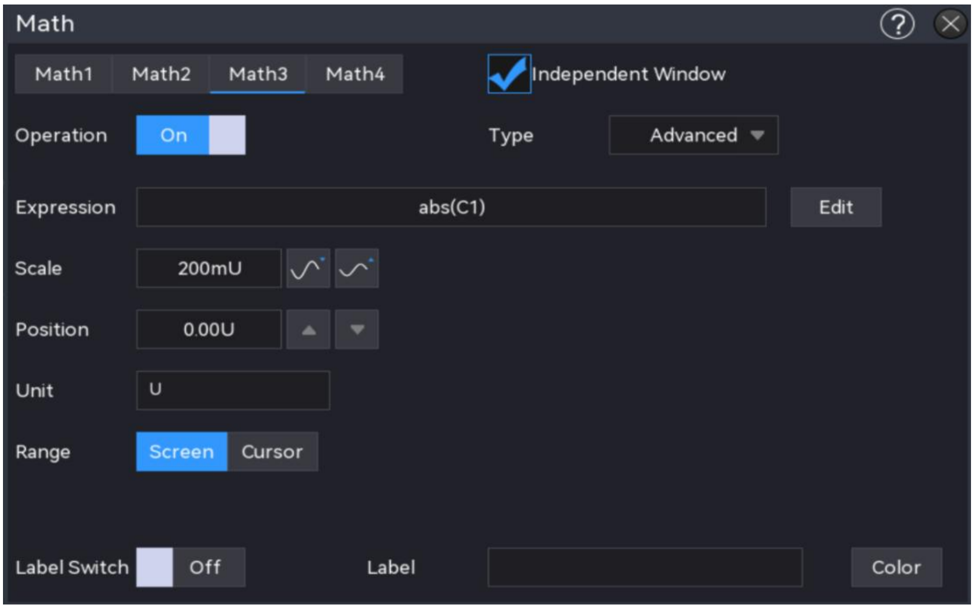
(6) Vertical Position

Set the vertical offset of Math wave in the operation result display window, it can be set using the following steps.

- Tap the vertical scale icon ,  on the right to increase or decrease the vertical position.
- Double-click Vertical Position input field to open the numeric keypad and set the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

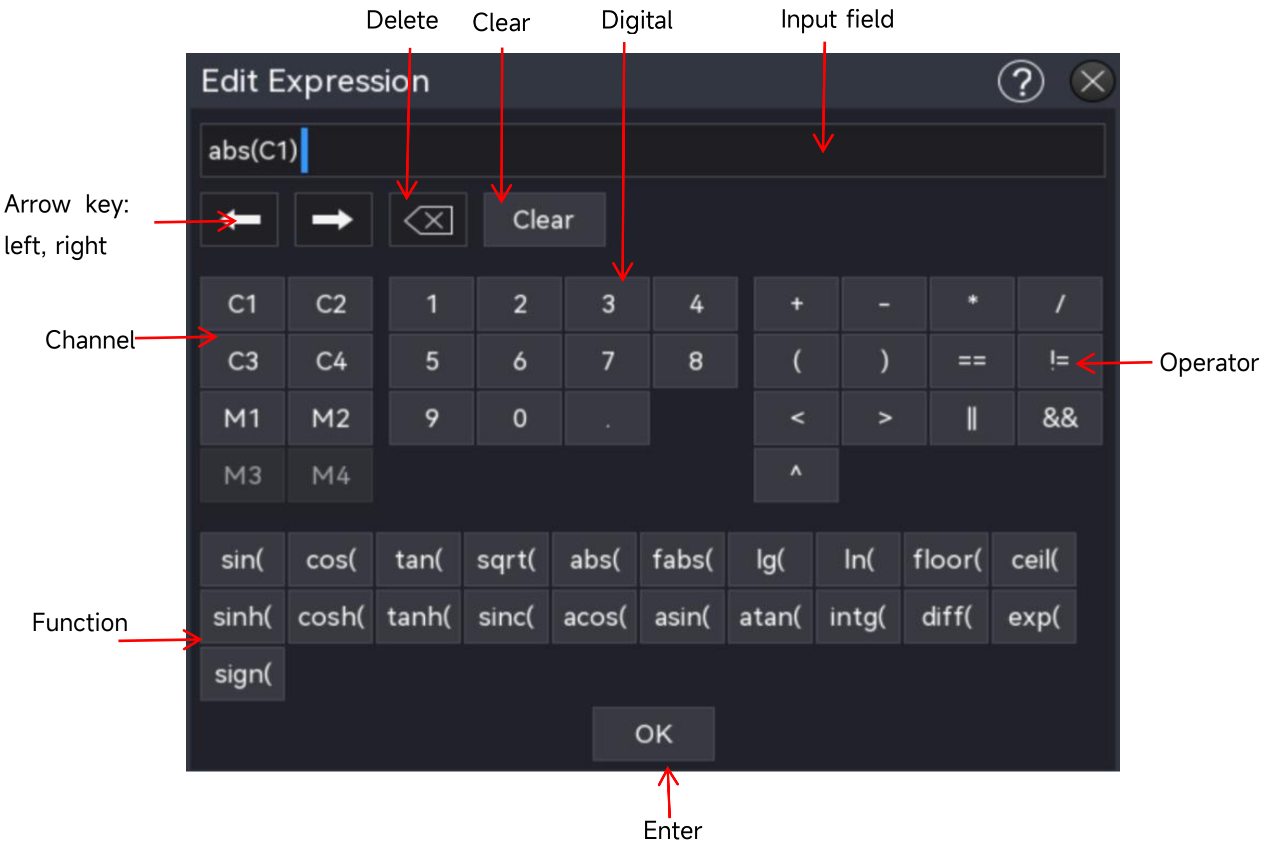
## 23.3. Advanced Operation

In the Math menu, select Type and choose Advanced to open the advanced settings. Advanced operations allow users to freely define mathematical operations for each signal input channel, enabling the generation of Math waveforms with different operation results.




(1) Expression

Double-click Expression input field or click Edit on the right open the expression editor. Mathematical functions and operations can be defined and edited in the expression window, as shown in the following figure.




Expression Window

(2) Edit Expression

a. Expression: It represents the formula consisting of channel, function, variation, and operator. The length of expression cannot be over 13 characters.

b. Channel: C1-C4, M1-M4

c. Function options: The functional description of each function option is shown in the following table.

| Function Name | Description |
|---|---|
| Sin | Calculates the sine of the selected source. |
| Cos | Calculates the cosine of the selected source. |
| Sinc | Calculates the normalization value of the selected source. |
| Tan | Calculates the tangent of the selected source. |
| abs | Takes the absolute value of the selected source (integer absolute value) |
| fabs | Takes the absolute value of the selected source (floating-point absolute value) |
| exp | Calculates the exponent of the selected source. |
| Lg | Calculates the logarithm of the selected source. |
| ln | Calculates the logarithm of the selected source. |
| floor | The selected source is round down to an integer. |
| ceil | The selected source is round up to an integer. |
| sinh | Calculates the hyperbolic sine of the selected source. |
| cosh | Calculates the hyperbolic cosine of the selected source. |
| tanh | Calculates the hyperbolic tangent of the selected source. |
| Sinc | Calculates the normalization value of the selected source. |
| acos | Calculates the arccosine of the selected source. |
| asin | Calculates the arcsine of the selected source. |
| atan | Calculates the inverse tangent of the selected source. |
| intg | Calculates the integral of the selected source. |
| diff | Calculates the differential of the selected source. |
| sign | Calculates the polarity of the selected source. |

d. Operator: The functional description of each operator is shown in the following table.

| Operator | Description |
|---|---|
| + - * / ^ | Mathematical operator: add, subtract, multiply, |

| | |
|---|---|
| | divide, exponent |
| ( ) | Parentheses are used to raise the priority of operations in parentheses. |
| < > == != | Relation operator: greater than, less than, equal to, unequal to |
| \|\| , && | Logical operator: or, and |
| 0-9 | Perform digit operation |

(3) Vertical Scale

Set the vertical scale of the Math waveform in the result display window using one of the following methods:

- Tap the vertical scale icon , on the right to increase or decrease the vertical scale.
- Double-click Vertical Scale input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(4) Vertical Position

Set the vertical offset of Math wave in the operation result display window using one of the following methods:

- Tap the vertical scale icon , on the right to increase or decrease the vertical position.
- Double-click Vertical Position input field to open the numeric keypad and set the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(5) Unit

Set the result unit of mathematical advanced operations. Double-click Unit input field to open the numeric keypad and set the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(6) Calculation Area

Advanced math calculations can be performed within a specified area. Two calculation modes are available: Screen and Cursor Area.

- Screen: Performs calculations across the entire display area.
- Cursor Area: Defines the calculation area using Cursor AX and Cursor BX. Calculations are performed within this defined area. The method for setting threshold cursors is the same as for regular cursors. Refer to Time Measurement for more details.

# 24. FFT

Using FFT (Fast Fourier Transform) mathematical operations, a time-domain signal (YT) can be converted into a frequency-domain signal. This oscilloscope provides an FFT function that allows users to view the frequency spectrum of a signal while simultaneously observing its waveform in the time domain.

- Harmonic component and distortion in measurement system
- Perform the noise feature in DC power supply
- Vibration analysis

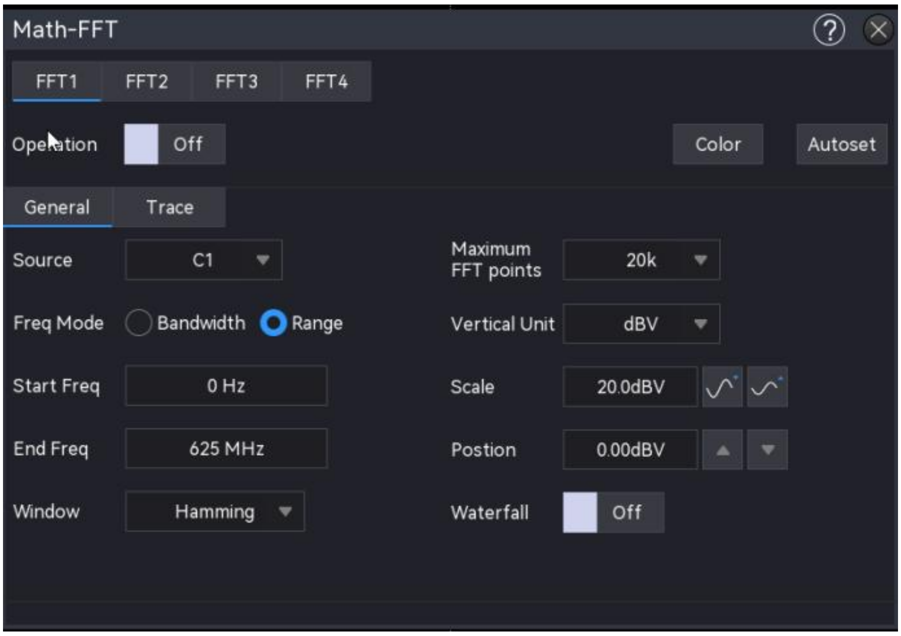
Access the FFT menu using the following method.

- Click Home icon in the top right-corner and select the FFT icon to open the FFT function.
- If the FFT function is added into the toolbar, click FFT icon in the toolbar in the top right-corner to open the FFT function.
- When FFT1-FFT4 is opened, click FFT1~FFT4 label at the bottom of the screen, click icon in the top right-corner to open the FFT function.

The oscilloscope supports four FFT operations: FFT1, FFT2, FFT3, and FFT4.

