UNI-T

Instruments.uni-trend.com



UPO6000HD Series High-Resolution Oscilloscopes User Manual

The manual applies to:

UPO6000HD Series

V1.0

June 2026

Foreword

Thank you for choosing this UNI-T instrument. For the 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 UPO6000HD 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. 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
ETL intertek 4007682	**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.
40	**EFUP**	This environment-friendly use period (EFUP) 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.

Warning

Any changes or modifications to this unit not expressly approved by the party responsible for compliance may void the user's authority to operate the equipment.

Note

This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference in a residential installation. This equipment generates uses and can radiate radio frequency energy and, if not installed and used in accordance with the instructions, may cause harmful interference to radio communications.

However, there is no guarantee that interference will not occur in a particular installation. If this equipment does cause harmful interference to the radio or television reception, which can be determined by turning the equipment off and on, the user is encouraged to try to correct the interference by one or more of the following measures:

- Reorient or relocate the receiving antenna.
- Increase the separation between the equipment and receiver.

- Connect the equipment into an outlet on a circuit different from that to which the receiver is connected.
- Consult the dealer or an experienced radio/TV technician for help.

Statement

This equipment complies with FCC radiation exposure limits set forth for an uncontrolled environment. This equipment should be installed and operated with a minimum distance of 20cm between the radiator & your body. This transmitter must not be co-located or operating in conjunction with any other antenna or transmitter.

Safety Requirements	
Warning	
Preparation before use	Please connect this device to AC power supply with the power cable provided. The AC input voltage of the line reaches the rated value of this device. See the product manual for specific rated value. The line voltage switch of this device matches the line voltage. The line voltage of the line fuse of this device is correct. 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.
Connect and disconnect properly	Do not connect or disconnect probes or test leads while they are connected to a voltage source. Use only insulated voltage probes, test leads, and adapters supplied with the product, or indicated by UNI-T to be suitable for 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. 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. Do not use this device in explosive, dusty, or high humidity conditions.
Do not operate in humid environment	Do not use this device in a humid environment to avoid the risk of internal short circuit or electric shock.
Do not operate in flammable and explosive environment	Do not use this device in a flammable and explosive environment to avoid product damage or personal injury.
Caution	
Abnormality	If this device may be faulty, please contact the authorized maintenance personnel of UNI-T for testing. Any maintenance, adjustment or parts replacement must be done by the relevant personnel of UNI-T.

Cooling	Do not block the ventilation holes at the side and back of this device. Do not allow any external objects to enter this device via ventilation holes. Please ensure adequate ventilation and leave a gap of at least 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 environments.

- 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.

Ventilation openings are located on the rear and side panels of the instrument. Ensure that these openings remain unobstructed to maintain proper airflow. To prevent dust accumulation from blocking the vents, clean the instrument enclosure regularly.

The enclosure is not waterproof. Disconnect the power supply before cleaning. Wipe the enclosure

with a dry or slightly damp soft cloth.

2.2. Connecting the Power Supply

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

Voltage Range	Frequency
100 V-240 VAC (Fluctuations ±10%)	50 Hz /60 Hz
100 V-120 VAC (Fluctuations ±10%)	400 Hz

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. UPO6000HD Series Overview

UPO6000HD series high-resolution oscilloscopes include the following models.

Model	Analog Channel	Analog Bandwidth
UPO6108HD	8	1GHz
UPO6058HD	8	500MHz
UPO6038HD	8	350MHz
UPO6106HD	6	1GHz
UPO6056HD	6	500MHz
UPO6036HD	6	350MHz

Note: This manual is based on the UPO6108HD

The UPO6000HD is a high-resolution oscilloscope that integrates eight signal channels. 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, 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 UPO6000HD 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, tap "Vertical Scale" to adjust the vertical scale.

c. Operation Step

Use an arrow ">" represent the next step. For example, Storage > Save represents that in the Storage Menu, 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.

f. Rotary Knob

Underlined text indicates a rotary knob. For example, vertical Position knob.

5. Getting Started Guide

- General Inspection
- Before Use
- Front Panel
- Rear Panel
- Operation Panel
- User Interface
- Touch Screen
- Parameter Setting
- Remote Control

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

5.1. General Inspection

It is recommended to inspect the instrument by following the steps below before using the UPO6000HD series oscilloscopes for the first time.

(1) Check for Damages caused by Transport

If the shipping carton or foam cushioning is severely damaged, contact your UNI-T distributor immediately before using the instrument.

(2) Check Accessories

The details of the supplied accessories are described in the UPO6000HD 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 the Power Supply

Refer to Connecting the Power Supply, use the assembled power line or other power line that meets the local country standards to connect the oscilloscope.

If the rear-panel power switch is OFF, the front-panel soft power button (lower left) will not light up, even when the instrument is connected to a power source, indicating that the soft power button is disabled.

When the rear-panel power switch is ON, the front-panel soft power button (lower left) lights red. Press the button to power on the oscilloscope.

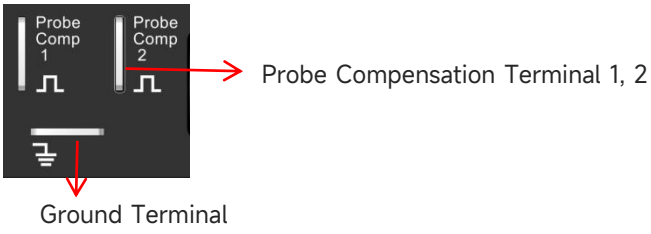
(2) Boot Check

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

(3) Connecting the Probe

This oscilloscope is supplied with two probes for compensation. Connect the probe BNC connector to the C1 input BNC connector on the oscilloscope. Connect the probe tip to the probe compensation terminal 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 Compensation Terminal and Ground Terminal

(4) Function Check

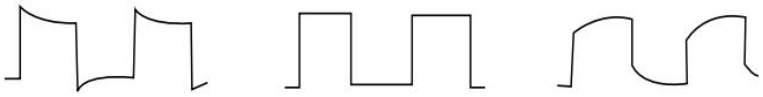
Press the Autoset button to display a square wave (amplitude 3 Vpp, frequency 1 kHz). Repeat Step 3 to check all channels.

(5) 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 press the Autoset button.
- 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	No.	Description
1	Display area	6	Vertical Control
2	Autoset & Run/Stop	7	Probe Compensation Signal Terminal & Ground Terminal ①
3	Trigger Control	8	Analog Channel Input Terminal ②
4	Function Module	9	Microphone
5	Horizontal Control	10	Soft Power Switch

①: Probe Compensation Signal Terminal & Ground Terminal: Connect the probe BNC connector to the C1 input BNC connector on the oscilloscope. Connect the probe tip to the probe compensation terminal and connect the probe ground alligator clip to the corresponding ground terminal to output the instrument signal. For more details, refer to the section Before Use - Connecting the Probe.

②: Analog Channel Input Terminal: Connect the oscilloscope probe or BNC cable to these BNC connectors to input a signal into the oscilloscope.

5.4. Rear Panel



Rear Panel

1. EXT Trig: The input terminal of external trigger
2. AUX OUT: Output terminal for trigger output, Pass/Fail output, DVM output
3. 10MHz REF: 10 MHz REF IN&OUT, BNC. Use this port to import the external reference clock signal or export 10 MHz clock signal generated by the internal crystal oscillator of the instrument.
4. USB HOST: Supports USB device
5. HDMI: Supports to connect an external displayer with HDMI port
6. USB Device: USB Device for communication between the oscilloscope and a PC
7. LAN: Connects to LAN for remote control
8. Power switch: After the AC power socket is properly connected, turn on the power switch. Then, press the soft power button on the front panel to start the oscilloscope.

9. AC power input socket: Use the assembled power cable to connect the oscilloscope to AC power. For more details, refer to the section Before Use - Connecting the Power Supply.
10. Safety lock: Lock the oscilloscope at fixed position (sold separately).

5.5. Operation Panel

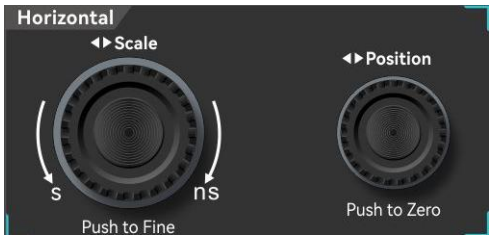
(1) Vertical Control

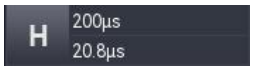


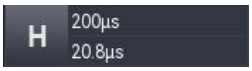
- Math: Press this button open the mathematical operation menu to perform math operation (add, subtract, multiply, divide), digital filter, and advanced operation. For more details, refer to the section Mathematical Operation.
- Ref: Load a reference waveform from Local or USB for comparison with the measured waveform. For more details, refer to the section Reference Waveform.
- FFT: Press this button to open fft menu .
- Bus: Press this button to access Bus menu for configuring these protocols: RS232, I^2C, SPI, CAN, CAN-FD, LIN, FlexRay, Audio, 1553B, Manchester, SENT, and ARINC429. For more details, refer to the section Protocol Decoding.
- 1, 2, 3, 4, 5, 6, 7, 8:Analog channel setting buttons correspond to C1, C2, C3, C4, C5, C6, C7, C8. The eight channel tabs are color-coded to match the waveform colors on the display and the channel input connectors. Press any buttons to enter the corresponding channel menu (activate or disable the channel). For more details, refer to the section Vertical Channel.
- Scale: Vertical scale rotary knob adjusts the vertical scale of the selected channel. Turn the knob clockwise to decrease the scale and counterclockwise to increase the scale. As the vertical scale is adjusted, the waveform amplitude changes accordingly. The scale value 

at the bottom left of the display updates in real time. The vertical scale is step with 1-2-5, press this rotary knob to switch between coarse and fine adjustment of the vertical scale.
- Position: Vertical position rotary knob adjusts the vertical position of the selected channel. Press this rotary knob to reset the channel position to the vertical center.

(2) Horizontal Control



- Scale: Horizontal time base rotary knob adjusts the time base for all channels. As the time base is adjusted, the waveform is compressed or expanded in the horizontal direction. The time base value 

of the display updates in real time. The time base is step with 1-2-5, press this rotary knob to switch between coarse and fine adjustment of the time base.
- Position: Horizontal position rotary knob adjusts the trigger point to the left or right. As the position is adjusted, the waveforms of all channels move left or right. The horizontal position value 

of the display updates in real time. Press this rotary knob to reset the channel position to the horizontal center.

(3) Trigger Control



- Single: Press the Single button to activate the single trigger mode. The button lights up.
- Normal: Press the Normal button to activate the normal mode. The button lights up.
- Auto: Press the Auto button to activate the automatic mode. The button lights up.
- Slope: Press the Slope button to switch the trigger edge to rising, falling, or either edge. The corresponding button lights up.
- Position: Press the trigger Position rotary knob to adjust the trigger level. Turn clockwise to increase the level and counterclockwise to decrease it. As the trigger level is adjusted, the trigger level value in the top-right corner 

of the display updates in real-time.

  When in single mode, press the knob to quickly reset the trigger level to 50%.
- When the trigger mode is READY or TRIGED, the corresponding READY or Trig'd indicator lights up.

(4) Run/Stop, Auto Control



Run/Stop: Press the Run/Stop button to switch between Run or Stop modes.

Run: In Run mode, the button lights up green.

Stop: In Stop mode, the button lights up red.



Autoset: Press the Autoset button, the oscilloscope automatically adjusts the vertical scale, time base, and trigger mode based on the input signal to display the optimal waveform.

(5) Function Module



- Cursors: Press the Cursors button to access the cursor measurement menu for configuring time, voltage, or screen measurements for each source. For more details, refer to the section Cursor Measurement.
- Meas: Press the Meas button to access the measurement menu for configuring counter, voltmeter, parameter snapshot, measurement statistics, add measurement, clear measurement, and global settings. For more details, refer to the section Automatic Measurement.
- Quick Meas: Press the Quick Meas button to quickly open the parameter measurement. The parameter measurement information bar will be displayed.
- Analyze: Press the Analyze button to open the analysis menu and access functions such as voltmeter, counter, power analysis, waveform recording, and pass/fail testing.
- : Press the Home button to open the home menu directly.
- : Press the User button to activate the user-defined function, refer to the section Utility Function - Quick Operation.
- : Press the Shortcut button to quickly save picture, waveform file, and setup file. For the format and storage path, refer to the section Utility Function - Quick Operation.
- : Press the Clear button to clear all recalled waveforms, and measurement statistics.
-  Arrow keys (left/right): Use the arrow keys to select digits when adjusting values with multifunction knobs, allowing digit-by-digit navigation.
- DVM: Press the DVM button to open the digital multimeter menu directly.
- Counter: Press the Counter button to open the counter menu directly.
- Touch/Lock: Press the Touch/Lock button to enable/disable the touch screen function. Press the Touch/Lock button once to disable the touch screen function, and the indicator lights up. Press the button again to enable the function, and the indicator lights off.
- Default: Press the Default button to reset all oscilloscope settings to their factory default

values.