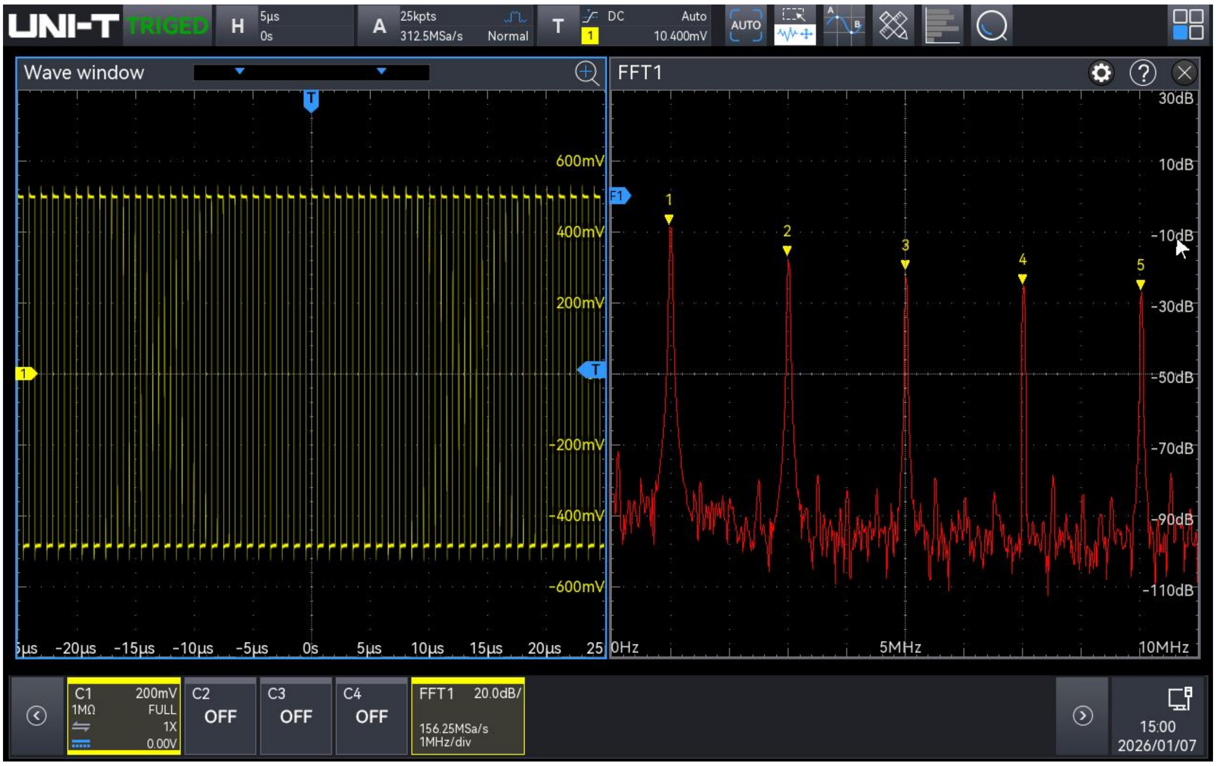
Each FFT result is displayed in a separate window. In the FFT menu, select FFT1–FFT4 to configure the corresponding FFT settings. In this chapter, FFT1 is used as an example to describe FFT operation.

FFT setting menu has two submenus, Normal and Trace. Click the Normal or Trace to open the corresponding submenu, or slide the menu to select and configure the settings.



(1) Operation

Click Operation to open the FFT operation window.




(2) Autoset

Click Autoset key to automatically adjust the vertical range and position, ensuring the FFT waveform is displayed in the center of the screen.

(3) Channel Color

Set the waveform and label colors for Math channels, Reference (Ref) waveforms, and FFT waveforms.

Click the Channel Color to open the channel color setting pop-up window, where source and color can be selected.

Source: Click the Source drop-down list to select the signal source for color configuration. Available options: M1, M2, M3, M4, R1, R2, R3, R4, FFT1, FFT2, FFT3, FFT4.

Color: Tap the color selection ring, then drag or rotate the ring to choose the desired color.

(4) Source

Click Source to select C1-C4, M1-M4, or Ref A-Ref D as the trigger source.

(5) FFT Count

The number of points processed by the FFT spectrum, it can be set to 1 k, 2 k, 5 k, 10 k, 20 k, 50 k, 100 k, 200 k, 500 k, 1M, 2M, 5M.

(6) Frequency Mode

a. Range

- Start frequency: The frequency in the left window. Click Start Frequency input field to open the numeric keypad to set the start frequency.

- Stop frequency: The frequency in the right window. Click Stop Frequency input field on the right of the window to open the numeric keypad to set the stop frequency. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

b. Bandwidth

- Center frequency: The frequency corresponds to the center of window. Click Center Frequency input field on the right of the window to open the numeric keypad to set the center frequency. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.
- Bandwidth: The frequency range of frequency domain waveform, set the bandwidth for FFT sweep. Click Bandwidth input field on the right of the window to open the numeric keypad to set the bandwidth. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(7) Vertical Unit

The FFT calculation result supports three vertical unit settings: Vrms, dBV, and dBm. Vrms, dBV, and dBm display the vertical amplitude in linear, logarithmic, and absolute power modes, respectively. dBm is available only when the channel impedance is set to 50 Ω.

For viewing the FFT spectrum over a large dynamic range, dBV is recommended.

(8) Window Function

Window functions are used to reduce spectral leakage in FFT analysis. Each window function is optimized for different waveform characteristics. Select the appropriate window function based on the type and characteristics of the measured waveform, as shown in the following table.

Window Function Table

| **Window function** | **Feature** | **Waveform** |
|---|---|---|
| Rectangle | Provides the best frequency resolution but the poorest amplitude resolution; equivalent to using no window. | Transient or short pulse: The signal level returns to nearly the same value before and after the pulse.<br>Equal-amplitude sine wave: A sine wave with nearly identical amplitude and frequency.<br>Wideband random noise: Random noise with a broad frequency spectrum that changes slowly over time. |
| Hanning | Compared with the rectangle window, it provides better | Measure sine wave, period and narrow-band random noise waveforms. |

| | | |
|---|---|---|
| | amplitude resolution but poorer frequency resolution. | |
| Hamming | Provides slightly better frequency resolution than the Hanning window. | Transient or short pulse: The signal level is very different before and after. |
| Blackman | Provides the best amplitude resolution but the poorest frequency resolution. | Measure a single-frequency signal to identify higher-order harmonics. |
| Flat Top | Provides high amplitude measurement accuracy | For signals that do not have an accurate reference standard but require precise amplitude measurements. |

(9) Vertical Scale

Set the vertical scale of FFT wave in the display window, it can be set using the following steps

- Tap the vertical scale icon ,  on the right to increase or decrease the vertical scale.
- Double-click Vertical Scale input field to open the numeric keypad and set the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(10) Vertical Position

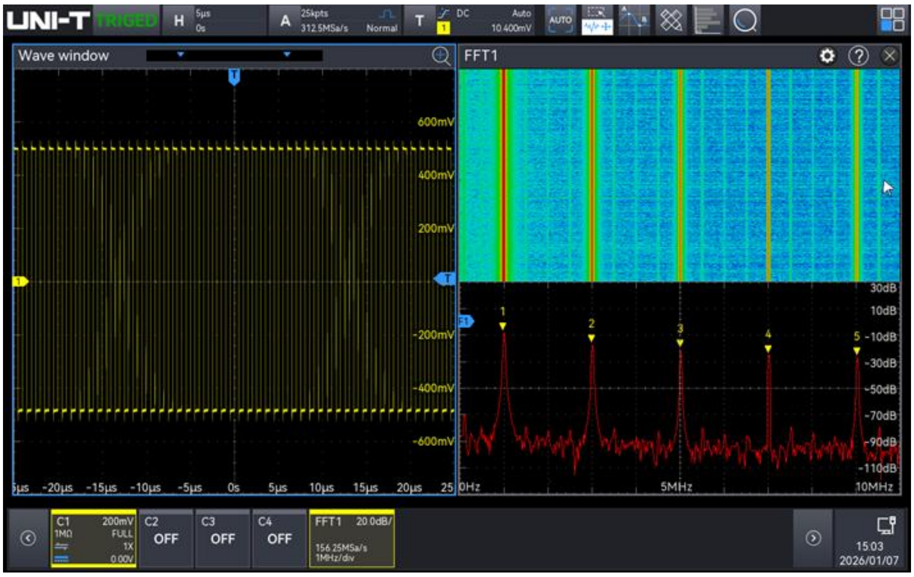
Set the vertical offset of FFT wave in the display window, it can be set using the following steps.

- Tap the vertical position icon ,  on the right to increase or decrease the vertical scale.
- Double-click Vertical Position input field to open the numeric keypad and set the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(11) Waterfall Curve

Click Waterfall Curve to turn the waterfall curve on or off in the FFT waveform.

- ON: The spectrum and waterfall curve are split to upper and lower part for display. The waterfall curve reflects the change in dB value over time in the spectrum and has record function. The record is up to 440(the spectrum with respect to the waterfall curve).
- OFF: Displays FFT wave and its coordinates.




(12) Segment Selection

After opening the waterfall chart, the Segment selection can be configured while the oscilloscope is in the STOP state. By adjusting the segment selection, the spectrum waveform corresponding to a specific time point on the waterfall diagram can be viewed. Double-click Segment selection input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 1 to 440.

(13) Trace

In FFT setting menu, click Trace r slide the menu to left and right to enter the trace setting menu. The trace is used to display the graph of the points drawn on the raw data after the FFT operation.

a. Trace

- Normal: Displays the real-time spectrum waveform using the current sampled data. The spectrum trace is shown in red.
- Average: Displays the averaged spectrum waveform calculated from multiple acquisitions. The spectrum trace is shown in blue.

  - Average time: Set the number of average calculations, double-click Average time input field to open the numeric keypad and set the average time. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range is from 2 to 8192. The greater the number, the smoother the average spectrum.
- Maximum hold: The spectrum waveform graph shows the maximum value of the data from multiple samples, and the spectrum waveform display in yellow.
- Minimum hold: The spectrum waveform graph shows the minimum value of the data from multiple samples, and the spectrum waveform displays in green.

b. Sampling Mode

- OFF: Turn off the current detection waveform.
- + Peak: It takes and displays the maximum value in each sample interval.
- - Peak: It takes and displays the minimum value in each sample interval.
- Average: It takes and displays the average value in each sample interval.
- Sampling: It takes and displays the first value in each sample interval.

(14) Spectrum Marker

The spectrum marker is used to mark points on the spectrum and display the corresponding **frequency** and **voltage** values. Marker settings include marker trace, marker type, marker point quantity, marker list, and marker sorting.

a. Marker trace: Select the spectrum waveform as the marker source. Spectrum waveforms are generated by different detection types in the detection module. Click the Marker Trace drop-down list.

   Available options: Normal, Average, Max Hold, Min Hold.

b. Marker type: Multiple marker types are supported: Auto, Threshold, Manual.

   - Auto: Automatically places markers according to the spectrum measurement results.
   - Threshold: A set threshold is used as the comparison condition. Marker points are displayed only when the peak value exceeds the threshold; otherwise, no marker points are shown.

     Double-click the Marker Threshold input field to open the numeric keypad and enter the threshold. For details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting. Alternatively, select the Marker Threshold, and use the Multipurpose rotary knob to adjust the threshold.
   - Manual: Drag the marker line manually to select marker points. Tap the Peak key to automatically move the marker cursor to the peak of the trace.

c. Marker count: Set the maximum number of marker points. Double-click the Marker Count input field to open the numeric keypad and enter the marker count. For details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting. Alternatively, select the Marker Count, and use the Multipurpose rotary knob to adjust the marker count

   Range: 1 to 50.

d. Marker list: Click the Marker list to ON or OFF.

   ON: Displays the marker information, including point number, frequency, and voltage.

   OFF: Hides the marker list.

e. Marker sorting: Set the sorting mode for marker points in the marker list.

   - Amplitude Order: Points in the marker list are sorted by amplitude in ascending order (from smallest to largest).

- Frequency Order: Points in the marker list are sorted by frequency in ascending order (from smallest to largest).
- Amplitude Reverse Order: Points in the marker list are sorted by amplitude in descending order (from largest to smallest).
- Frequency Reverse Order: Points in the marker list are sorted by frequency in descending order (from largest to smallest).

**Note**: If the selected marker trace is not currently displayed, no marker can be placed. The trace must be visible before markers can be applied.

# 25. Search and Navigation

The search function allows the user to quickly find and highlight the events of interest, and then use the event navigation to quickly find the highlighted signals to view. Waveform search criteria can be set to edge, pulse width, slope, runt, window, delay, timeout, duration, setup & hold, Nth edge and code pattern. Navigation allows the user to quickly view and locate waveforms. Navigation includes time navigation, event navigation and frame segment navigation.

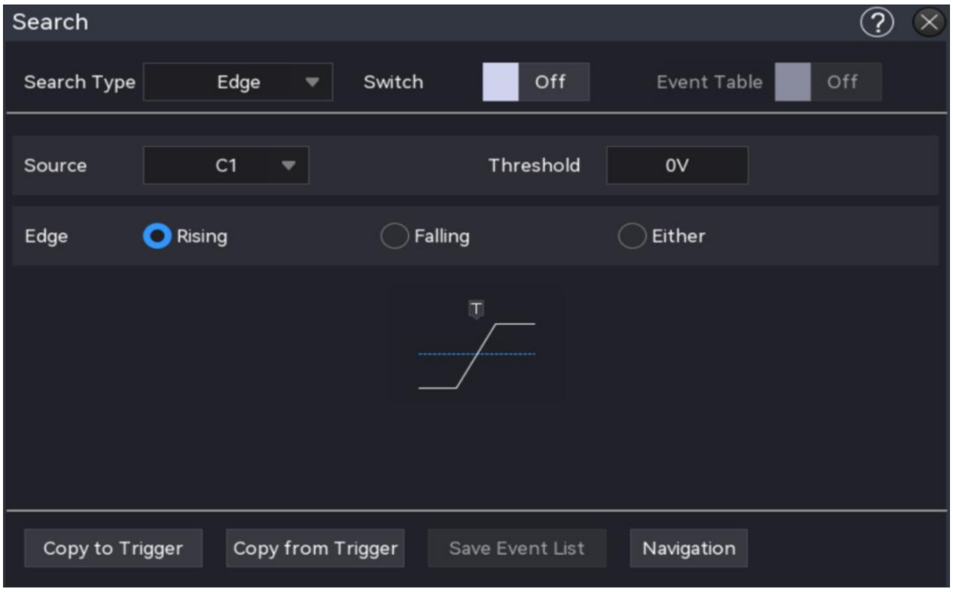
## 25.1. Search and Navigation

The search function can be opened by the following steps.

- Click Home icon , select the search icon to open the search function.
- If the search icon is added to the toolbar, click search icon in the toolbar in the top right-corner to open the search function.

The search function looks for waveform specific edge and pulse width events and marks them with small, inverted triangles ( ) along the top of the waveform scale.

Click Home icon in the top right-corner and select the search icon to open the search function, as shown in the following figure.




1. Turn on/off search function

   Click Search to turn the search function on or off.

2. Search Type

   Click Search Type to select edge, pulse width, slope, runt, over-amplitude, delay, timeout, duration, setup & hold, Nth edge, and code pattern.

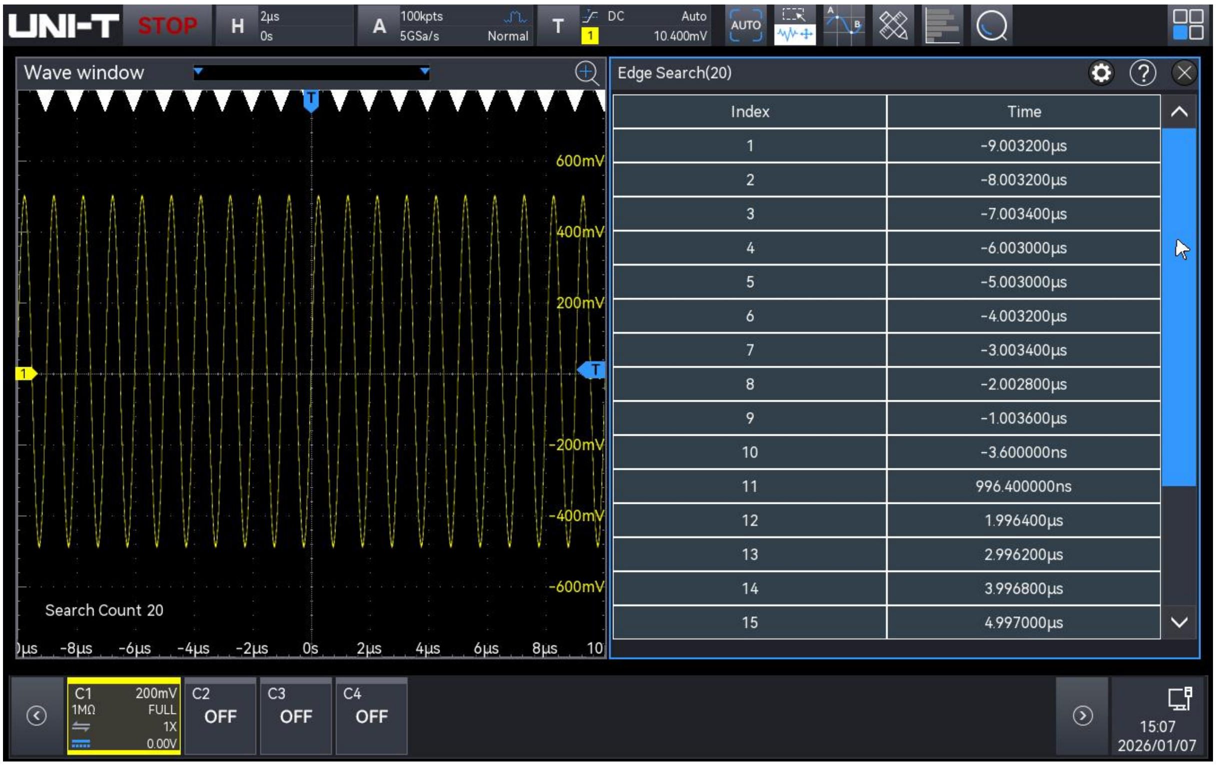
- Edge search: Click Search Type and select Edge, for the edge type setting (source, trigger coupling, edge type, trigger level). Refer to Edge Trigger for more details.
- Pulse width search: Click Search Type and select Pulse width, for the pulse width setting (source, polarity, upper limit, lower limit). Refer to Pulse Width Tigger for more details.
- Slope search: Click Search Type and select Slope, for the slope setting (source, edge type, condition, upper time limit, lower time limit). Refer to Slope Trigger for more details.
- Runt search: Click Search Type and select Runt, for the runt setting (source, polarity, runt condition, upper time limit, lower time limit). Refer to Runt Trigger for more details.
- Over-amplitude search: Click Search Type and select Over-amplitude, for the Over-amplitude setting (source, edge type, search position, over-amplitude time). Refer to Over-amplitude Trigger for more details.
- Delay search: Click Search Type and select Delay, for the delay setting (source, edge type, delay condition, upper time limit, lower time limit). Refer to Delay Trigger for more details.
- Timeout search: Click Search Type and select Timeout, for the timeout setting (source, edge type, timeout type). Refer to Timeout Trigger for more details.
- Duration search: Click Search Type and select Duration, for the timeout setting (source, code pattern, upper time limit, lower time limit). Refer to Duration Trigger for more details.
- Setup & Hold search: Click Search Type and select Setup & Hold, for the setup & hold setting (data source, clock source, edge type, data type, trigger condition, time). Refer to Setup & Hold Trigger for more details.
- Nth edge search: Click Search Type and select Nth edge, for the Nth edge setting (source, edge type, search position, time). Refer to Nth Edge Trigger for more details.
- Code pattern search: Click Search Type and select Code pattern, for the code pattern setting (source, code patter). Refer to Pattern Trigger for more details.

3. Event Table

Click switch on the right side of the Event Table to turn the marker table display on or off. When the event table is enabled, the marker table interface is displayed as shown in the figure below. The marker table lists the events for markers on waveforms currently visible in the waveform view, updating dynamically as you zoom in or adjust the waveform. The user can perform the following operations on the Marker Table:

- Click any row in the marker table while acquisition is stopped (STOP mode) selects the corresponding event, and the inverted triangle marker of the selected event turns red.
- Click icon in the top-right corner of the marker table to open the search menu.

- Press and hold the gray title bar above the event table to drag and reposition the window.
- Click icon in the top-right corner of the marker table to open the Help document.
- Click icon in the top-right corner of the marker table to close the marker table.




4. Copy to trigger

   Click Copy to Trigger to copy the settings of the selected search type to its corresponding trigger type. For example, if the current search type is Edge, click Copy to Trigger will apply the edge search settings to the Edge Trigger settings.

5. Trigger Self-Copy

   Click Trigger Self-Copy to copy the trigger settings of the selected trigger type to the search settings. For example, if the current trigger type is Edge Trigger, click Trigger Self-Copy will copy the edge trigger settings to the Edge search settings.

   To use the Trigger Self-Copy function, the search type should be set first. Then, copy the corresponding trigger settings from the trigger menu.

6. Save Event Table

   When the operating state is RUN/STOP, the time and decoding data in the current event list can be exported.

   Click Save event table key in the decoding menu to open the export setting menu, the data can be saved as *.csv, *.html, and *.pdf files to the internal storage or an external USB flash drive (when a USB is detected).

   For the setting steps, refer to the section of Save and Load.

   **Note**: When the operating state is RUN, the decoding data may be unstable, the user can manually stop the oscilloscope to export a stable decoding signal.

7. Navigation

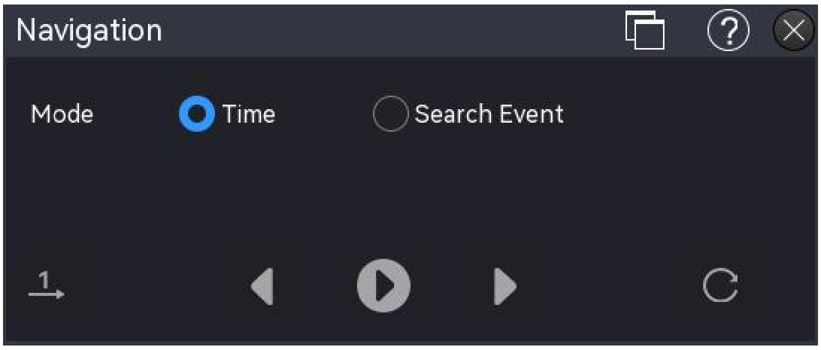
   Click Navigation key to jump to the navigation module.

## 25.2. Navigation

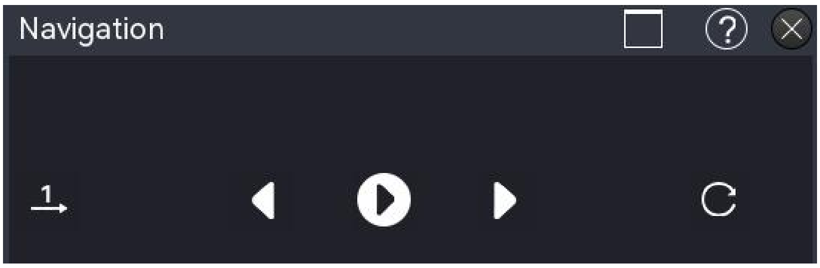
The navigation function includes time navigation and search event navigation.

The navigation menu can be accessed by the following methods.

- Click Home icon Home, select the navigation icon to open the navigation function.
- If the navigation icon is added to the toolbar, click navigation icon in the toolbar in the top right-corner to open the navigation function.
- Click Navigation key in search module to open the navigation function.




The navigation menu is shown in the figure above. Click icon in the menu to minimize the navigation menu and simplify the interface, as shown in the figure below.




**Note**: The navigation function is only available when the operation status is STOP (acquisition stopped).

(1) Time Navigation

The time navigation mode is only available in the YT time base mode.

- Playback

  After selecting time navigation, click key in the menu to start or stop playback. Use the previous or next key to move the waveform. Playback will stop automatically when it reaches either end of the waveform.

- Playback Mode

The playback mode is divided into two modes: single playback and cycle playback. Click icon in the lower left corner of the menu to switch the playback order.

: Playback starts from the start frame and ends at the end frame, stopping automatically.

: Click icon in the bottom left corner of the screen to switch between modes.

- Playback Order

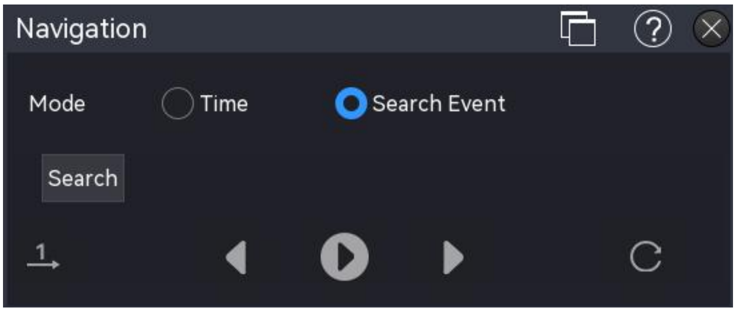
The playback order can be set to sequential playback or reverse playback. Click icon in the lower right corner of the menu to switch the playback order.