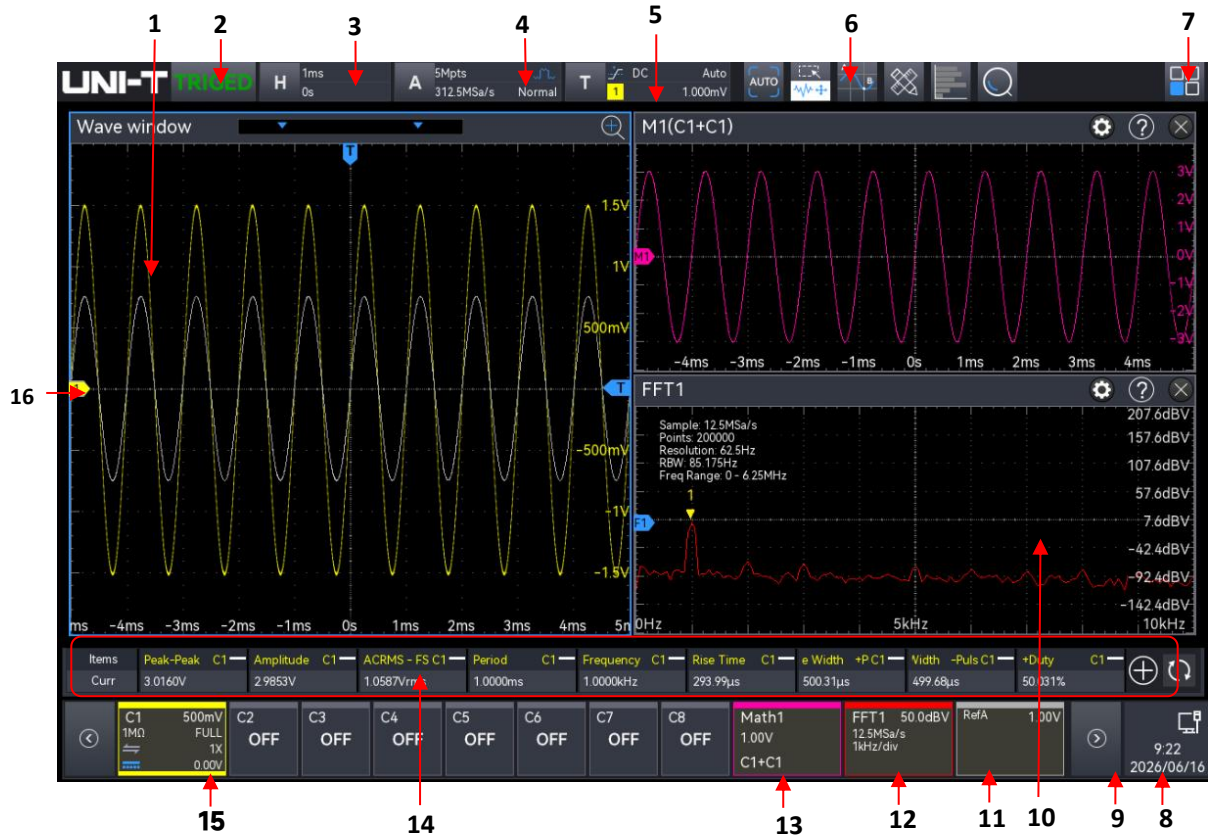
- Multipurpose: The multipurpose rotary knob becomes active when the Math menu is selected. The knob lights up, indicating that it can be used to adjust numeric values.

(6) Home Menu

Press the Home icon in the top-right of the screen to open the Home quick menu, including: measurement, cursors, voltmeter, counter, math, reference, regional histogram, search, decoding, navigation, recording, marker, FFT, power analysis, pass/fail test, storage, display, utility, help, area draw, auto, default, and single. The quick menu provides direct access to the corresponding function menus.



5.6. User Interface



1. Display: Displays C1 - C8 waveform measurement windows, reference waveforms, and math waveforms.
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 the 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. A maximum of 9 icons can be displayed.
7. Home menu: Opens the function guide menu. Select a function to access the corresponding menu.
8. Notification: Displays the USB, LAN connection icon, and time. Click this area to open the setting menu. Refer to the Notification section for more details.
   - USB: When a USB device is connected, a USB icon is displayed in this area.
   - LAN: When the LAN connection is established, a LAN icon is displayed in this area.

- Time: Displays the current time and allows setting the system time.

9. Volts/div signal bar: When multiple Volts/div information boxes are shown at the bottom of the screen, press the arrow keys ,  to scroll and reveal hidden boxes.
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 Ref1 - Ref4 status and vertical scale. A maximum of 4 Ref labels can be displayed.
12. FFT label: Displays the FFT1 - FFT4 status, vertical scale, and frequency per division. A maximum of 4 FFT labels can be displayed.
13. Math label: Displays M1 - M8 status, vertical scale, and operation type. A maximum of 4 Math labels can be displayed.
14. Measured result window: Displays counter, DVM measurements, parameter measurements, and statistical results. Tap to open/close the measurement window in the Measure module.
15. Channel label: Displays C1 - C8 status, vertical scale, impedance, bandwidth limit, inverse phase, probe attenuation ratio, and vertical offset.
16. Analog channel icon: Displays C1 - C8 icons. The channel icon matches the waveform color.

5.7. Touch Screen

UPO6000HD series features 12.1-inch high-sensitivity capacitive touch screen with multiple point touch control and gesture control. It is equipped with an intuitive operating system and a responsive touchscreen, delivering clear waveform display and a smooth user experience.

Touch control functions include tap, pinch, drag, and rectangle drawing.

Note: All on-screen menus support touch screen operation.

(1) Tap

Use one finger to slightly tap an icon or text on the screen to operate it, as shown in the figure below.

Tap gestures can be used to:

- Tap a menu on the screen to enter its settings.
- Tap the function icon in the top-right of the screen to open the corresponding function.
- Tap the pop-up numeric keypad to set the parameter.
- Tap the virtual keyboard to enter label names and file names.
- Tap a message window, then tap the close icon ⊗ in the top-right of the screen to close the pop-up window.

- Tap another window displayed on the screen, then proceed with the setup.
- Tap a message window, then tap the help icon ⑦ in the top-right of the screen to open the corresponding function help menu.



Tap Gestures

(2) Pinch

Place two fingers on the screen and move them together or apart. Pinch gestures can zoom in or zoom out the waveform. To zoom out, pinch inward (move two fingers together), then spread them apart; to zoom in, spread two fingers apart, then pinch them together, as shown in the figure below.

Pinch gestures can be used to:

- Adjust the horizontal time base of the waveform by pinching in the horizontal direction.
- Adjust the vertical scale of the waveform by pinching in the vertical direction.



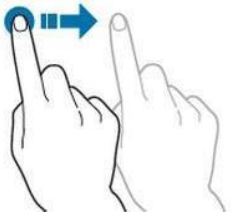
Pinch Gestures

(3) Drag

Use one finger to press and drag a selected item to the desired position, as shown in the figure below.

Drag gestures can be used to:

- Drag a waveform to adjust its position.
- Drag a window to change its position.
- Drag a cursor to change its position.



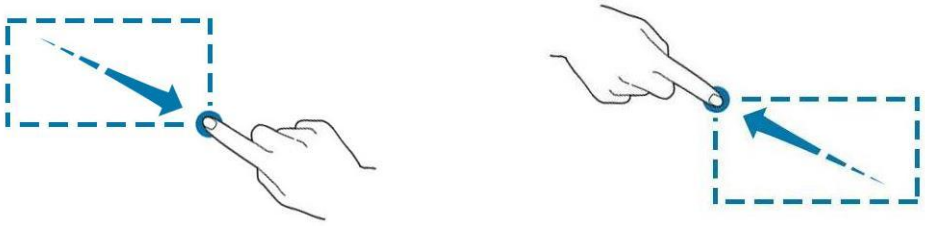
Drag Gestures

(4) Rectangle Drawing

Open the Home menu and tap the Rectangle Drawing icon to enable the function. Drag your finger on the screen to draw a rectangle, as shown in Figures (a) and (b). When the rectangle

is drawn, a menu will appear. At this point, you can select Area A, Area B, Intersection, or Non-intersect.

To define a trigger area, drag your finger from the bottom right to the top left of the screen.



(a) (b)

Rectangle Drawing Gestures

Area A

- Draw Area A trigger zone
- Enable Area A
- Open the Area A menu

Area B

- Draw Area B trigger zone
- Enable Area B
- Open the Area B menu

Note: Tap Rectangle Drawing to switch between rectangle drawing mode and waveform operation mode. When the rectangle drawing icon shows, rectangle drawing mode is enabled; when the waveform operation icon , waveform operation mode is enabled. The default mode is waveform operation.

5.8. Parameter Setting

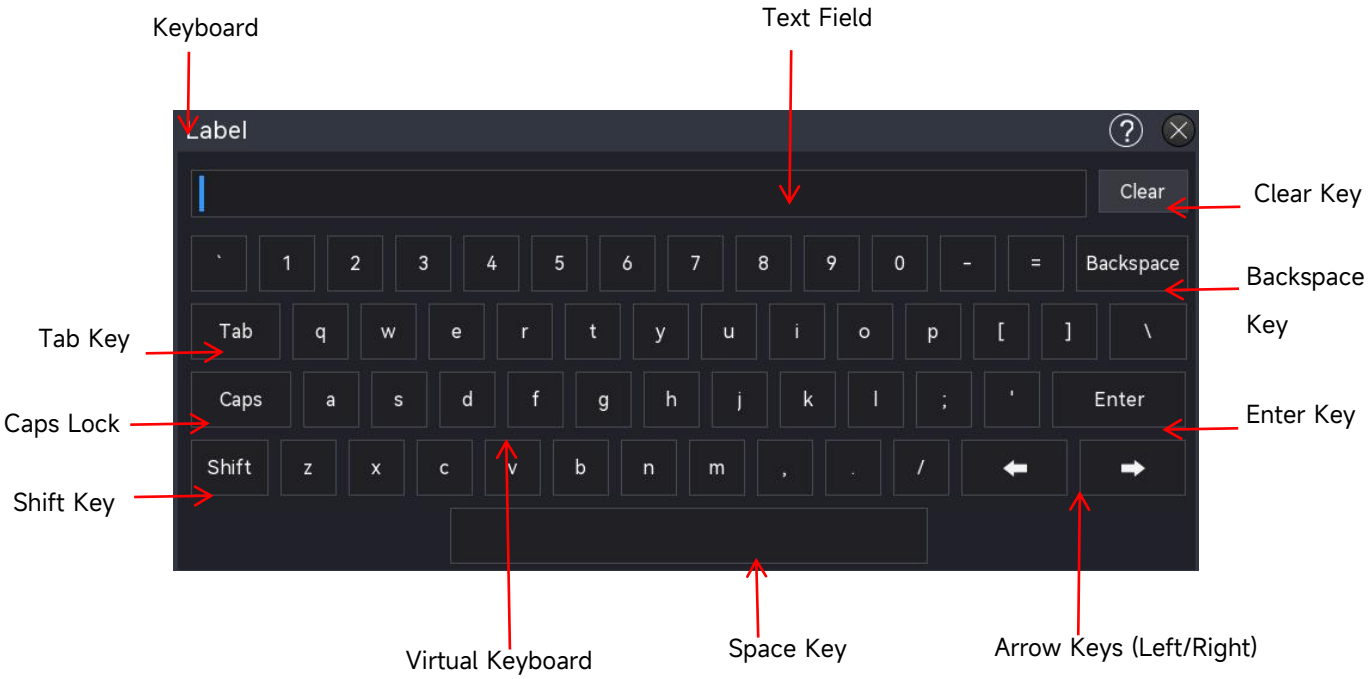
UPO6000HD series supports two setup methods for parameter configuration: multifunction rotary knob Multipurpose and touch screen. The setting steps are as follows.

(1) Multipurpose Rotary Knob

For the parameter of time and voltage, once the parameter is selected, rotate the Multipurpose A knob on the front panel to enter the parameter value.