: The waveform plays to the right of the center point of the screen, moving from the center to the left.

: The waveform plays to the left of the center point of the screen, moving from the center to the right.

(2) Search Event Navigation

After completing an event search using the Search function, the user can use the Search Event Navigation to quickly view the search events. The Search Event Navigator screen is shown below.




- Search

After selecting the Search Event Navigation in the navigation interface, the user can click Search to open the Search menu and set the search conditions. Please refer to the Search Function for the relevant search settings.

- Playback

After selecting the search navigation, click key in the menu to start or stop playback. The user can use the previous key (left search point) or the next key (right search point) to display the search points in the center of the screen. Playback will automatically stop when it reaches the leftmost or rightmost mark point.

- Playback Mode

The playback mode is divided into two modes: single playback and cycle playback. Click icon in the lower left corner of the menu to switch the playback order.

: Playback automatically stops after playing from the first search point to the last search point.

: Playback repeats from the first to the last search point until the operation is stopped manually.

- Playback Order

  The playback order can be set to sequential playback  or reverse playback . Click icon in the lower right corner of the menu to switch the playback order.

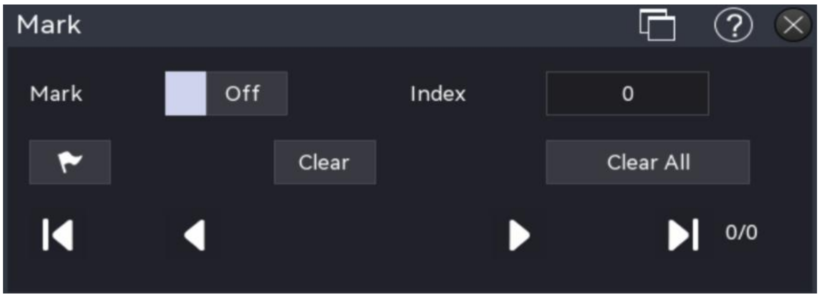
  : Play from the leftmost search point to the rightmost search point.

  : Play from the rightmost search point to the leftmost search point.

## 25.3. Marker Navigation

The marker navigation can be accessed by the following methods.

- Click Home icon Home, select the marker navigation icon  to open the marker navigation function.
- If the navigation icon is added to the toolbar, click navigation icon  in the toolbar in the top right-corner to open the marker navigation function.




(1) Marker: Click to open the marker list.

(2) Index: For the current navigation marker point, double-click Index input field to open the numeric keypad and set this value. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(3) : Mark the midpoint of the current waveform area with the marking symbol ▽.

(4) Clear: Clear the midpoint marker in the waveform area.

(5) Clear all: Clear all markers.

(6) Click previous  key to navigate to the previous marker point (decreasing the marker index number), or click next  key to navigate to the next marker point (increasing the marker index number). Click  key to navigate to the first marker point or click  key to navigate to the last marker point.

# 26. Function/Arbitrary Waveform Generator (Gen)

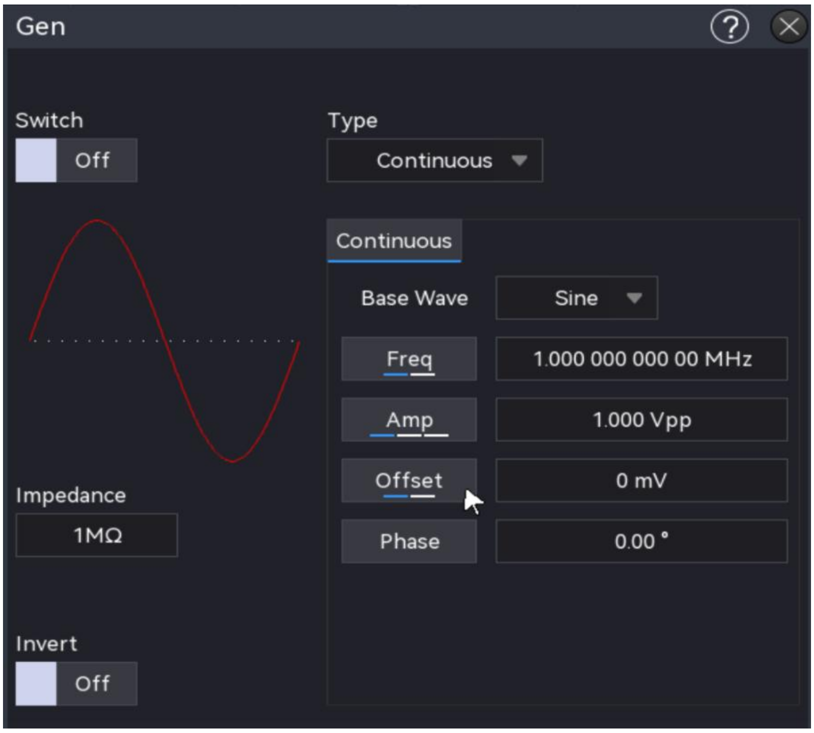


The UPO6000LP features a built-in function/arbitrary waveform generator. It uses direct digital synthesis (DDS) technology to generate accurate and stable waveforms with a frequency resolution as low as 1 μHz. The UPO6000LP provides a cost-effective solution for function/arbitrary waveform generation.

## 26.1. Open Function/Arbitrary Waveform Generator

Access the Gen menu using the following method.

- Click Home icon in the top right-corner and select the Gen icon to open the Gen menu.
- If the Gen function is added into the toolbar, click counter icon in the toolbar in the top right-corner to enter the Gen menu.




The oscilloscope supports one Gen signal outputs.

(1) Output Switch

Click Switch to turn the Gen to on or off.

ON: Outputs the current Gen signal.

OFF: Disables the Gen signal output.

(2) Output Type

Click Type to select the output signal type to continuous waveform, AM waveform, or FM waveform, amplitude shift keying (ASK), frequency shift keying (FSK), and sweep.

(3) Output Impedance

Double-click Output Impedance input field to open the numeric keypad and set the impedance. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The selected output impedance must match with the impedance of the connected oscilloscope, otherwise, the amplitude and offset level of waveform on the screen will be incorrect.

(4) Reversed Output

Click Reversed Output to turn the reversed output to on or off.

ON: Reverses the AC output signal.

OFF: The AC output signal will not be reversed.

(5) Waveform Parameter

Double-click parameter input field to open the numeric keypad and set the parameter. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(6) Waveform figure: Displays the waveform of Gen signal.

## 26.2. Carrier Waveform Output

When the Gen function is enabled, the instrument outputs a sine waveform with a frequency of 1 kHz and an amplitude of 100 mVpp by default. This section describes how to configure the instrument to output different types of waveforms.

(1) Waveform Type

Click Carrier Wave to select the waveform to sine, square, ramp, pulse wave, arbitrary, noise, or DC wave. For details on the parameters associated with each waveform, refer to the following table.

| **Continuous Wave** | **Parameter** | **Frequency Range** | **Amplitude Range** | **Offset Range** |
|---|---|---|---|---|
| Sine wave | Frequency/cycle, amplitude/high level, DC offset/low level, phase (±360°) | 1 μHz-50 MHz | 20 mVpp-6 Vpp (high impedance); 10 mVpp-3 Vpp (50 Ω) | ±3 V (high impedance); ±1.5 V (50 Ω) |

| | | | | |
|---|---|---|---|---|
| Square wave | Frequency/cycle, amplitude/high level, DC offset/low level, phase (± 360°), duty ratio (1%-99%)/pulse width | 1 μHz-15 MHz | 20 mVpp-6 Vpp (high impedance); 10 mVpp-3 Vpp (50 Ω) | ±3 V (high impedance); ±1.5 V (50 Ω) |
| Ramp wave | Frequency/cycle, amplitude/high level, DC offset/low level, phase (± 360°), symmetry (0.1%-99.9%) | 1 μHz-400 kHz | 20 mVpp-6 Vpp (high impedance); 10 mVpp-3 Vpp (50 Ω) | ±3 V (high impedance); ±1.5 V (50 Ω) |
| Pulse wave | Frequency/cycle, amplitude/high level, DC offset/low level, phase (± 360°), pulse width/ duty ratio (1%-99%), rising, falling edge | 1 μHz-15 MHz | 20 mVpp-6 Vpp (high impedance); 10 mVpp-3 Vpp (50 Ω) | ±3 V (high impedance); ±1.5 V (50 Ω) |
| Arbitrary wave | Frequency/cycle, amplitude/high level, DC offset/low level, phase (± 360°) | 1 μHz-5 MHz | 20 mVpp-6 Vpp (high impedance); 10 mVpp-3 Vpp (50 Ω) | ±3 V (high impedance); ±1.5 V (50 Ω) |
| Noise | Amplitude/high level, DC offset/low level | | 20 mVpp-6 Vpp (high impedance); 10 mVpp-3 Vpp (50 Ω) | ±3 V (high impedance); ±1.5 V (50 Ω) |
| DC | DC | | | ±3 V (high impedance); ±1.5 V (50 Ω) |

(2) Frequency

When the Gen function is enabled, the instrument outputs a default sine wave with a frequency of 1 kHz and an amplitude of 100 mVpp. Double-click Frequency input field to open the numeric keypad and set the frequency. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(3) Amplitude

The default waveform is a sine wave with an amplitude of 100 mVpp. Double-click Amplitude to open the numeric keypad and set the amplitude. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(4) DC Offset

The default DC offset of waveform is 0 V. Double-click DC Offset to open the numeric keypad and set the DC offset. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(5) Phase

The default phase of waveform is 0°. Double-click Phase to open the numeric keypad and set the phase. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(6) Duty Ratio of Pulse Wave

The default frequency is 1 kHz, and the duty ratio is 50% of the pulse wave. Double-click Duty Ratio to open the numeric keypad and set the duty ratio. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(7) Rising/Falling Time

The default rising/falling time is 12 of the pulse waves. Double-click Rising or Falling Time to open the numeric keypad and set the rising/falling time. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(8) Ramp Symmetry

The default symmetry of ramp wave is 50%. Double-click Symmetry to open the numeric keypad and set the symmetry. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

## 26.3. Advanced Application

The Gen function supports Amplitude Modulation (AM), Frequency Modulation (FM), Amplitude Shift Keying (ASK), Frequency Shift Keying (FSK), and Sweep modes. To configure modulation, open the Gen settings menu and set Type to the desired modulation mode.

(1) Amplitude Modulation (AM)

In AM, the modulation waveform consists of carrier wave and modulation wave. The amplitude of carrier waves will change with the amplitude of modulation waves.

a. Enable AM

In the Gen menu, set Type to AM. Configure the parameters for the carrier wave and AM parameters as shown in the following figure.


b. Select Carrier Wave

Click Carrier to select the carrier waveform: sine, square, ramp, or arbitrary wave.

- Carrier Wave Setting

  After selecting the AM carrier waveform, configure the corresponding parameters. For the carrier wave setting, refer to the section of Carrier Waveform.

c. Modulation Wave Setting

Modulation wave: sine, square, square, rising ramp, falling ramp, arbitrary, and noise wave. The default is sine wave. When AM is enabled, the modulation waveform defaults to sine. Click Modulation Wave to change the modulation wave type. For details on the modulation wave, refer to the following table.

| Modulation Wave | Description |
|---|---|
| Square wave | Duty ratio is 50%. |
| Rising ramp | Symmetry is 100%. |
| Falling ramp | Symmetry is 0%. |
| Arbitrary wave | Use automatic sampling to limit the arbitrary wavelength at 4 kpts. |
| Noise | White Gaussian noise |

- Modulation Frequency

  The modulation frequency range is 2 mHz to 50 kHz (default 100 Hz). When AM is enabled, the default modulation frequency is 100 Hz. Double-click Modulation

Frequency input field to open the numeric keypad and set the modulation frequency. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

- Modulation Depth

  The modulation depth indicates the change of amplitude, expressed in percentage. AM modulation depth is from 0% to 120%. The default is range is 100%. Double-click Modulation Depth input field to open the numeric keypad and set the modulation depth. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

  - When the modulation depth is 0%, it outputs a constant amplitude (half the amplitude of the carrier wave amplitude).

  - When the modulation depth is 100%，the output amplitude is change with the modulation wave.

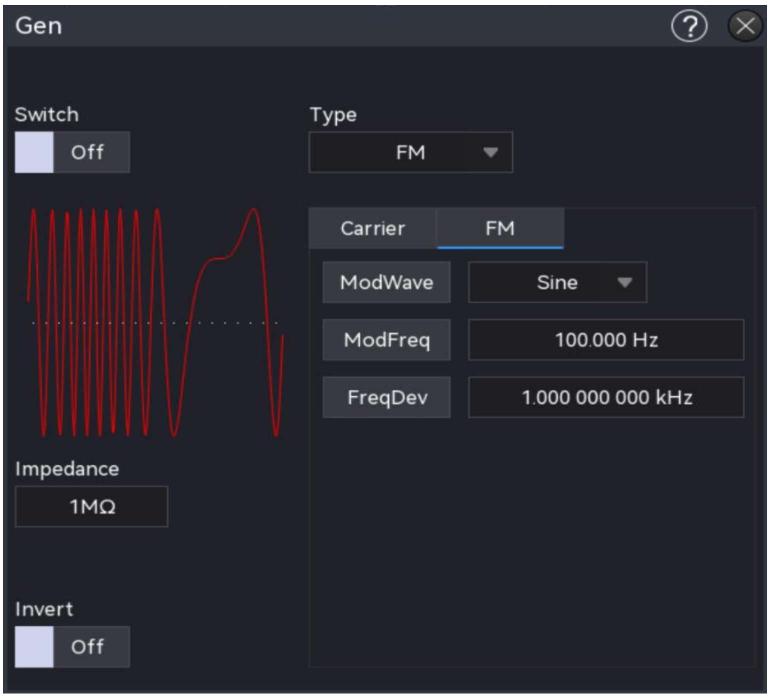
  - When the modulation depth is greater than 100%, the output amplitude will not over than 10 Vpp (the load is 50 Ω).

(2) Frequency Modulation (FM)

In FM, the modulation waveform consists of carrier wave and modulation wave. The frequency of carrier waves will change with the amplitude of modulation waves.

a. Enable FM

In the Gen menu, set Type to AM. Configure the parameters for the carrier wave and FM parameters as shown in the following figure.




b. Select Carrier Wave

Click Carrier to select the carrier waveform: sine, square, ramp, or arbitrary wave.

- Carrier Wave Setting

  After selecting the FM carrier waveform, configure the corresponding parameters. For the carrier wave setting, refer to the section of Carrier Waveform.

c. Modulation Wave Setting

Modulation wave: sine, square, square, rising ramp, falling ramp, arbitrary, and noise wave. The default is sine wave. When FM is enabled, the modulation waveform defaults to sine. Click Modulation Wave to change the modulation wave type. For details on the modulation wave, refer to the following table.

| **Modulation Wave** | **Description** |
|---|---|
| Square wave | Duty ratio is 50%. |
| Rising ramp | Symmetry is 100%. |
| Falling ramp | Symmetry is 0%. |
| Arbitrary wave | Use automatic sampling to limit the arbitrary wavelength at 4 kpts. |
| Noise | White Gaussian noise |

- Modulation Frequency

  The modulation frequency range is 2 mHz-50 kHz (default 100 Hz). When AM is enabled, the default modulation frequency is 100 Hz. Double-click Modulation Frequency input field to open the numeric keypad and set the modulation frequency. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

- Frequency Offset

  The frequency offset indicates the offset of the frequency of the FM-modulated waveform relative to the carrier frequency, and the FM frequency offset can be set from the minimum DC to half the maximum current carrier bandwidth, and the default frequency offset is 100 Hz. Double-click Frequency Offset input field to open the numeric keypad and set the frequency offset. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The maximum frequency offset is 12.5 MHz.

  - Frequency offset ≤ Carrier frequency, if the frequency offset is greater than the carrier frequency, the instrument will automatically limit the frequency offset to the maximum of the current carrier frequency.
  - The sum of frequency offset and carrier frequency ≤ Maximum frequency of the current carrier wave, if the frequency offset is set to an invalid value, the instrument will automatically limit the frequency offset to the maximum of the current carrier frequency.

(3) Output Arbitrary Wave

This oscilloscope has saved 200 arbitrary waves. Refer to Built-in Arbitrary Wave Table for details.

- Enable Arbitrary Wave Function

  This oscilloscope includes 200 built-in arbitrary waveforms. The instrument will output the arbitrary wave according to the current setting, as shown in the following figure.


- Select Arbitrary Wave

  The user can select the local built-in arbitrary wave or external arbitrary wave. When the arbitrary wave is enabled, double-click Arbitrary Wave to select the required arbitrary wave. Load the saved waveform from Gen, which includes common waveforms, mathematical waveforms, segmented modulation, biological waveforms, medical waveforms, automotive waveforms, engineering waveforms, window functions, trigonometric functions, inverse trigonometric functions, noise, and more. Refer to Built-in Arbitrary Wave Table for details.

(4) Amplitude Shift Keying (ASK)

In amplitude shift keying (ASK), digital signals “0” and “1” are represented by varying the amplitude of the fundamental signal. The system outputs different amplitudes based on the logic level of the modulating signal. The modulation modes of each channel are independent of one another, allowing users to configure either the same or different modulation modes for each channel.

a. Enable ASK

   In the Gen menu, set Type to ASK. Configure the parameters for the carrier wave and ASK parameters as shown in the following figure.


b. Select Carrier Wave

Click Carrier to select the carrier waveform: sine, square, ramp, or arbitrary wave.

- Carrier Wave Setting

After selecting the ASK carrier waveform, configure the corresponding parameters. For the carrier wave setting, refer to the section of Carrier Waveform.

c. Modulation Wave Setting

The modulation speed should be configured.

- Modulation Speed

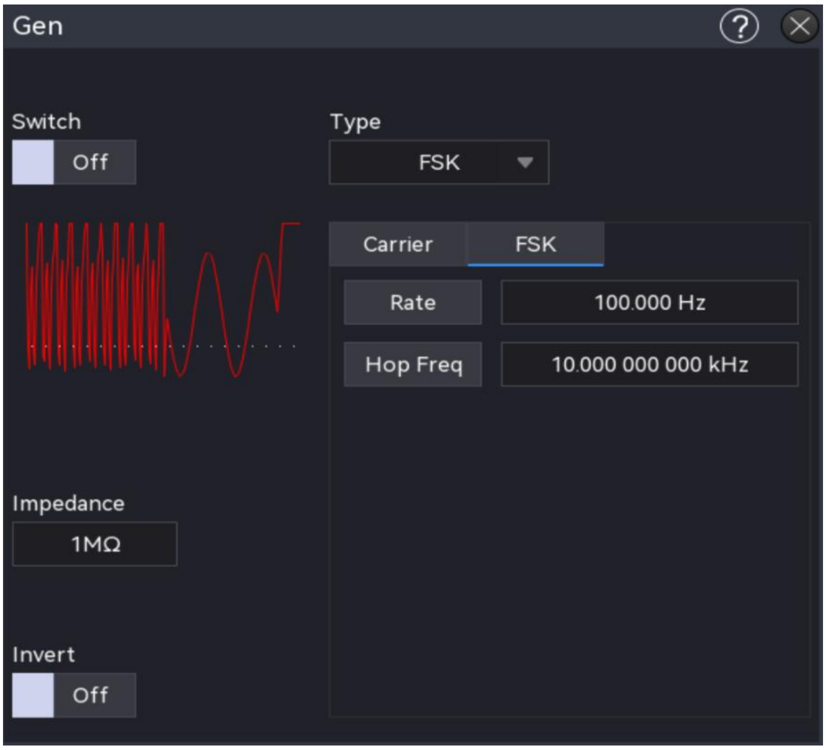
When the ASK is enabled, the modulation speed can be configured. The modulation frequency range is 2 mHz-50 kHz (default 100 Hz). When ASK is enabled, the default modulation frequency is 100 Hz. Double-click Modulation Frequency input field to open the numeric keypad and set the modulation speed. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(5) Frequency Shift Keying (FSK)

In frequency shift keying (FSK), the function/arbitrary waveform generator can be configured to switch between two preset frequencies: the fundamental frequency and the hopping frequency. Depending on the logic state of the modulating signal, either the fundamental frequency or the hopping frequency is output. The modulation modes for each channel are independent, allowing users to configure the same or different modulation modes for each channel.

a. Enable FSK

In the Gen menu, set Type to FSK. Configure the parameters for the carrier wave and FSK parameters as shown in the following figure.




b. Select Carrier Wave Setting

Click Carrier to select the carrier waveform: sine, square, ramp, or arbitrary wave.

■ Carrier Wave

After selecting the FSK carrier waveform, configure the corresponding parameters. For the carrier wave setting, refer to the section of Carrier Waveform.

c. Modulation Wave Setting

The hopping frequency and rate should be configured.

■ Hopping Frequency

When FSK is enabled, the default hopping frequency is 100 Hz. Double-click Hopping Frequency input field to open the numeric keypad and set the hopping frequency. For details on using the numeric keypad, refer to section 5.7 Parameter Setting. The setting range for the hopping frequency is determined by the carrier wave. For the specific setting range of each carrier wave, refer to the section of Carrier Waveform.

■ Rate

When FSK is enabled, the default rate is 100 Hz. Double-click Rate input field to open the numeric keypad and set the rate. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

(6) Sweep

When the sweep mode is selected, the output frequency of the Function/Arbitrary Waveform

Generator varies linearly or logarithmically from the start frequency to the stop frequency within the specified sweep time. The generator can produce Sine, Square, Ramp, Pulse, and Arbitrary waveforms (excluding DC) as sweep outputs. The sweep modes for each channel operate independently, allowing you to configure the same or different sweep modes for each channel.

a. Enable Sweep

In the Gen menu, set Type to Sweep. Configure the parameters for the carrier wave and sweep parameters as shown in the following figure.


b. Select Carrier Wave Setting

Click Carrier to select the carrier waveform: sine, square, ramp, or arbitrary wave.

- Carrier Wave Setting

  After selecting the sweep carrier waveform, configure the corresponding parameters. For the carrier wave setting, refer to the section of Carrier Waveform.

c. Sweep Setting

The start frequency, stop frequency, and sweep time should be configured.

- Sweep Type

  The function/arbitrary waveform generator supports two types of sweeps: linear and logarithmic.

  ① Linear Sweep: The waveform generator changes the output frequency linearly throughout the sweep.

  ② Logarithmic Sweep: The waveform generator changes the output frequency logarithmically during the sweep.

- Start Frequency, Stop Frequency

  The start frequency and stop frequency represent the lower and upper limits of the frequency sweep. The function/arbitrary waveform generator always sweeps from the start frequency to the stop frequency and then returns to the start frequency. Double-click Start Frequency/ Stop Frequency input field to open the numeric keypad and set the frequency. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

  ① When the start frequency < stop frequency, the function/arbitrary waveform generator sweeps from low frequency to high frequency.

  ② When the start frequency > stop frequency, the function/arbitrary waveform generator sweeps from high frequency to low frequency.

  ③ When the start frequency = stop frequency, the function/arbitrary waveform generator outputs a fixed frequency.

  By default, the start frequency is set to 1 kHz and the stop frequency to 2 kHz. However, the range of configurable starting and stopping frequencies varies for different sweep waveforms. Amplitude modulation for the frequency setting ranges of each sweep waveform, refer to the section of Carrier Waveform.

- Sweep Time

  Set the time required to sweep from the start frequency to the stop frequency. The default is 1 second, with a configurable range of 1 ms to 500 s. Double-click Sweep Time input field to open the numeric keypad and set sweep time. For details on using the numeric keypad, refer to section 5.7 Parameter Setting.

# 27. Toolbar

Click Home icon  in the top-right corner of the screen, then Click Edit key at the bottom to enter the toolbar menu. The toolbar allows editing, deleting, and rearranging shortcut menu icons.