(2) Touch Screen

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



① Character String

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

a. Input Field

Enter text: input letters, numbers, and special characters.

b. Clear Key

Delete all contents in the input field.

a. Caps Key

Switch between upper and lower case.

b. Tab Key

Insert two spaces at a time.

c. Shift Key

Switch among numbers, special characters, and upper/lower case letters.

d. Arrow Keys (Left/Right)

To edit content, press the ←, → key to move the cursor to left or right, and then make the desired changes.

e. Space Key

Insert one space in the input field.

f. Backspace Key

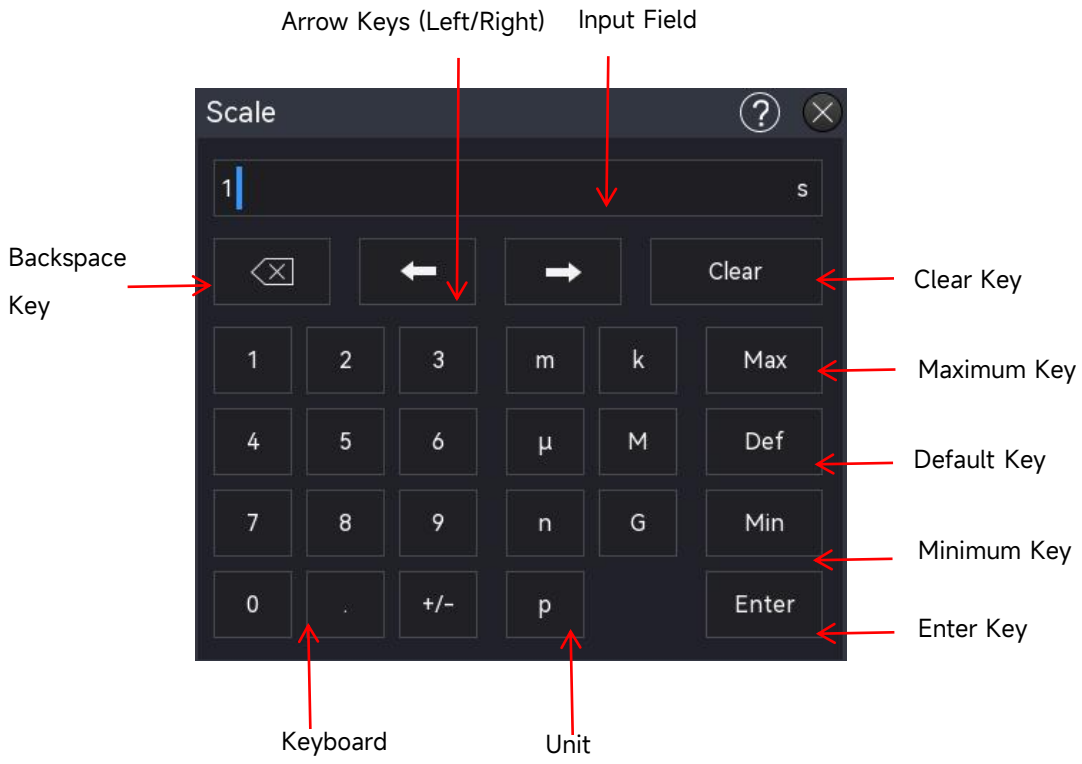
Delete a single character. This key is useful for deleting characters in long input fields.

g. Enter Key

Confirm the setting and close the virtual keyboard.

② Numeric Value

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



a. Entering Values and Units

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.

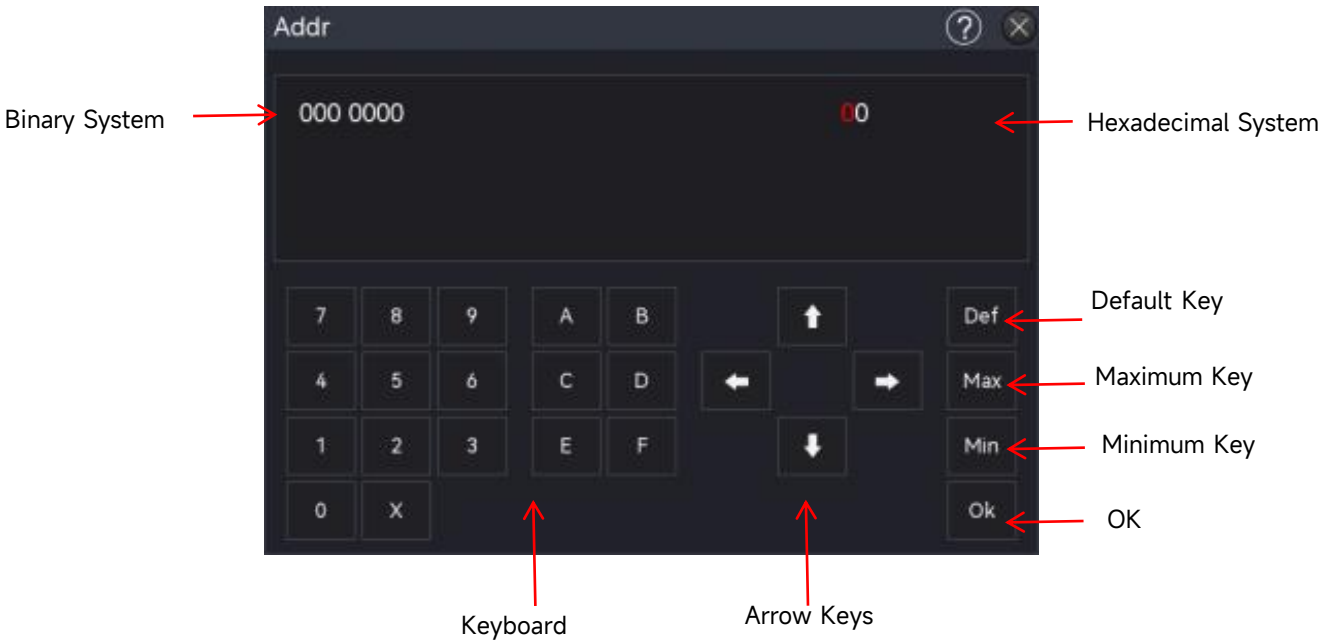
Available operations on the numeric keypad:

- Delete the entered parameter value.
- Set the parameter to the maximum or minimum value (sometimes specifically the maximum or minimum value for the current state).
- Set the parameter to its default value.
- Clear the parameter input field.
- Move the cursor to modify the parameter value.

③ Binary and Hexadecimal System Values

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

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



a. After entering all the values and pressing the Ok button, the numeric keypad will automatically close, completing the parameter setting.

Available operations on the numeric keypad:

- Move the cursor to modify the parameter value.
- Set the parameter to the maximum or minimum value (sometimes specifically for the current state).
- Set the parameter to its default value.
- Clear the parameter input field.
- Delete the entered parameter value.

5.9. Remote Control

UPO6000HD 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).

UPO6000HD 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 *Programming Manual*.

(2) PC Software Control (Instrument Manager)

The user can use 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.

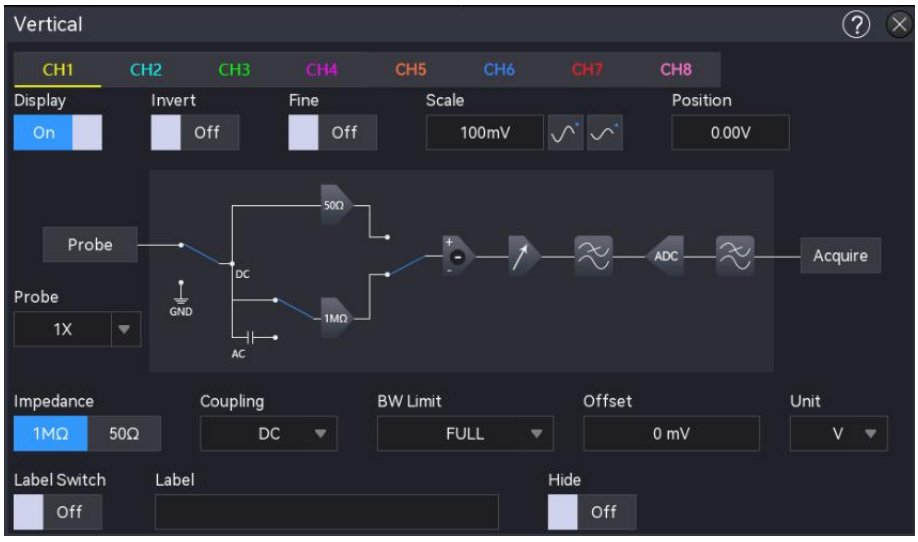
6. Vertical Control System

- Vertical Scale
- Open/Activate/Close Analog Channel
- Offset
- Position
- Channel Coupling
- Bandwidth Limit
- Probe
- Inverse Phase
- Impedance
- Unit
- Channel Fold
- Label

UPO6000HD 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.

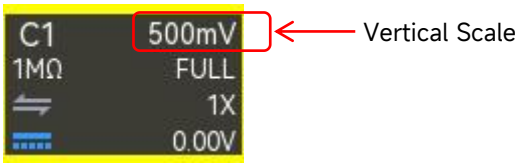
Access the Channel Setting menu using the following method.

- 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. As the vertical scale is adjusted, the waveform amplitude changes accordingly. The scale value at the bottom left of the display updates in real time.



The vertical scale range depends on the currently selected probe and input impedance.

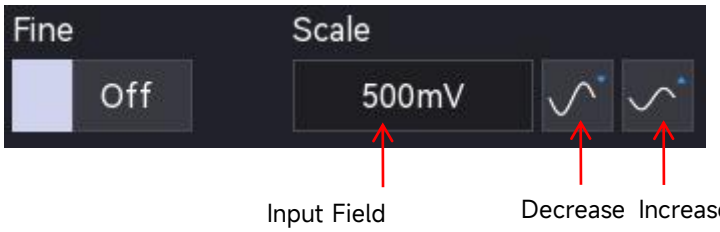
Default probe attenuation ratio: 1X

Input Impedance 1 MΩ: Vertical scale range is from 500 μV/div to 10 V/div.

Input Impedance 50 Ω: Vertical scale range is from 500 μV/div to 1 V/div.

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

- Use the vertical Scale rotary knob on the front panel to set the vertical scale.
  Clockwise: Turn clockwise to decrease the vertical scale.
  Counterclockwise: Turn counterclockwise to increase the vertical scale.
- Tap the screen and use a pinch gesture to adjust the vertical scale. For more details, refer to the section Touch Screen - Pinch.
- Open the Channel Setting menu, select C1, and double-click the Vertical Scale menu to open the numeric keypad and enter the vertical scale. Enter the desired vertical scale value using the Multipurpose knob. Alternatively, tap the icons , to change it.



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 setting is coarse tuning.

Use the vertical Scale rotary knob on the front panel to switch the coarse tuning and fine 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%.

Note: The fine tuning automatically enabled when using the touch screen set the vertical scale.

6.2. Open/Activate/Close Analog Channel

C1 - C8 analog channels have three states: open, close, and activated.

(1) Open Analog Channel

- When the channel is disabled, press the channel button [1] on the front panel to turn on C1. The button lights up.
- Tap the channel label at the bottom of the screen to turn on C1.
- In the Channel Setting menu, select C1 and set it to ON to turn on C1, or set it to OFF to turn it off.

(2) Close Analog Channel

- When C1 is opened and in the activated state, press the channel button [1] on the front panel or 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 press the channel button [1] on the front panel or double-click channel label at the bottom of the screen to turn off C1.
- Open the Channel Setting menu and select C1, set it to OFF to turn off C1.

(3) Activate Analog Channel

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

- Press the channel button [1] on the front panel to activate C1.
- Tap the channel label at the bottom of the screen to activate C1.
- In the Channel Setting menu, select C1 and activate it.



Activated State



Open but not Activated



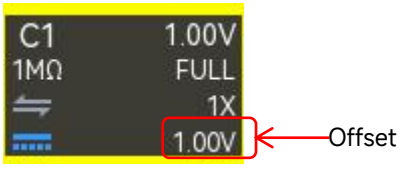
OFF State

6.3. Offset

During measurements with an oscilloscope, small offsets in a channel's zero voltage may occur due to factors such as component temperature drift or external environmental interference. These offsets can affect vertical parameter measurements. UPO6000HD series allows users to apply a zero offset voltage to calibrate the channel's zero point, thereby improving measurement accuracy. Adjusting the vertical offset through the channel label at the bottom of the screen updates the

display in real-time, as shown in the following figure. The vertical offset range is dependent on the current input impedance, probe attenuation ratio, and vertical scale settings.

Scale	Offset Range
500 μV/div ~ 5 mV/div	±2V (50Ω or 1MΩ)
5.1 mV/div ~ 1 V/div	±5V (50Ω)
5.1 mV/div ~ 100 mV/div	±5V (1MΩ)
102 mV/div ~ 1 V/div	±50V (1MΩ)
1.02 V/div ~ 10 V/div	±200V (1MΩ)



When C1 is activated, the vertical offset can be set using the following methods.

- In the Channel Setting menu, select C1, 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 5.8 Parameter Setting.
- Select Offset input field, use the arrow keys ←, → to position the cursor, then use the Multipurpose to adjust the offset.

6.4. Position

Vertical position refers to the offset of the channel signal's zero point in the vertical direction relative to the center of the screen. Adjusting the vertical position will move the waveform of the corresponding channel up or down.



When C1 is activated, the vertical position can be set using the following methods.

- Use the vertical Position rotary knob on the front panel to set the vertical position.

  Clockwise: Turn the knob clockwise to increase the vertical position.

  Counterclockwise: Turn the knob counterclockwise to decrease the vertical position.

  Reset: Press the rotary knob to quickly reset the position to zero.
- Tap the screen and use a drag gesture to adjust the vertical position. For more details, refer to the section Touch Screen - Drag.

- Open the Vertical menu, double-click Position input field to open the numeric keypad and directly enter the position value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.
- Select Position input field, use the arrow keys ←, → to position the cursor, then use the Multipurpose to adjust the position value.

  Clockwise: Turn the knob clockwise to increase the vertical position.

  Counterclockwise: Turn the knob counterclockwise to decrease the vertical position.

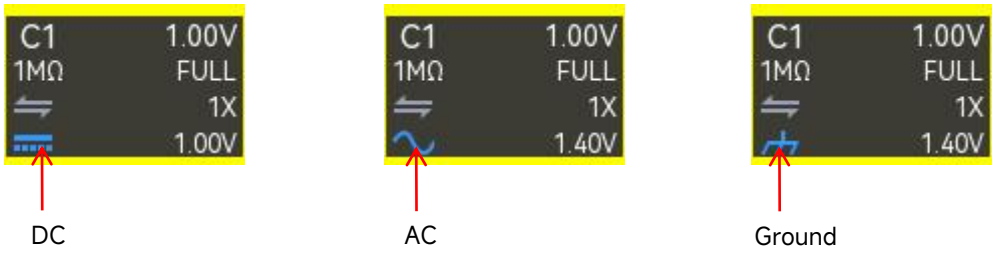
6.5. Channel Coupling

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

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

- DC: When the coupling mode is set to DC, both the DC and AC components of the measured signal will pass through.
- AC: When the coupling mode is set to AC, the DC component of the measured signal will be blocked, allowing only the AC component to pass.
- Ground: When the coupling mode is set to Ground, both the DC and AC components of the measured signal will be blocked, effectively grounding the input.

Once the channel coupling mode is set, the selected mode will be displayed 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 the channel label at the bottom of the screen to open the Channel Setting menu, then tap Bandwidth Limit to select the bandwidth limit value. When the bandwidth limit is enabled, the selected bandwidth limit value will be displayed in the channel label at the bottom of the screen, as

shown in the following figure.



UPO6000HD series supports the following bandwidth limit settings: 20 MHz and FULL.

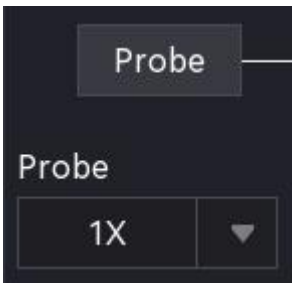
- 20 MHz: High-frequency components of the measured signal above 20 MHz are attenuated.
- FULL: The full bandwidth of the measured signal is passed without limitation, including high-frequency components.

6.7. Probe

(1) Probe Attenuation Ratio

To match the probe attenuation coefficient, set the probe attenuation ratio in the Channel Settings menu. For example, when using a probe with a 10:1 attenuation ratio, the probe attenuates the measured signal by a factor of 10 before it reaches the oscilloscope. Therefore, the probe setting in the oscilloscope should be set to ×10, so that the displayed value is correctly compensated and reflects the actual input voltage.

Click the channel tab at the bottom of the screen to open the Channel Settings menu, then tap the Probe Attenuation Ratio drop-down list to select the appropriate attenuation ratio.



Probe attenuation ratio: 0.001X, 0.01X, 0.1X, 1X, 10X, 100X, 1000X, and Custom.

- When the channel unit is set to **A**, it indicates a current probe. In this case, the probe attenuation ratio can be set to 5 mV/A, 10 mV/A, 50 mV/A, 100 mV/A, 200 mV/A, 500 mV/A, 1 V/A, or a custom value.
- Custom: Use the numeric keypad to set the probe attenuation ratio.

(2) Probe Setting

When the oscilloscope detects a probe with a probe attenuation ratio identification pin (where different resistors represent different attenuation ratios), it automatically identifies the probe attenuation ratio, sets the corresponding probe attenuation ratio, and displays information such as the manufacturer, probe model, serial number, and probe magnification in the probe settings menu.

Click Probe Setting to enter the Probe Setting menu. In this menu, the probe indicator and probe key function can be configured, and information such as manufacturer, probe model, serial number, and probe magnification can be viewed.

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

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

- Run/Stop: Pressing the active probe key switches the oscilloscope between Run and Stop states.
- Single: Pressing the active probe key switches the oscilloscope trigger mode to Single.
- Screenshot: Pressing the active probe key triggers an automatic screenshot capture.

6.8. Inverse Phase

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



- OFF: When the inverse phase is disabled, the waveform is displayed normally.
- ON: When the inverse phase is enabled, the waveform polarity is inverted. The results of mathematical operations and waveform measurements are also updated accordingly, as shown in the following figure.



Inverse Phase: OFF Inverse Phase: ON

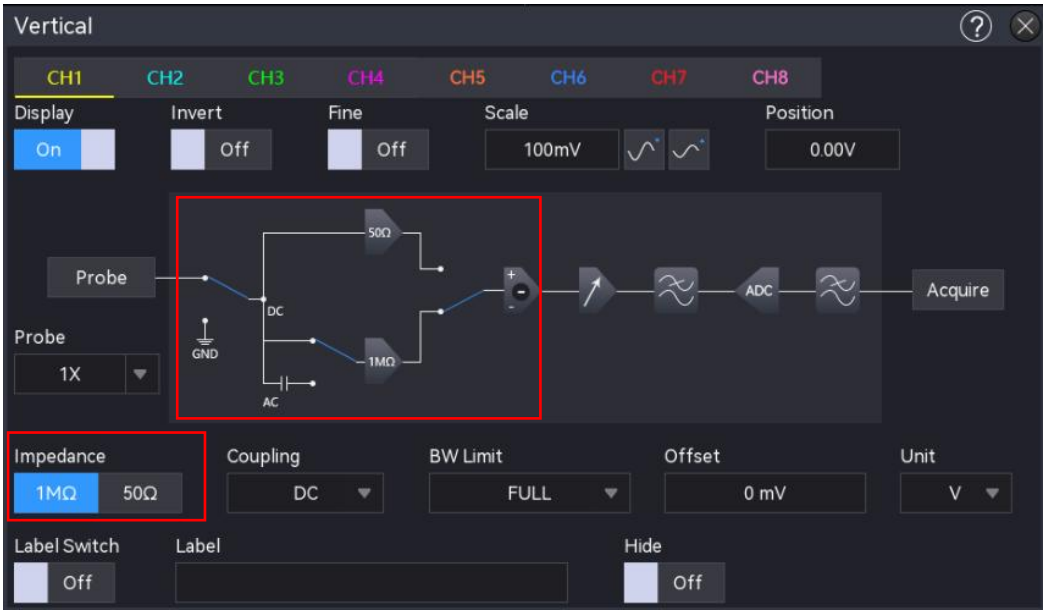
6.9. Impedance

To reduce the electric load cause by the interaction of the oscilloscope and the circuit to be

measured, this oscilloscope provides two input impedance mode 1 MΩ (default) and 50 Ω. In the Channel Setting menu, click the Impedance to select the input impedance.

- 1 MΩ: At this setting, the oscilloscope input impedance is very high, so the current drawn from the measured circuit is negligible.
- 50 Ω: Matches the oscilloscope input to a device with a 50 Ω output impedance.

The circuit diagram in the Channel Setting menu changes according to the selected input impedance, as shown in the following figure.



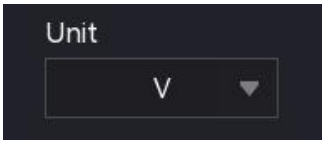
The input impedance setting affects the range of the channel's vertical scale and vertical offset.

Note: After the oscilloscope automatically recognizes the probe, the input impedance is automatically set to the appropriate mode, and no manual configuration is required.

6.10. Unit

Click the channel tab at the bottom of the screen to open the Channel Setting menu, then tap the 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 update accordingly.



6.11. Channel Fold

Click Channel Fold to set the channel's waveform state to ON (hide) or OFF (displayed).

- ON: When the Channel Fold is enabled, the channel waveform is hidden from display.
- OFF: When the Channel Fold is disabled, the channel waveform is displayed normally.

6.12. Label

The instrument uses the channel number as the default channel identifier. However, users can assign a custom name to each channel according to their preferences (e.g., CH1).

Click the channel tab at the bottom of the screen to open the Channel Setting menu, then tap Label to select ON (displayed) or OFF (hide) the channel label. The channel label can also be set by double-clicking the Label input field to open the virtual keyboard to directly enter the character string.

For details on using the numeric keypad, refer to section 5.8 Parameter Setting-Character String.

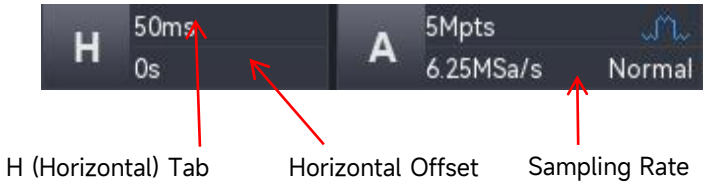


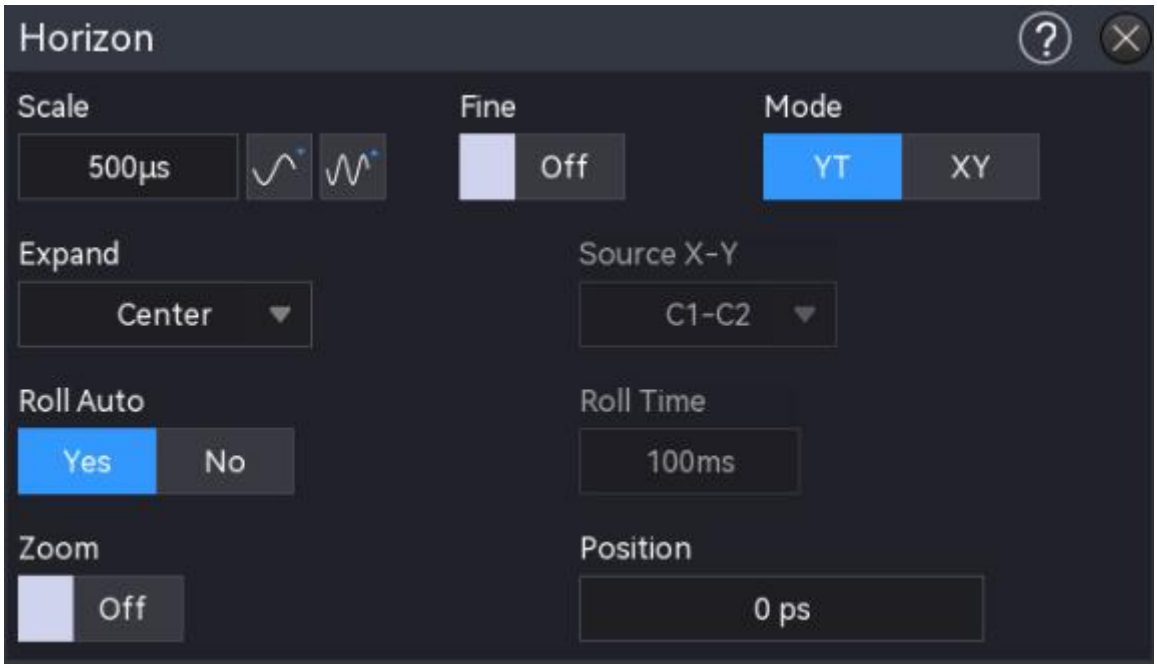
7. Horizontal Control System

- Horizontal Scale
- Horizontal Extension
- Auto Roll Mode
- Fastest Roll Mode
- Window Expansion
- XY
- Horizontal Position

Access the Horizontal Control System menu using the following method.

- Tap the H (Horizontal) tab on the top to enter the horizontal menu.





7.1. Horizontal Scale

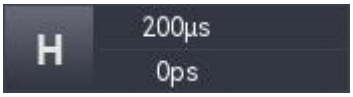
The horizontal scale, also known as the horizontal time base, represents the time value per division

in the horizontal direction of the screen, typically expressed in s/div. The available range of horizontal scales is shown in the table below.