(1) Add Toolbar

The toolbar contains all the function icon that can be added to the toolbar. Click to select the function to be added to the toolbar, click it again to deselect. The selected icon displays √ in the top right-corner.

(2) Sequence

In the toolbar pop-up window, the function menu added to the toolbar can be adjusted by dragging the corresponding icons left and right, to adjust the order of the icons displayed in the toolbar.

(3) Default Setting

By default, UPO6000LP toolbar displays 6 icons: AUTO, Area Draw, Cursor, Measure, Area Histogram, and Search.

(4) Delete

Delete all the menus from the toolbar.

# 28. Histogram

The histogram performs statistical analysis on the waveform in both the vertical and horizontal directions and can be used for jitter analysis and signal integrity analysis.

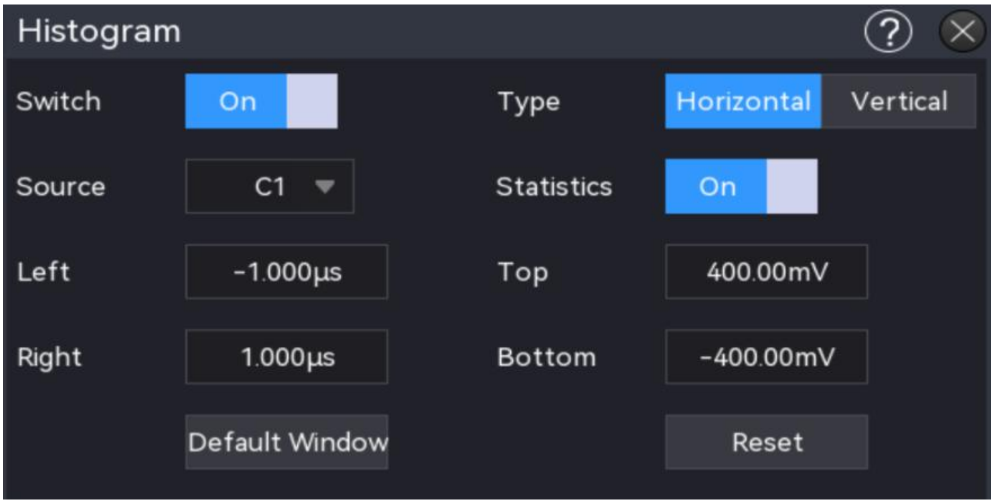
Jitter analysis: The histogram is essential for jitter analysis, enabling statistical evaluation of the value range and distribution of many data samples. This allows verification of product performance and quality, as well as identification and diagnosis of intermittent issues. It is particularly valuable for analyzing random events such as noise or jitter, facilitating the assessment and verification of jitter characteristics.

Signal integrity analysis: In signal integrity analysis, the histogram displays the amplitude distribution of the signal. By examining the histogram, users can evaluate signal stability and quality, detect abnormal values or fluctuations, and ensure the integrity and reliability of the signal.

The histogram is divided into vertical and horizontal histograms. The window is divided into several rows and columns. When the histogram is open, the default size of the measurement window is ±2 divisions (vertical/horizontal). Click and drag the histogram measurement window to move its position.

Access the Histogram menu using the following method.

- Click Home icon in the top right-corner and select the histogram icon to open the histogram menu.
- If the histogram is added to the toolbar, click histogram icon to open the histogram menu.
- When the histogram pop-up window is opened, click icon to open the histogram menu.




(1) Switch

Click Switch to turn the histogram function on or off.

(2) Type

Click Type to select the histogram statistic type.

- Horizontal: Divides the limit window into multiple columns and displays the number of triggers in each column as a histogram at the bottom of the grid.
- Vertical: Splits the limit window into rows and displays the number of triggers in each row as a histogram on the left side of the grid.

(3) Source

Click Source drop-down menu to select the source for histogram statistical analysis. The available source channels are analog channels C1 to C4.

(4) Histogram Statistics

Click Statistics to turn the statistics function on or off.

- ON: Displays the histogram statistics pop-up window.
- OFF: Hides the histogram statistics pop-up window.

(5) Histogram Boundary

The histogram measurement window allows setting the left/right and upper/lower boundaries. When the histogram is opened, a default histogram area is generated. There are two methods to set the boundaries.

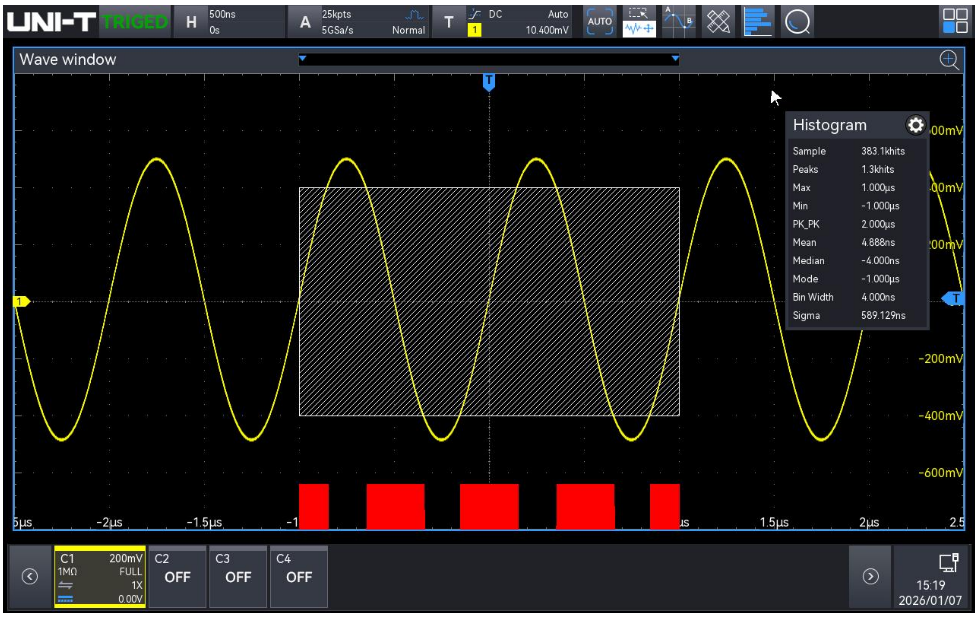
- In the histogram pop-up window, select the Left Boundary, Right Boundary, Upper Boundary, or Lower Boundary input windows to adjust the boundaries. Boundaries can be set to the screen area, with fine tuning available for precise adjustments.
- Select and drag the four histogram boundaries to modify the size of the measurement window. This method is more suitable for coarse adjustments.

(6) Default Window

Click Default to set the histogram statistics window to ±2 div vertically and ±2 div horizontally.

(7) Reset

Click Reset to clear the histogram statistics and restart the measurement.




The visual component of the histogram is a bar graph displayed to the left of the vertical waveform histogram or at the bottom of the horizontal waveform histogram. As waveform data is continuously acquired or measurements are updated, the height of each bar changes to reflect the distribution and frequency of the measured values within the defined histogram range. The bar graph represents how often specific value ranges occur, allowing users to observe peaks, spread, and overall distribution characteristics.

(8) Histogram Statistical Results

Histogram data results are displayed in the Histogram Statistics pop-up window, which can be opened to view the statistics.

- Sample: The total number of samples that fall within the histogram area.
- Peaks: The number of samples that fall within the highest bar area.
- Max: The maximum value in the sample.
- Min: The minimum value in the sample.
- Pk-Pk: The difference between the maximum and minimum values.
- Mean: The average of all statistical samples (mathematical expectation).
- Median: The value that divides the histogram into two equal parts, each containing the same number of samples.
- Mode: The most frequently occurring data value in the statistical sample.
- Bin Width: The width of each bar in the histogram, representing the width of a column.
- Sigma: The standard deviation (σ) of all statistical samples.

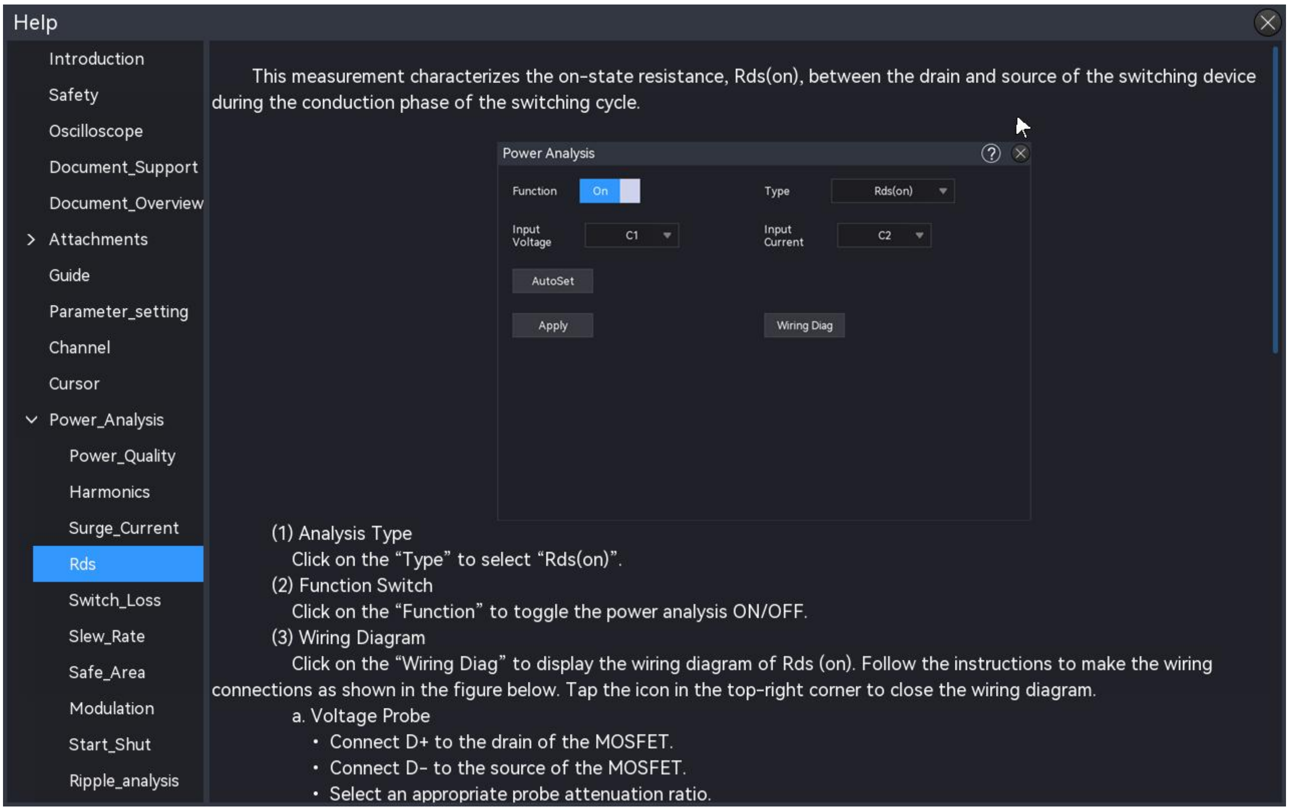
# 29. Help System

The help system provides information about the function keys (including menu keys) on the front panel.

The user can access the help system by following these steps.

- In Home menu, click Help icon to open the help menu.
- In each function men, click Help icon in the top right to open the relevant help menu.

The help screen is divided into two parts, the help options on the left and the help display area on the right. Selecting a help option displays the corresponding content in the help display area.

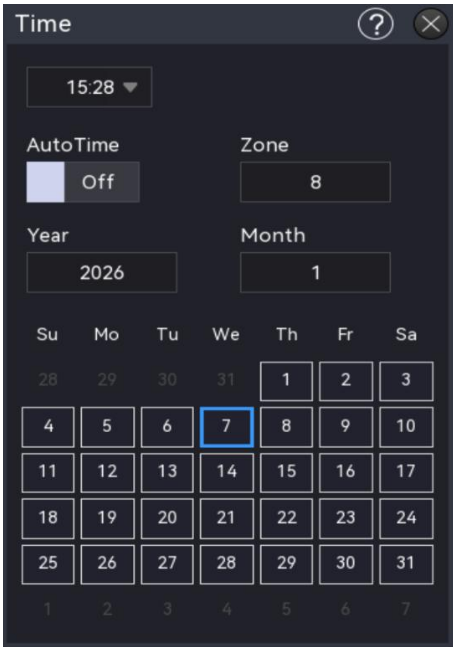


# 30. Notification Setting

Click notification area at the bottom of the screen to open the time settings and file browser settings.