When adjusting the horizontal time base, it changes in a 1–2–5 step sequence, e.g., 200 ps/div, 500 ps/div, 1 ns/div, 2 ns/div, 5 ns/div…500 s/div, 1 ks/div.

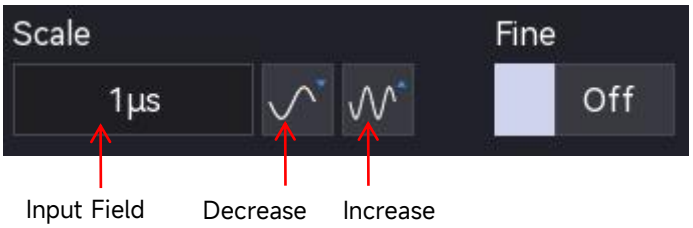
Model	Range
1GHz	500 ps/div - 1 ks/div
500MHz	500 ps/div - 1 ks/div
350MHz	1 ns/div - 1 ks/div

When the horizontal time base is adjusted, the waveforms of all channels are horizontally expanded or compressed relative to the currently selected horizontal expansion reference (see *Horizontal Expansion*). The horizontal time base displayed in the upper-left corner of the screen updates in real time, as shown in the following figure.



Adjust the horizontal time base using the following methods.

- Use the horizontal Scale rotary knob on the front panel to set the horizontal scale.
  Clockwise: Turn clockwise to decrease the horizontal scale.
  Counterclockwise: Turn counterclockwise to increase the horizontal scale.
- Tap the screen and use a pinch gesture to adjust the horizontal scale. For more details, refer to the section Touch Screen - Pinch.
- Tap the H (Horizontal) tab on the top to enter the horizontal menu, then double-click the Horizontal Scale field to open the numeric keypad and enter the horizontal scale value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. The value can also be adjusted using the Multipurpose knob. Alternatively, tap the icons , to change the value.



UPO6000HD series supports horizontal fine tuning. Use the horizontal Scale rotary knob on the front panel to switch between coarse tuning or fine tuning.

- Coarse Tuning: Tap the icons , to change the value. The horizontal scale is adjusted within the allowable range using the 1–2–5 step sequence.

- Fine Tuning: Tap the icons , to change the value. In fine adjustment mode, the horizontal scale can be adjusted more precisely, allowing finer control of all channel waveforms within the allowable range using the 1–2–5 step sequence.

7.2. Horizontal Extension

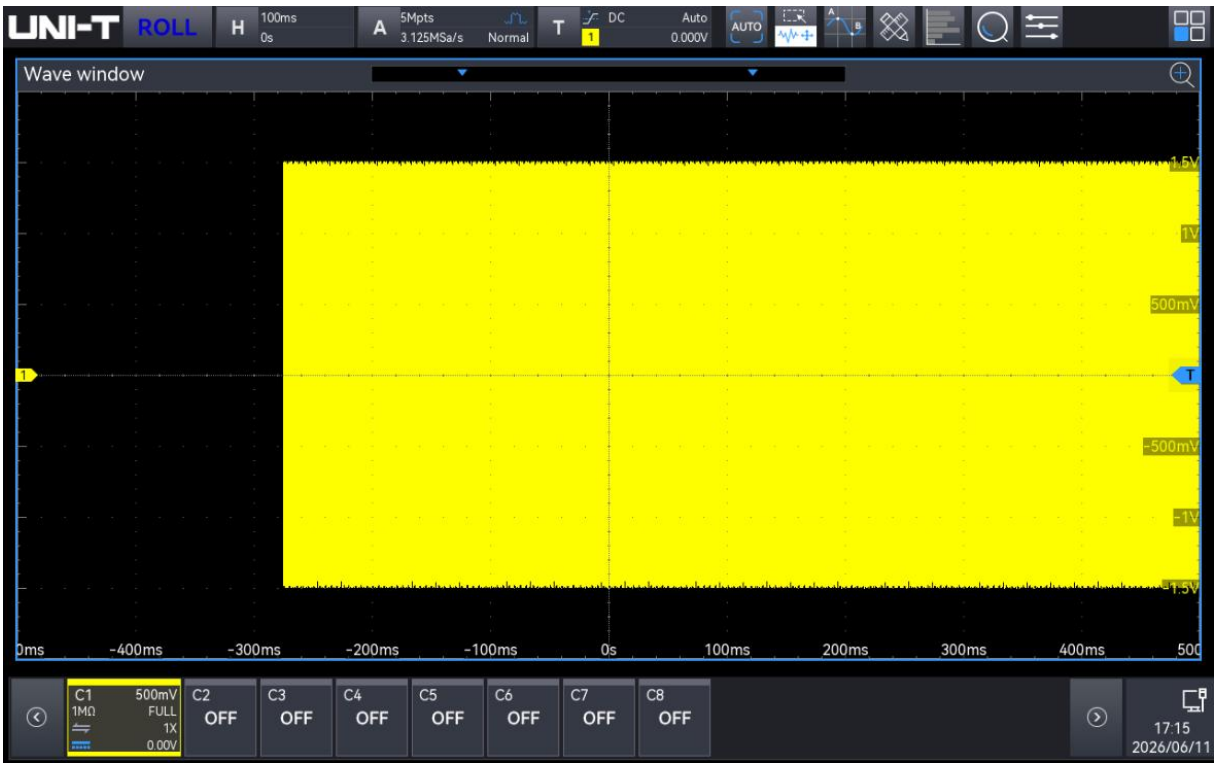
The horizontal extension defines the reference position used for horizontal expansion or compression when adjusting the horizontal time base. In the Horizontal menu, select Horizontal Extension to set the reference position to Center, Left, Right, or Trigger Point. The default setting is Center.

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

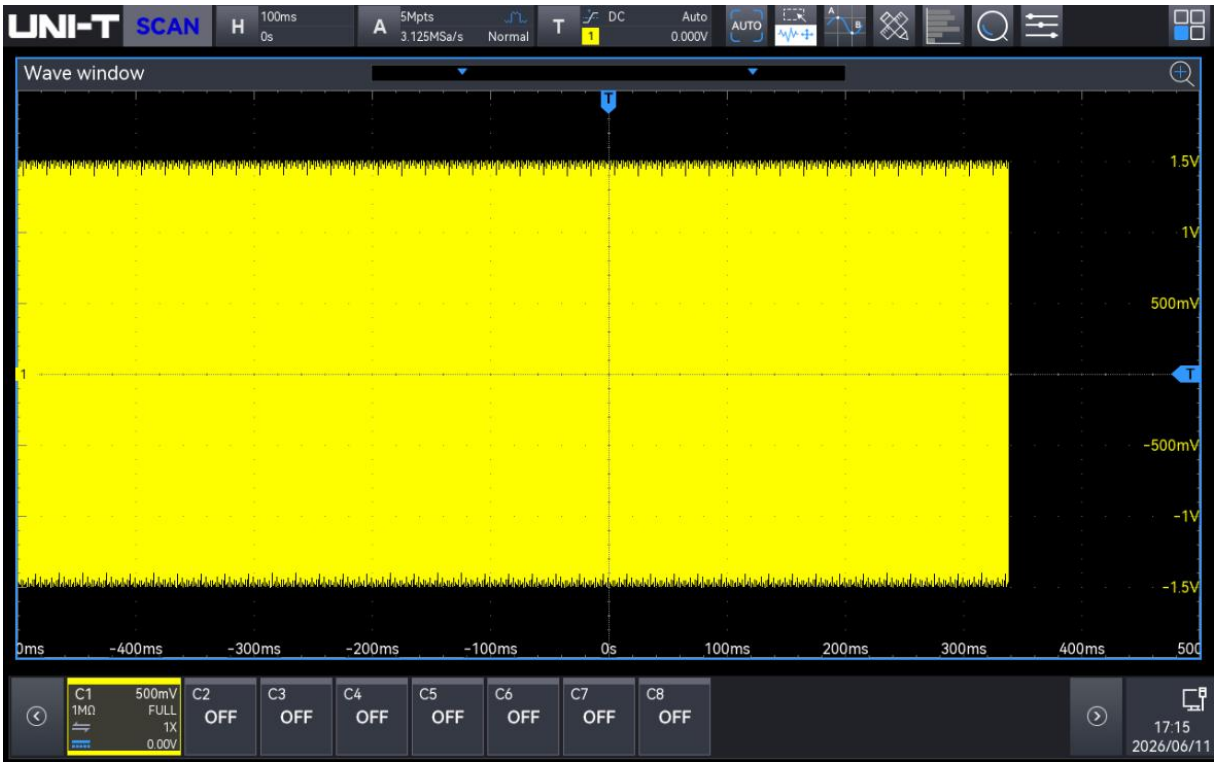
7.3. Auto Roll Mode

Click the Auto Roll Mode menu to set whether the oscilloscope enters SCAN or ROLL mode when the current time base is slower than the fastest roll time base. The setting can be switched between Yes and No.

- Yes: When the time base is slower than the fastest roll time base, the oscilloscope enters ROLL mode. In this mode, the oscilloscope continuously displays the waveform as a time-domain trend. The waveform is updated from right to left, and the latest data is displayed on the far right.



- No: When the time base is slower than the fastest roll time base, the oscilloscope does not enter ROLL mode and remains in SCAN mode. In SCAN mode, the oscilloscope operates in slow sweep mode. When observing low-frequency signals in this mode, it is recommended to set the channel coupling to DC. The waveform is updated from left to right starting from the trigger position, and the latest data is displayed on the far left.



7.4. Fastest Roll Mode

The Fastest Roll Mode defines the time base threshold for switching between ROLL and SCAN modes. It is a preset system parameter and only provides status indication. It cannot be manually configured.

7.5. Horizontal Position

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.

Adjust the horizontal position using the following methods.

- Select Position input field, use the arrow keys ←, → to position the cursor, then use the Multipurpose to adjust the position value.

  Clockwise: Turn the knob clockwise to increase the horizontal position.

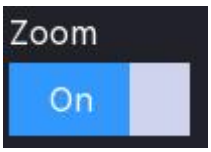
  Counterclockwise: Turn the knob counterclockwise to decrease the horizontal position.
- Use the horizontal Position rotary knob on the front panel to set the horizontal position.

  Clockwise: Turn the knob clockwise to increase the horizontal position.

  Counterclockwise: Turn the knob counterclockwise to decrease the horizontal position.
- Tap the screen and use a drag gesture to adjust the horizontal position. For more details, refer to the section Touch Screen - Drag.
- Double-click Position input field to open the numeric keypad and directly enter the position value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.

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) tab at the top of the screen to enter the Horizontal menu, then select Zoom and turn it ON or OFF.
- Tap the enlarge icon in the upper-right corner of the screen to quickly enable time base

expansion. Tap the exit icon to disable time base expansion.

(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 Horiz Zoom or Vert Zoom input field to open the numeric keypad and enter the value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.
- Tap "+" or "-" next to Horiz Zoom and Vert Zoom input field on the screen to increase or decrease the horizontal zoom ratio (time base) or vertical zoom ratio (volts/div), respectively.
- Select Horiz Zoom or Vert Zoom input field, then use Multipurpose rotary knob to adjust 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. The position of the zoom area can be adjusted using the horizontal Position rotary knob. The size of the zoom area can also be adjusted by changing the horizontal or vertical zoom ratio.

(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 20 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 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

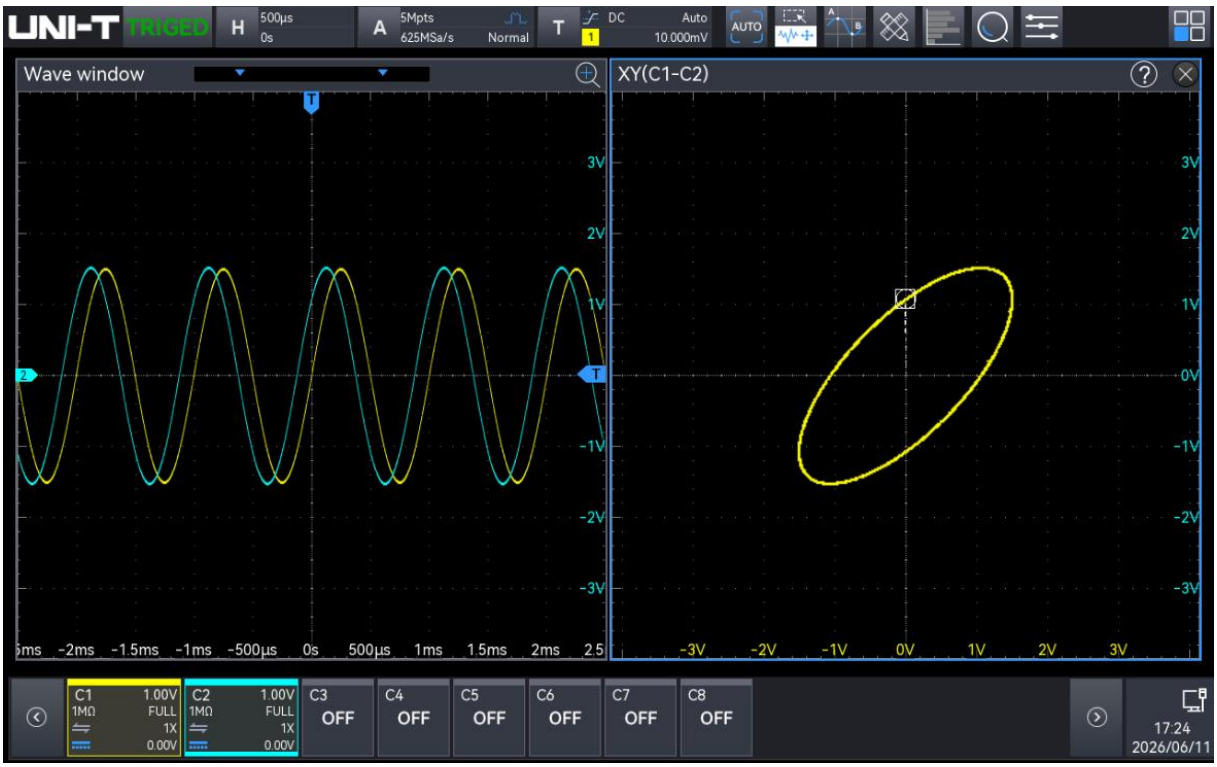
Set the waveform to generate a Lissajous curve, which can be selected from C1-C2, C1-C3, C1-C4, C2-C3, C2-C4, C3-C4, C5-C6, C5-C7, C5-C8, C6-C7, C6-C8, or C7-C8.

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

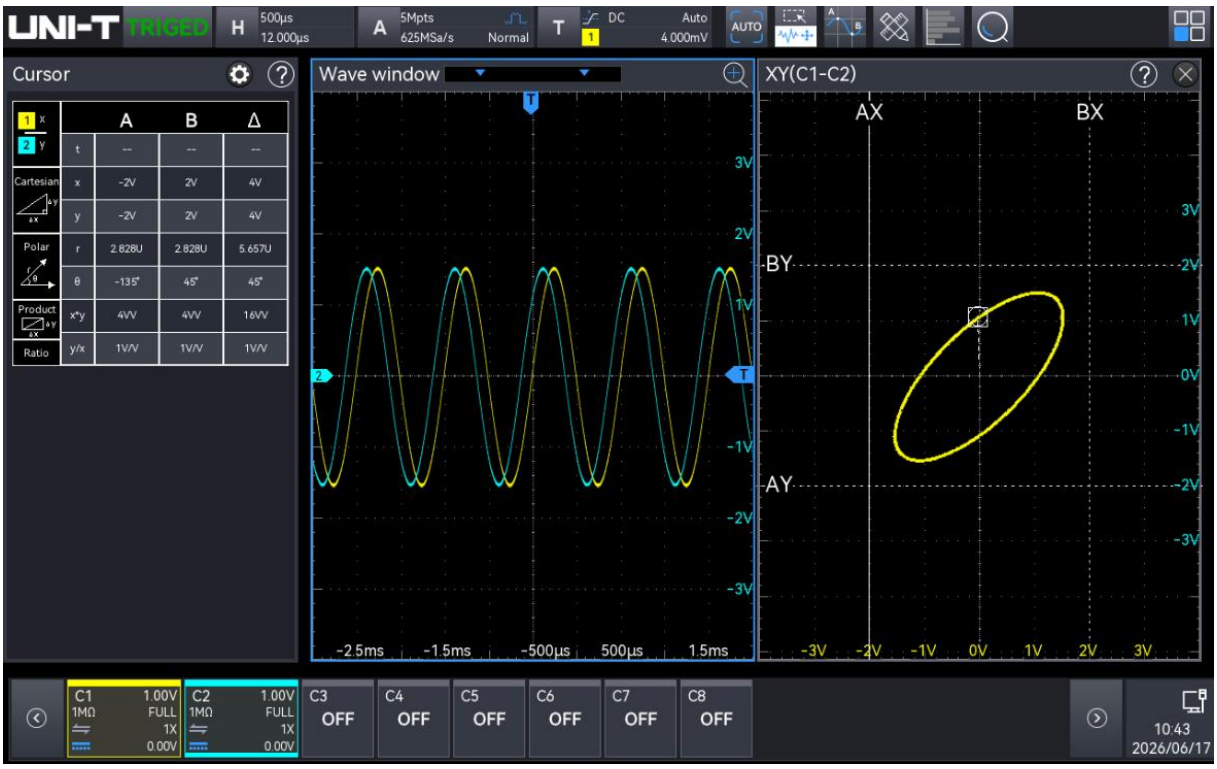
If X - Y is set to C1-C2, input the C1 signal on the horizontal axis (X) and the C2 signal on the vertical axis (Y).

In XY mode, when the X-axis channel is active, use the vertical Position rotary knob to move the XY curve horizontally. When the Y-axis channel is active, use the vertical Position rotary knob to move the XY curve vertically.

The amplitude of each channel can be adjusted using the vertical Scale rotary knob. The time base can be adjusted using the horizontal Scale rotary knob to improve the display of the Lissajous curve. 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 button, 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 accordingly. For voltage measurement, the cursor is displayed in the XY window. For details on using the time and voltage cursors, refer to Cursor Measurement.

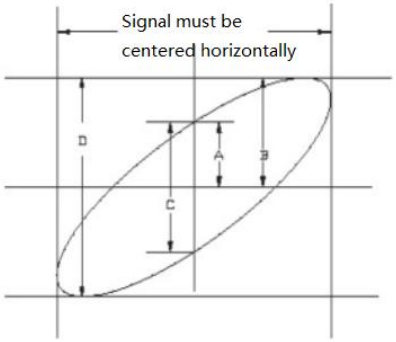
Cursor ①: Time, rectangular coordinates, polar coordinates, product, and ratio.

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

△: Delta (numerical difference between two cursors).

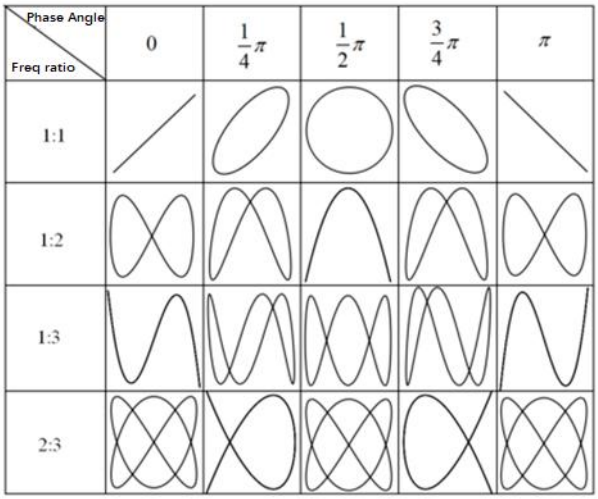
Application of XY Mode

The phase difference between two signals of the same frequency can be conveniently observed using a Lissajous curve. The following figure illustrates the principle for measuring phase difference using this method.



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

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



8. Trigger System

- Noun Explanation of Triggering System
- Edge Triggering
- Pulse Width Triggering
- Video Triggering
- Slope Triggering
- Runt Pulse Triggering
- Window Triggering
- Delay Triggering
- Timeout Triggering
- Duration Triggering
- Setup & Hold Triggering
- Nth Edge Triggering
- Pattern Triggering
- RS232 Triggering
- I^2C Triggering
- SPI Triggering
- CAN Triggering
- CAN-FD Triggering
- LIN Triggering
- FlexRay Triggering
- Audio Triggering
- 1553B Triggering
- Manchester Triggering

- SENT Triggering
- ARINC429 Triggering
- Area 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 acquires the waveform together with its surrounding data and displays it on the screen. Although the oscilloscope continuously acquires waveforms during operation, only those that satisfy stable trigger conditions are displayed.

The trigger function ensures that each time base sweep or acquisition begins from a defined reference condition, synchronizing successive acquisitions. This synchronization allows waveforms to be consistently aligned, resulting in 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 the rising edge of analog channel 1 enables efficient capture of the desired waveform. A clear understanding of the signal under test is essential for obtaining stable and relevant acquisitions.

The UPO6000HD series provides a range of advanced trigger types, including various serial bus triggers. Each trigger type is described in detail in the following sections.

Models supporting advanced protocol trigger and decoding, with standard configuration availability listed in the table below.

Trigger Name	Protocol Trigger	Models	Standard/ Option
Computer serial bus triggering and analysis	RS232/422/485/UART	UPO6000HD Series	Standard
Embedded serial bus triggering and analysis	I^2C, SPI	UPO6000HD Series	Standard
Automobile serial bus triggering and analysis	CAN, LIN	UPO6000HD Series	Option
Automobile serial bus triggering and analysis	CAN-FD	UPO6000HD Series	Option
Automobile serial bus triggering and analysis	FlexRay	UPO6000HD Series	Option
Automobile sensor bus triggering and analysis	SENT	UPO6000HD Series	Option
Audio serial bus triggering	Audio, LJ, RJ, TDM	UPO6000HD Series	Option

and analysis			
Aerospace serial bus triggering and analysis	MIL-STD-1553, ARINC 429	UPO6000HD Series	Option
Wireless communication trigger and analysis	Manchester	UPO6000HD Series	Option

Access the Trigger menu using the following method.

- Tap the T (Trigger) label at the top to open 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 - C8),an external trigger (EXT), or the AC line (mains electricity).

Tap the T (Trigger) label at the top to open the Trigger Setting menu, then select Source to configure the trigger source.

a. Analog channel: Select any of the analog input channels (C1 - C8) on the front panel of the oscilloscope as the trigger signal.

b. External trigger: Select the EXT Trig input 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 from -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 devices, enabling stable synchronization.

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

(2) Impedance

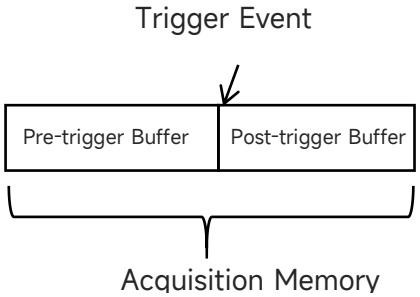
When the trigger source is set to EXT, the impedance of external trigger can be set to 1 MΩ, or 50 Ω.

- 1 MΩ: At this setting, the oscilloscope input impedance is very high, so the current drawn from the measured circuit is negligible.
- 50 Ω: Matches the oscilloscope input to a device with a 50 Ω output impedance.

(3) 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 methods.

- Press the Auto, Normal, or Single button on the front panel to switch between these modes.
- Tap the T (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 50 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 button 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.

Press the Single button again. The waveform on the screen will be cleared, and the oscilloscope will immediately enter Single mode.

The single mode is suitable for the following conditions.

- Capture a single, accidental, or non-periodic event, such as power-up, power-down waveform.
- A rare trigger event.

(4) 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 T (Trigger) label at the top of the screen to open the trigger menu, then select Trigger Coupling to configure 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.

(5) 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 set to C1 - C8, rotate the trigger Position rotary knob on the front panel to adjust the trigger level. If the trigger window is open, tap the trigger level input field, then use the Multipurpose rotary knob to set the trigger level. Alternatively, double-click the trigger level input field to open the virtual keyboard and enter the trigger level. For details on using the numeric keypad, refer to section 5.8 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 window trigger, high level and low level must be set. If the trigger window is open, tap the High or Low input field to open the virtual keyboard and set the trigger level. Two trigger level icons are displayed on the right.

- When the trigger source is set to AC Line, there is no trigger level.
- When the trigger source is set to EXT, rotate the trigger Position rotary knob on the front panel to adjust the trigger level. If the trigger window is open, tap the trigger level input field, then use the Multipurpose rotary knob to set the trigger level. Alternatively, double-click the trigger level input field to open the virtual keyboard and enter the trigger level. For details on using the numeric keypad, refer to section 5.8 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.

(6) 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 T (Trigger) label at the top of the screen to open the trigger menu, then select Trigger Holdoff input field to open the numeric keypad and set the trigger holdoff time (until the waveform is stably triggered, the default setting is 80 ns). For details on using the numeric

keypad, refer to section 5.8 Parameter Setting. The trigger holdoff time can also be adjusted using the Multipurpose rotary knob. Range: 0 s to 10 s.

(7) Noise Rejection

Noise rejection is used to attenuate high-frequency noise in a signal, reducing the probability of false triggers.