## 30.1. Time Setting

Click Time at the bottom of the screen to open the time setting menu, and set auto time, time zone, year, month, date, and time (hours, minutes, and seconds).




(1) Auto Time

Click Auto Time to turn the setting on or off. The automatic time setting can only synchronize with Beijing Time when the oscilloscope is connected to a network. If the oscilloscope is not connected, the time will remain based on the current manual settings.

(2) Time Zone

Double-click Zone input field to open the numeric keypad and set the time zone. For the details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting. The time zone can be set from -11 to 12.

(3) Year

Double-click Year input field to open the numeric keypad to enter the year. For the details on the use of the numeric keyboard, refer to the section of 5.7 Parameter Setting. The year can be set from 2000 to 2050.

(4) Month

Double-click Month input field to open the numeric keypad to enter the month. For the details on the use of the numeric keypad, refer to the section of 5.7 Parameter Setting. The time zone can be set from 1 to 12.

(5) Date

Double-click Date at the bottom of the screen to set the date, the selected date is displayed in blue.

(6) Time

Double-click Time to display the two dial wheels for setting the current time. Scroll the left wheel to set the hour and the right wheel to set the minutes. Click blank area to confirm the time setting.

## 30.2. File Browser

Click the USB icon at the bottom right to directly enter the file browser. For details on using the file browser, refer to the section of File Browser.

# 31. Additional Function Key



## 31.1. Automatic Setting

The Auto function automatically adjusts the time base, vertical scale, and trigger parameters based on the characteristics of the input signal, allowing the waveform to be displayed clearly on the screen without manual adjustment. Click the AUTO icon in the toolbar to enable automatic settings.

a. Automatic settings are suitable only for simple, single-frequency signals. Effective automatic adjustment cannot be achieved for complex or combined waveforms.

b. The measured signal must have a frequency of at least 10 Hz and an amplitude of at least 12 mVpp, and the duty cycle of a square wave must be greater than 5%.

## 31.2. Factory Setting

In the auxiliary menu, click About, then click Reset to quickly restore the oscilloscope to factory settings. All locally saved data will be deleted. The factory settings of UPO6000LP series are listed in the table below.

| System | Function | Factory Setting |
|---|---|---|
| Vertical System | C1 | 200 mV/div |
| | Vertical offset | 0 (Vertical midpoint) |
| | Offset | 0 (Vertical midpoint) |
| | Coupling | DC |
| | Bandwidth limit | Full bandwidth |
| | Volts/div scale | Coarse tuning |
| | Probe | 1× |
| | Inverse Phase | OFF |
| | Unit | V |
| | C2 - C4 | OFF |
| | MATH, FFT, REF | OFF |
| Horizontal System | Extension window | OFF |
| | Fine tuning | Coarse tuning |

| | | |
|---|---|---|
| | Horizontal extension | Center |
| | Auto roll mode | OFF |
| | Mode | YT |
| | Source X-Y | C1-C2 |
| | Horizontal time base | 1 μs/div |
| | Horizontal offset | 0 (Horizontal midpoint) |
| Trigger System | Trigger type | Edge |
| | Trigger polarity 1 | Rising edge |
| | Coupling mode | DC |
| | Level | 0 V |
| | Trigger mode | Auto |
| | Trigger holdoff | 80 ns |
| | Source 1 | C1 |
| | Noise suppression | OFF |
| Sampling system | Acquisition mode | Normal |
| | Storage depth | Auto |
| | Interpolation | sinc |
| Display | Display format | Vector |
| | Grid display | Full display |
| | Window Transparence | 50% |
| | Persistence | Auto |
| | Stop persistence | Unchecked |
| | Temperature color | OFF |
| | Grid brightness | 50% |
| | Waveform brightness | 50% |
| MATH | Type | Basic operation |
| | Source A | C1 |
| | Operator | + |
| | Source B | C1 |
| | Vertical scale | 200 mV |
| | Vertical position | 0 V |
| | Label state | OFF |
| FFT | FFT window function | Hanning |
| | FFT unit | dB |
| | FFT count | 8 k |

| | | |
|---|---|---|
| | Waterfall curve | OFF |
| Measurement | Parameter snapshoot | OFF |
| | Parameter measurement | OFF |
| | Measurement statistical | OFF |
| | Digital voltmeter | OFF |
| | Frequency counter | OFF |
| | Indicator | OFF |
| Bus Decoding | Decoding type | RS232 |
| | Bus state | OFF |
| | Display format | Hexadecimal |
| | Event list | OFF |
| | Bus position | 0 |
| | RS232 source | C1 |
| | RS232 polarity | Positive |
| | RS232 bitrate | 100 bps |
| | RS232 bit width | 5 bits |
| | RS232 bit sequence | LSB |
| | RS232 stop bit | 1 bit |
| | RS232 parity bit | None |
| Other System | Square wave | 1 kHz |
| | Synchronized output | Idle |
| | IP | DHCP |
| | Language | Current setting |
| | Cursor type | OFF |
| | Horizontal cursor | Fixed delay |
| | Synchronized move | OFF |
| | Auto channel | Autoset |
| | Auto sampling | Autoset |
| | Auto trigger | Autoset |
| | Auto signal | Autoset |
| | Channel | C1 |
| | RUN/STOP | RUN |

# 32. System Prompt and Troubleshooting

- System Prompt
- Troubleshooting

## 32.1. System Prompt

This chapter describes the system prompts, with detailed explanations provided in the table below.

| | |
|---|---|
| USB flash drive is inserted. | This prompt appears when a USB flash drive is successfully inserted. |
| USB flash drive is removed. | This prompt appears when a USB flash drive is successfully removed. |
| Save path: | This prompt appears when waveform, image, setting file, decoding list, or Bode plot list (csv, pdf, html) is successfully saved. |
| Activation successful | This prompt appears when an option is activated again. |
| No activation file! | This prompt appears when a USB flash drive is inserted but contains no corresponding activation file. |
| Autoset is completed. | This prompt appears when Autoset is completed. |
| Frequency too low, please check! | This prompt appears when no signal is applied during power analysis. |
| Fine Tuning: On | This prompt appears when fine tuning for volts/div, time base is turned on. |
| Fine Tuning: Off | This prompt appears when fine tuning for volts/div, time base is turned off. |
| System upgrade is succussed. | This prompt appears when the system upgrade is succussed. |
| 10 MHz synchronization successful. | This prompt appears when 10 MHz synchronization input is successful. |
| Automatically adjusting memory depth according to FFT Points | This prompt appears when setting oscilloscope storage depth while opening FFT. |
| Load is succussed. | This prompt appears when setting files are successfully loaded. |
| Creation Successful. | This prompt appears when a folder is successfully |

| | |
|---|---|
| | created in the file browser. |
| Please select a file. | This prompt appears when renaming, copying, deleting, importing REF files, loading test masks, loading Bode plot CSV data, or selecting arbitrary waveform with no content selected. |
| Deletion Successful. | This prompt appears when a file is successfully deleted. |
| Copy Successful. | This prompt appears when a file is successfully copied. |
| Renaming Successful. | This prompt appears when a file is successfully renamed. |
| Paste Successful. | This prompt appears when a file is successfully pasted. |
| Activated. | This prompt appears when an option is activated. |
| Function in trial mode; for long-term use, please purchase an activation code! | This prompt appears when using an option during the trial period. |
| Function not activated; please purchase an activation code! | Function not activated; please purchase an activation code! |
| Auto-calibration completed. | This prompt appears when auto-calibration is completed. |
| Auto-calibration exited. | This prompt appears when exiting during the auto-calibration process. |
| Delete this file? | This prompt appears when prompting to delete a file. |
| File already exists; do you want to overwrite it? | This prompt appears when saving a file but a file with the same name already exists in the specified path. |
| Already adjusted to the limit! | This prompt appears when adjustments have reached the limit and cannot be continued. This occurs when adjusting the vertical scale factor, time base, horizontal position, vertical position, and trigger level to their limits. |
| No valid data! | This prompt appears when loading a REF file that contains errors. |
| FFT is disabled in the current mode. | This prompt appears when opening FFT and entering SCAN or ROLL modes. |
| AWG channel overloaded; please check the circuit! | This prompt appears when the AWG is overloaded with a power signal. |

| | |
|---|---|
| Analog channel overloaded; please check the circuit! | This prompt appears when analog channels C1 - C4 are overloaded with a power signal. |
| Exceeds the maximum length. | This prompt appears when input exceeds the maximum length in the letter or number keyboard. |
| 10 MHz synchronization clock input not detected! | This prompt appears when the 10 MHz synchronization signal is not connected. |
| The signal generator is not turned on. Please open it and try again! | This message appears when the signal generator is off while performing waveform save, waveform CSV export, REF quick reference, or pass/fail operations. |
| USB drive is not inserted! | This prompt appears when the USB drive is not connected during option activation or full activation operations. |
| License is not detected. Please purchase it and try again. | This prompt appears when the USB drive is connected but no activation file is found during option activation or full activation operations. |
| The current fundamental wave does not support the selected modulation; switching to a supported fundamental wave! | This prompt appears when an unsupported modulation fundamental wave is selected in continuous wave mode. |
| Too many parameters; please delete measurement items first! | This prompt appears when 27 custom parameters are added in Measure, and an attempt is made to add more. |
| This function is not supported in average sampling mode! | This prompt appears when attempting to record waveforms in average sampling mode. |
| Input channel has no signal! | This prompt appears when there is no signal connected during the Autoset operation. |
| Waveform recording function is not supported in the current mode! | This prompt appears when attempting to record waveforms in ROLL/SCAN mode. |
| This operation is prohibited during waveform recording! | This prompt appears when attempting to modify time base settings, vertical position, open FFT, open XY, or expand windows during waveform recording. |
| Time base setting is not in the recommended range; bandwidth limitation may be inaccurate! | This prompt appears when adjusting time base settings for custom bandwidth. |
| Disk is full. | This prompt appears when there is insufficient disk space to continue saving content. |

| | |
|---|---|
| Expression error. | This prompt appears when there is an error in mathematical expressions input for advanced calculations. |
| Upgrade failed. | This prompt appears when the upgrade fails or is canceled. |
| Fundamental wave type error. | This prompt appears when an unsupported fundamental wave type is selected for amplitude modulation (AM), frequency modulation (FM), amplitude shift keying (ASK), frequency shift keying (FSK), or sweeping. |
| Loading failed. | This prompt appears when loading a mask fails during testing. |
| Paste failed. | This prompt appears when the copied file has been deleted and a paste operation is attempted. |
| Directory does not exist. | This prompt appears when the corresponding file cannot be found in the import file path due to the directory being deleted. |
| No upgrade file. | This prompt appears when there is no upgrade file available for the upgrade operation. |
| Parameter error. | This prompt appears when Gen parameters are set to their limit values during channel copy operations. |
| Invalid address. | This prompt appears when manually setting the IP address, subnet mask, or gateway incorrectly. |
| File name is empty. | This prompt appears when there is no content entered in the file name input window. |
| Please pause before saving the waveform in ROLL/SCAN mode! | This prompt appears when attempting to save a waveform in ROLL/SCAN mode. |
| Already at the latest version! | This message appears when the system is already running the latest version. |
| Incorrect upgrade package! | This prompt appears when an incorrect upgrade file is selected for the upgrade process. |
| Please use the navigation in Stop mode! | This prompt appears when entering the navigation function while in Run mode. |

## 32.2. Troubleshooting

(1) If the oscilloscope remains on a black screen without any display when pressing the power soft key.

a. Check if the power plug is properly connected and the power supply is normal. If the power switch is turned on, the power soft key on the front panel should be red. There should be a normal relay sound when the soft switch key is pressed.

b. If a relay sound is heard, it indicates that the oscilloscope has booted normally. Press the Default key and then the OK key. If the display returns to normal, this indicates that the waveform brightness or grid brightness was set too low.

c. Restart the oscilloscope after completing the above steps.

d. 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.

a. Check whether BNC connectors are connected properly.

b. Check whether the signal output channel is enabled.

c. Check whether the signal connecting line is connected to analog channel.

d. Check whether the signal source has DC offset.

e. Plug out the connected signal, to check whether the base line is within the screen range (If not, please perform self-calibration).

f. If the product still does not work properly, contact the UNI-T Service Center for assistance.