Tap the T (Trigger) label at the top of the screen to open the trigger menu, then select Noise Rejection to turn it ON or OFF.

(8) 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 false triggers caused by noise. Range: 0% to 100%.

(9) Force Trigger

Press the Force key to manually generate a trigger signal.

If the waveform does not appear on the screen in Normal or Single mode, press the Force key to acquire the signal baseline and verify that the acquisition is functioning correctly.

(10) Pre-trigger/Delay Trigger

The sampled data is acquired before and after the trigger event.

The trigger position is usually set at the horizontal center of the screen. This allows the user to observe approximately five divisions of pre-trigger and post-trigger data. The waveform can be moved horizontally to view additional pre-trigger information. By examining the pre-trigger data, events occurring before the trigger can be analyzed.

For example, when capturing a glitch at the start of a circuit, observing the pre-trigger data helps identify the cause of the glitch.

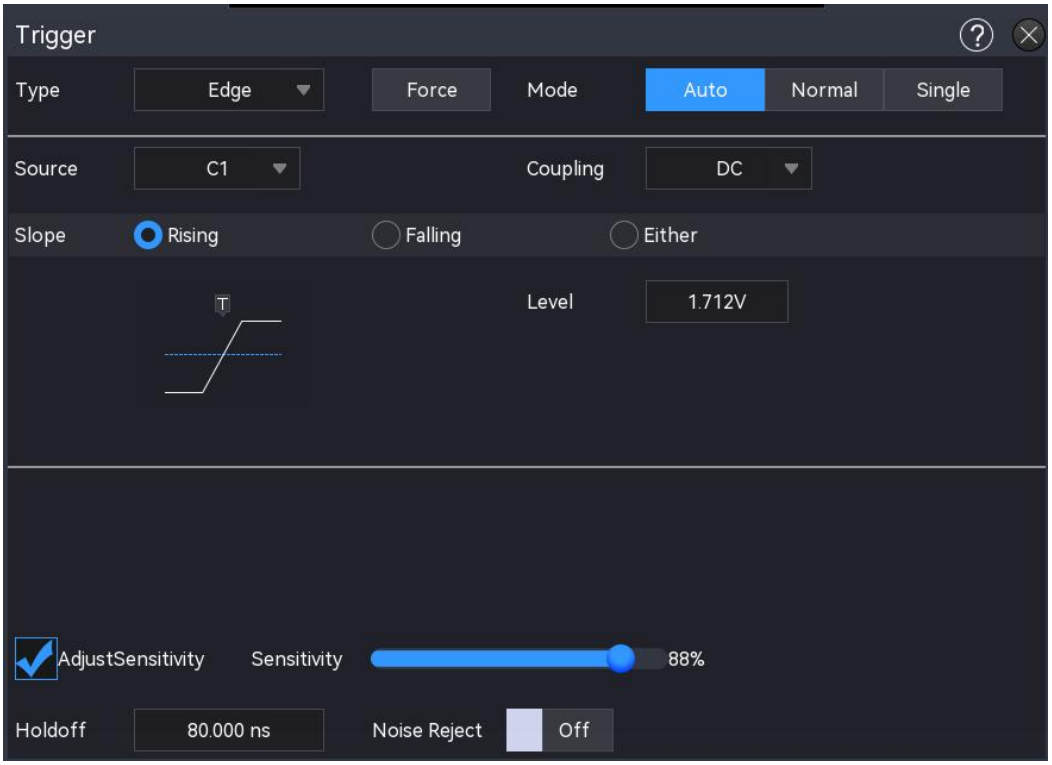
8.2. Edge Triggering

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 T (Trigger) label at the top of the screen to open the trigger menu to configure trigger settings.

(1) Trigger Type

Tap the T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 - C8, AC line, or EXT. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 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 input field, then use Multipurpose rotary knob, trigger Position rotary knob, or

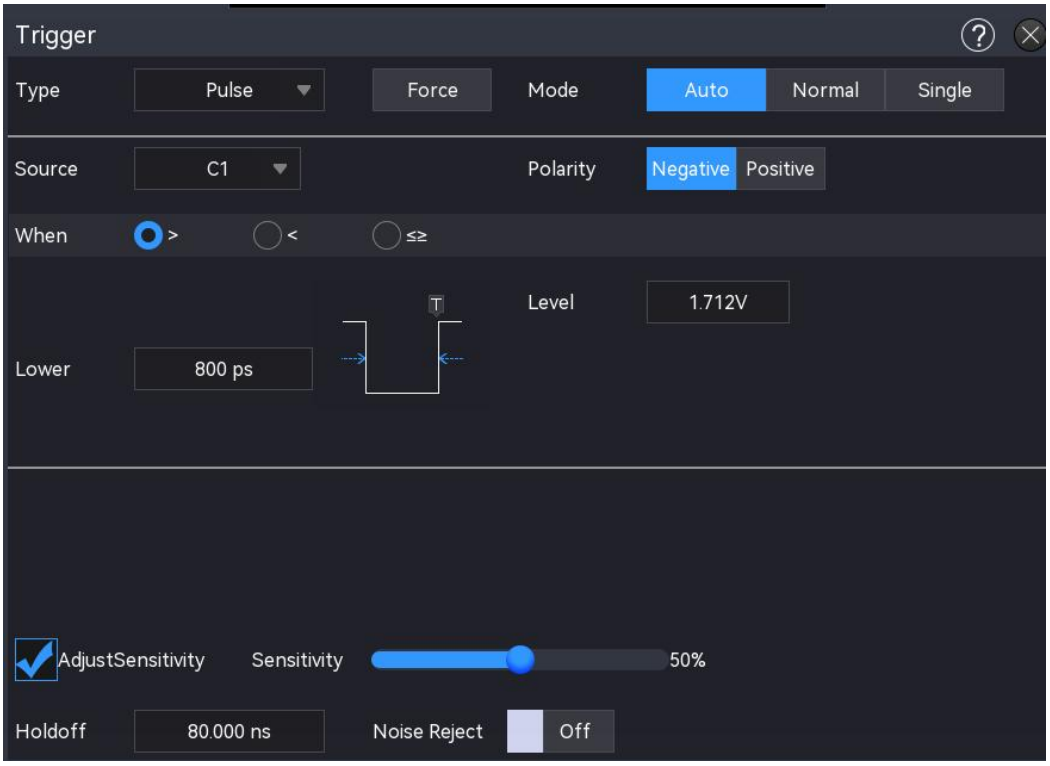
numeric keypad to adjust the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

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 T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list and select Pulse 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 - C8, AC line, or EXT. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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. Range: 800 ps to 4 s.

- When the trigger condition is set to > or <, click Upper or Lower input field to open the numeric keypad and enter the corresponding value. The upper/lower limits can also be set using the Multipurpose rotary knob.
- When the trigger condition is set to ⩽ ⩾, click Upper or Lower input field to open the numeric keypad and enter the corresponding value. The upper/lower limits can also be set using the Multipurpose rotary knob.

  Note: The lower limit must be less than or equal to the upper limit.

(6) Trigger Level

Tap Level input field, then use Multipurpose rotary knob, trigger Position rotary knob, or 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. The UPO6000HD can be triggered 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 T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 Format drop-down list to select the desired format.

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

b. NTSC: The field rate is 60 fields per second, with a frame rate of 30 frames per second and 525 scan lines per frame. The even field is displayed first, followed by the odd field.

c. SECAM: The frame rate is 25 frames per second, with 625 scan lines per frame, using interlaced scanning.

Video Format	Frame Frequency (Frame)	Sweep Type	TV Scan Line (Line)
NTSC	30	Interlaced	525

		scanning	
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.

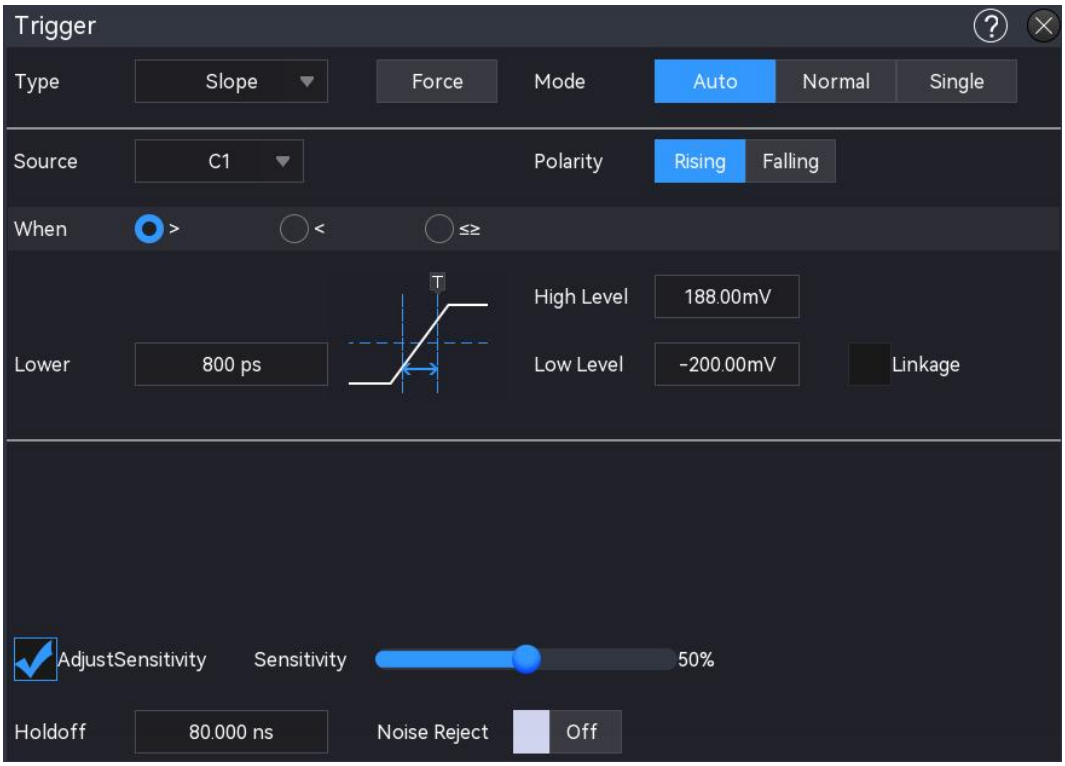
UPO6000HD series incorporates UNI-T proprietary digital 3D technology, featuring a multi-level greyscale display. This allows varying brightness levels to indicate the frequency of different signal components, helping users quickly assess signal quality and identify anomalies during debugging.

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 trigger 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 T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 – C8 For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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.

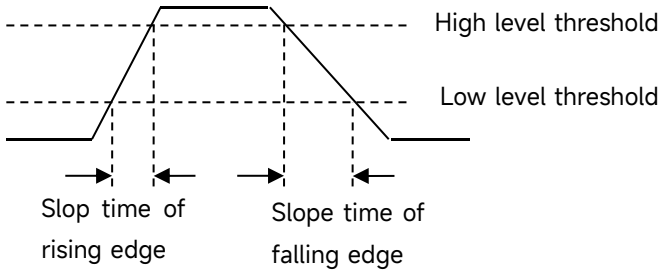
(5) Time Condition

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

Tap High or Low input field, then use the Multipurpose rotary knob, numeric keypad, or press the trigger Position rotary knob to set the trigger level. The selected trigger level is displayed as a solid line on the screen. 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 input field to open the numeric keypad and enter the corresponding value. The lower/upper time limits can also be set using the Multipurpose rotary knob.
- When the trigger condition is set to ⩽ ⩾, click Lower or Upper input field to open the numeric keypad and enter the corresponding value. The lower/upper time limits can also be set using the Multipurpose rotary knob.

  The lower time limit must be less than or equal to the upper time limit.

  Range: 400 ps to 1 s.