(3) The measured voltage amplitude value is 10 times larger or 10 times smaller than the actual value.

Check that the channel probe attenuation settings are consistent with the attenuation rate of the probe being used.

(4) There is a waveform display but not stable.

a. Check the trigger settings in trigger menu whether is consistent with the actual signal input channel.

b. Check the trigger type: The general signals should use Edge trigger. The waveform can only be displayed stably if the trigger mode is set correctly.

c. Try to change trigger coupling to HF rejection or LF rejection, to filter out the high-frequency or low-frequency noise that interfere with the trigger.

(5) No waveform is displayed after pressing the Run/Stop key.

a. Check whether the trigger mode is set to Normal or Single and verify if the trigger level exceeds the waveform range.

b. If the trigger mode is normal or single and the trigger level is in the center, set the trigger mode to Auto.

c. Press the AUTO key to automatically complete the above settings.

(6) Waveform refresh is very slow.

a. Check whether the acquisition method is set to Average and if the average times are large.

b. Check whether the memory depth is maximum

c. Check whether the trigger holdoff is large.

d. Check whether it is normal trigger and is slow time base.

e. All the above will lead to slow waveform refresh, it is recommended to restore the factory settings, then the waveform can be refreshed normally.

# 33. Appendix

## 33.1 Appendix A Maintenance and Cleaning

(1) General Maintenance

Keep the instrument away from the direct sunlight.

**Caution: Keep sprays, liquids and solvents away from the instrument or probe to avoid damaging the instrument or probe.**

(2) Cleaning

Check the instrument and probe frequently according to the operating condition. Follow these steps to clean the external surface of the instrument:

a. Please use a soft cloth to wipe the dust outside the instrument.

b. When cleaning the LCD screen, please pay attention and protect the transparent LCD screen.

c. When cleaning the dust screen, use a screwdriver to remove the screws of the dust cover and then remove the dust screen. After cleaning, install the dust screen in sequence.

d. Before cleaning, disconnect the power supply. Wipe the instrument with a slightly damp (not dripping) soft cloth. Do not use abrasive materials or chemical cleaning agents on the instrument or probes.

**Warning: Please confirm that the instrument is completely dry before use, to avoid electrical shorts or even personal injury caused by moisture.**

## 33.2 Appendix B Built-in Arbitrary Wave Table

| Type | Name | Description |
|---|---|---|
| Common function (15 types) | AbsSine | Absolute sine wave |
| | AbsSineHalf | Absolute half-sine wave |
| | AmpALT | Amplify sine wave |
| | AttALT | Attenuated sine waveform |
| | Gaussian_monopulse | Gaussian monocycle |
| | GaussPulse | Gaussian pulse |
| | NegRamp | Falling ramp |
| | NPulse | N-Pulse signal |
| | PPulse | P-Pulse signal |

| | | |
|---|---|---|
| | SineTra | TraSine wave signal |
| | SineVer | VerSine wave signal |
| | StairUD | Stair up and down waveforms |
| | StairDn | Stair-down waveform |
| | StairUp | Stair-up waveform |
| | Trapezia | Trapezoid waveform |
| Engine (25 types) | BandLimited | Band-limited signal |
| | BlaseiWave | Time-velocity curve of explosive vibration |
| | Butterworth | Butterworth filter |
| | Chebyshev1 | Chebyshev 1 filter |
| | Chebyshev2 | Chebyshev 2 filter |
| | Combin | Combination function |
| | CPulse | C-Pulse signal |
| | CWPulse | CW pulse signal |
| | DampedOsc | Time-displacement curve of damped oscillation |
| | DualTone | Dual-tone signal |
| | Gamma | Gamma signal |
| | GateVibar | Gate self-oscillation signal |
| | LFMPulse | Linear frequency modulation pulse |
| | MCNoise | Mechanical construction noise |
| | Discharge | Discharge curve of Ni-MH battery |
| | Pahcur | Current waveform of DC brushless motor |
| | Quake | Quake wave |
| | Radar | Radar signal |
| | Ripple | Power ripple |
| | RoundHalf | Round half waveform |
| | RoundsPM | Rounds phase modulation waveform |
| | StepResp | Step-response signal |
| | SwingOsc | Kinetic energy- time curve of swing oscillation |
| | TV | TV signal |
| | Voice | Voice signal |
| Maths | Airy | Airy function |

| | | |
|---|---|---|
| (27 types) | Besselj | Bessel I function |
| | Besselk | Modified Bessel function of the second kind |
| | Bessely | Bessel II function |
| | Cauchy | Cauchy distribution |
| | Cubic | Cubic function |
| | Dirichlet | Dirichlet function |
| | Erf | Error function |
| | Erfc | Complementary error function |
| | ErfcInv | Inverted complementary error function |
| | ErfInv | Inverted error function |
| | ExpFall | Exponential fall function |
| | ExpRise | Exponential rise function |
| | Gammaln | Natural logarithm of gamma function |
| | Gauss | Gaussian distribution (Normal distribution) |
| | HaverSine | HaverSine sine |
| | Laguerre | 4-times Laguerre polynomial |
| | Laplace | Laplace distribution |
| | Legend | 5-times Legend polynomial |
| | Log10 | Logarithmic function with the base of 10 |
| | LogNormal | Logarithmic normal distribution |
| | Lorentz | Lorentzian function |
| | Maxwell | Maxwell distribution |
| | Rayleigh | Rayleigh distribution |
| | Versiera | Versiera |
| | Weibull | Weibull distribution |
| | ARB_X2 | Square function |
| SectMod (5 types) | AM | Sectioned sine amplitude modulation signal |
| | FM | Sectioned sine frequency modulation signal |
| | PFM | Sectioned pulse frequency modulation signal |
| | PM | Sectioned sine phase modulation |

| | | |
|---|---|---|
| | | signal |
| | PWM | Sectioned pulse width frequency modulation signal |
| Bioelect (6 types) | Cardiac | Cardiac signal |
| | EOG | Electro-Oculogram |
| | EEG | Electroencephalogram |
| | EMG | Electromyogram |
| | Pulseilogram | Pulsilogram |
| | ResSpeed | Speed curve of the respiration |
| Medical (4 types) | LFPulse | Waveform of the low frequency pulse electrotherapy |
| | Tens1 | Waveform 1 of the nerve stimulation electrotherapy |
| | Tens2 | Waveform 2 of the nerve stimulation electrotherapy |
| | Tens3 | Waveform 3 of the nerve stimulation electrotherapy |
| Automotive (17 types) | Ignition | Ignition waveform of the automotive motor |
| | ISO16750-2 SP | Automotive starting profile with ringing |
| | ISO16750-2 Starting1 | Automobile voltage waveform 1 due to starting |
| | ISO16750-2 Starting2 | Automobile voltage waveform 2 due to starting |
| | ISO16750-2 Starting3 | Automobile voltage waveform 3 due to starting |
| | ISO16750-2 Starting4 | Automobile voltage waveform 4 due to starting |
| | ISO16750-2 VR | Automotive supply voltage profile for resetting |
| | ISO7637-2 TP1 | Automotive transients due to disconnects |
| | ISO7637-2 TP2A | Automotive transients due to inductance in wiring |
| | ISO7637-2 TP2B | Automotive transients due ignition switching off |

| | | |
|---|---|---|
| | ISO7637-2 TP3A | Automotive transients due to switching |
| | ISO7637-2 TP3B | Automotive transients due to switching |
| | ISO7637-2 TP4 | Automotive supply profile during starting |
| | ISO7637-2 TP5A | Automotive transients due to battery disconnect |
| | ISO7637-2 TP5B | Automotive transients due to battery disconnect |
| | SCR | SCR sintering temperature distribution |
| | Surge | Surge signal |
| Trigonome (21 types) | CosH | Hyperbolic cosine |
| | CosInt | Integral cosine |
| | Cot | Cotangent function |
| | CotHCon | Concave hyperbolic cotangent |
| | CotHPro | Protuberant hyperbolic cotangent |
| | CscCon | Concave cosecant |
| | CscPro | Protuberant cosecant |
| | CotH | Hyperbolic cotangent |
| | CscHCon | Concave hyperbolic cosecant |
| | CscHPro | Protuberant hyperbolic cosecant |
| | RecipCon | Concave reciprocal |
| | RecipPro | Protuberant reciprocal |
| | SecCon | Concave secant |
| | SecPro | Protuberant secant |
| | SecH | Hyperbolic secant |
| | Sinc | Sinc function |
| | SinH | Hyperbolic sine |
| | SinInt | Integral sine |
| | Sqrt | Square root |
| | Tan | Tangent |
| | TanH | Hyperbolic tangent |
| AntiTrigonome (16 types) | ACosH | Arc hyperbolic cosine |
| | ACotCon | Concave arc cotangent |

| | | |
|---|---|---|
| | ACotPro | Protuberant arc cotangent |
| | ACotHCon | Concave arc hyperbolic cotangent |
| | ACotHPro | Protuberant arc hyperbolic cotangent |
| | ACscCon | Concave arc cosecant |
| | ACscPro | Protuberant arc cosecant |
| | ACscHCon | Concave arc hyperbolic cosecant |
| | ACscHPro | Protuberant arc hyperbolic cosecant |
| | ASecCon | Concave arc secant |
| | ASecPro | Protuberant arc secant |
| | ASecH | Arc hyperbolic secant |
| | ASin | Arc Sinc |
| | ASinH | Arc hyperbolic sine |
| | ATan | Arc tangent |
| | ATanH | Arc hyperbolic tangent |
| Noise (6 types) | NoiseBlue | Blue noise |
| | NoiseBrown | Brown noise (Red noise) |
| | NoiseGray | Gray noise |
| | NoisePink | Pink noise |
| | NoisePurple | Purple noise |
| | NoiseWhite | White noise |
| Window function (17 types) | Bartlett | Bartlett window |
| | BarthannWin | Modified Bartlett-Hann window |
| | Blackman | Blackman window |
| | BlackmanH | BlackmanH window |
| | BohmanWin | Bohman window |
| | Boxcar | Rectangle window |
| | ChebWin | Chebyshev window |
| | GaussWin | Gaussian window |
| | FlattopWin | Flat Top weighted window |
| | Hamming | Hamming window |
| | Hanning | Hanning window |
| | Kaiser | Kaiser window |
| | NuttallWin | Nuttall-defined minimum 4-term Blackman-Harris window |
| | ParzenWin | Parzen window |

| | | |
|---|---|---|
| | TaylorWin | Taylor window |
| | Triang | Triangle window (Fejer window) |
| | TukeyWin | Tukey (tapered cosine) window |
| Complex Wavelets (7 types) | Complex Frequency B-spline | Complex frequency B-spline function |
| | Complex Gaussian | Complex Gaussian function |
| | Complex Morlet | Complex Morlet wavelet |
| | Complex Shannon | Complex Shannon function |
| | Mexican hat | Mexican hat wavelet |
| | Meyer | Meyer wavelet |
| | Morlet | Morlet wavelet |
| Other (34 types) | ABA_1_1 | |
| | ABA_1_2 | |
| | ALT_03 | |
| | ALT_04 | |
| | ALT_05 | |
| | AUDIO | |
| | COIL_2_1 | |
| | COIL_2_2 | |
| | DC_04 | |
| | ECT_1_2 | |
| | EGR_2 | |
| | EGR_3_2 | |
| | EST_03_2 | |
| | IAC_1_1 | |
| | INJ_1_1 | |
| | INJ_2 | |
| | INJ_3 | |
| | INJ_4 | |
| | INJ_5_6 | |
| | INJ_7 | |
| | KS_1_1 | |
| | MAF_1_1 | |
| | MAF_1_2 | |
| | MAF_5_3 | |

| | | |
|---|---|---|
| | MAP_1_1 | |
| | MAP_1_2 | |
| | MC_3 | |
| | Mexican hat | Mexican hat wavelet |
| | O2PROPA1 | |
| | O2PROPA2 | |
| | O2SNAP | |
| | STAR02_1 | |
| | TPS_1_1 | |
| | TPS_1_2 | |

# 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.

 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/