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

The formula for calculating the slew rate:

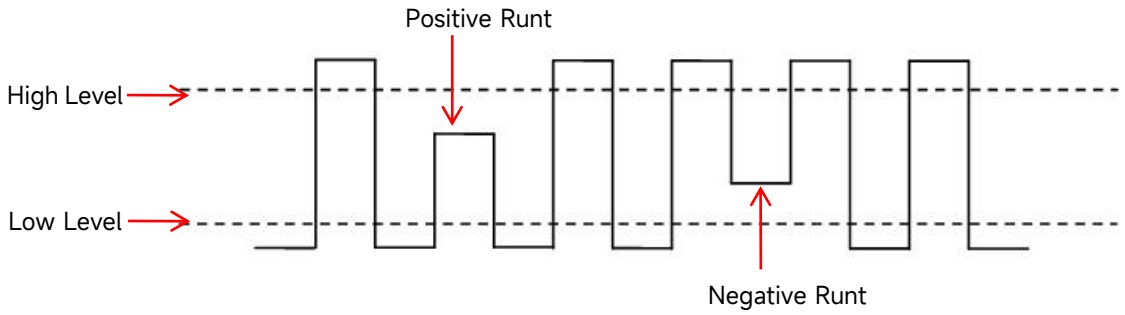
(High Level Threshold - Low Level Threshold) ÷ Time

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

8.6. Runt Triggering

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

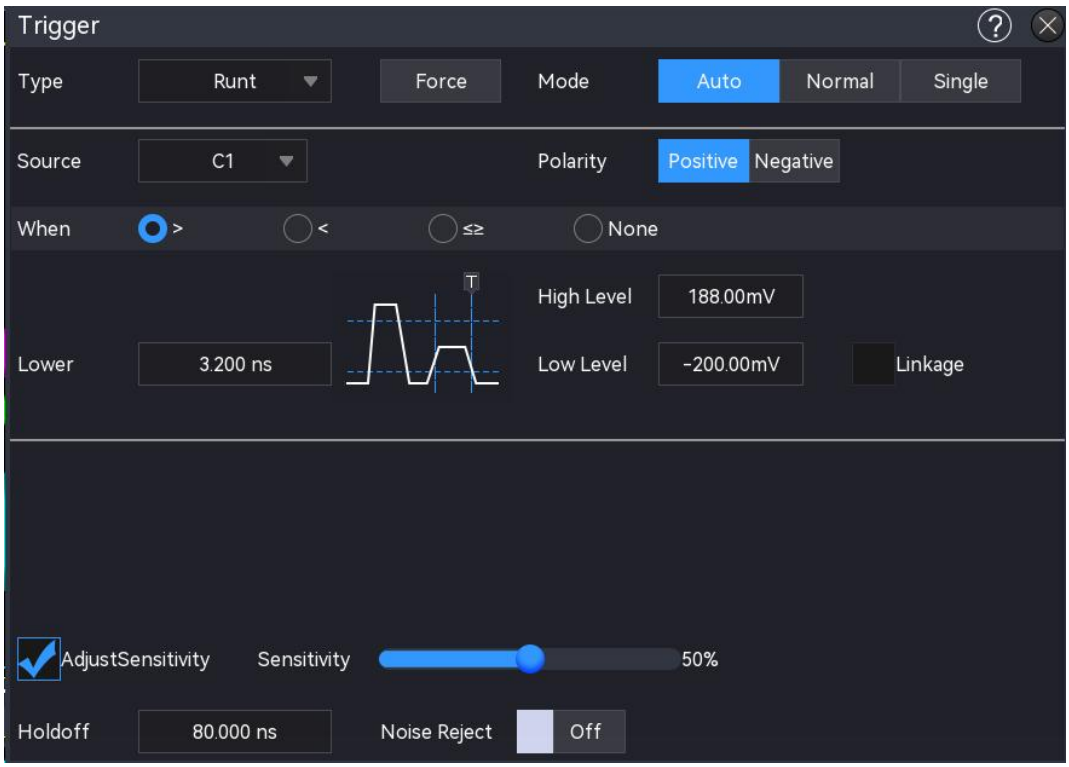
In this oscilloscope, the positive runt pulse is the pulse that crosses the lower limit of the trigger level but does not cross the upper limit of the trigger level; the negative runt pulse is the pulse that crosses the upper limit of the trigger level but does not cross the lower limit of the trigger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



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

(1) Trigger Type

Tap the T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 – C8 For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 triggering requires both the high level and low level to be set. The runt plus triggering can only be stable generated when all conditions are met.

Tap High or Low input field, then use the Multipurpose rotary knob, or numeric keypad to set the trigger 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 input field to open the numeric keypad and enter the corresponding value. The lower/upper time limits can also be set using the Multipurpose rotary knob.
- When the trigger condition is set to ⩽ ⩾, click Lower or Upper input field to open the numeric keypad and enter the corresponding value. The lower/upper time limits can also be set using the Multipurpose rotary knob.

  The lower time limit must be less than or equal to the upper time limit.

  Range: 3.2 ns to 10 s.

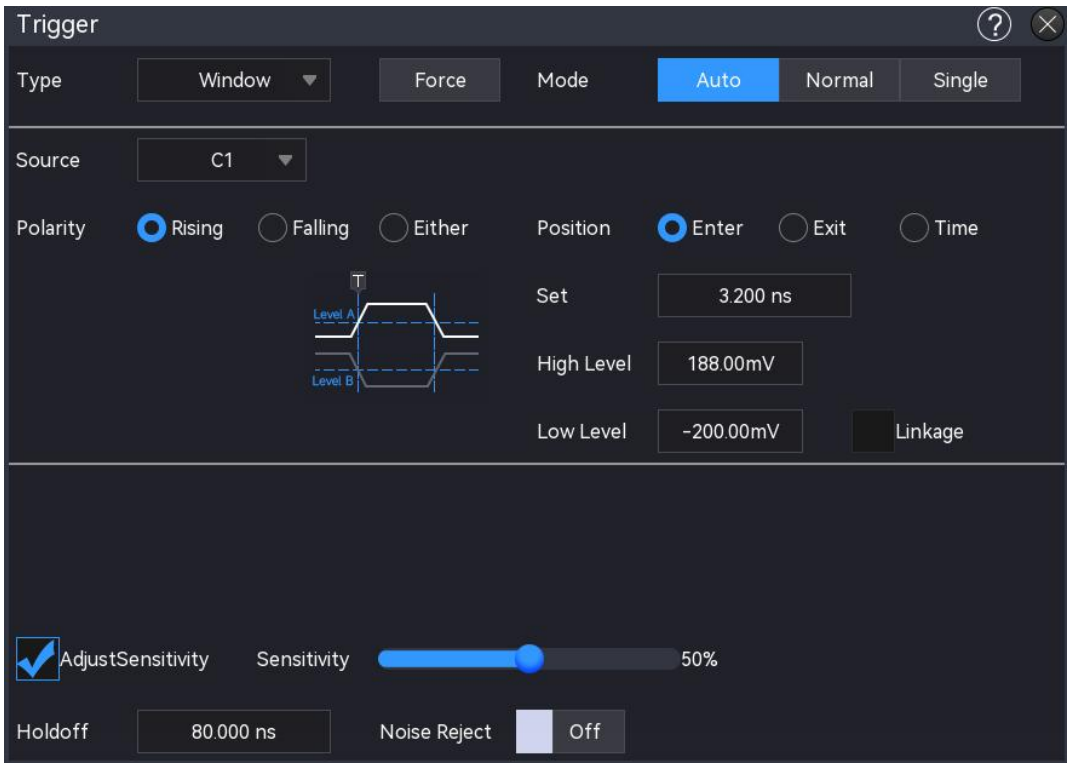
8.7. Window Triggering

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 T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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

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 greater than or 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 time can also be set using the Multipurpose rotary knob. The window time can be set from 3.2 ns to 10 s.

(7) High/Low Level

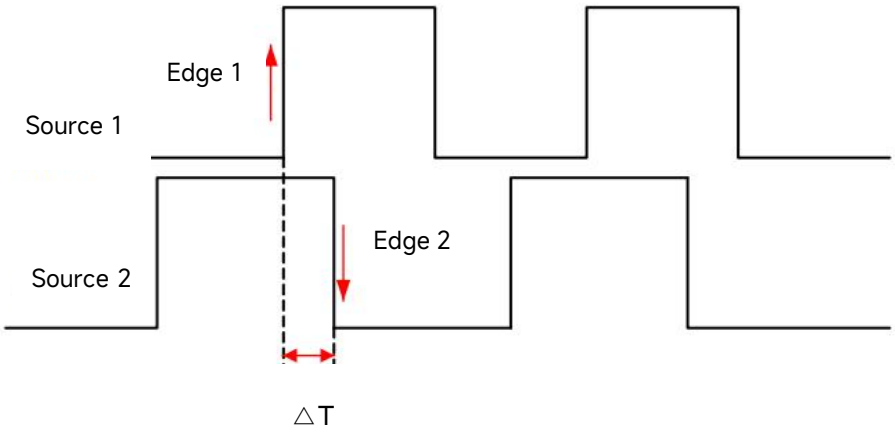
Window triggering requires both the high level and low level to be set. Window triggering can only be stable generated when all conditions are met.

Tap High or Low input field, then use the Multipurpose rotary knob, or numeric keypad to set the trigger 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.



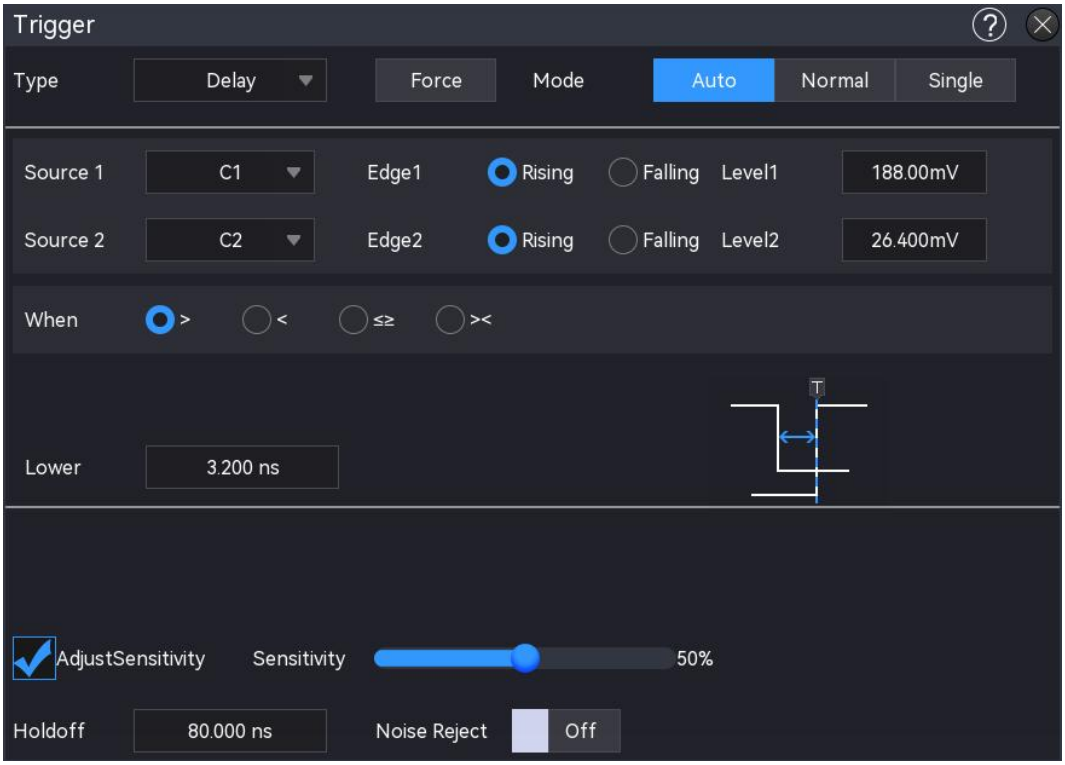
Edge 1 is set as the rising edge. Edge 2 is set as the falling edge. △T is the area marked in red.

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 T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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) Trigger Source 1

Click Source 1 to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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) Trigger Source 2

Click Source 2 to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 less 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 input field to open the numeric keypad and enter the corresponding value. The lower/upper time limits can also be set using the Multipurpose rotary knob.
- When the trigger condition is set to ⩽ ⩾, click Lower or Upper input field to open the numeric keypad and enter the corresponding value. The lower/upper time limits can also be set using the Multipurpose rotary knob.

  The lower time limit must be less than or equal to the upper time limit.

  Range: 3.2 ns to 10 s.

(8) Level 1, Level 2

Delay Trigger requires both Trigger Source 1 and Trigger Source 2 to be set. It can only trigger stably when all conditions are met.

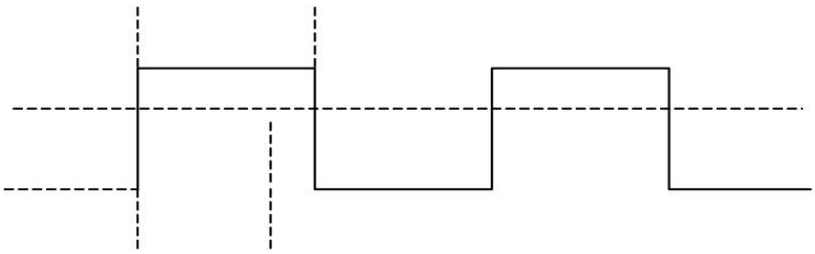
Tap Level 1 or Level 2 input field, then use the Multipurpose rotary knob, or numeric keypad to set the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

8.9. Timeout Triggering

The oscilloscope will be triggered when the time interval (△T) from the rising edge (or falling edge)

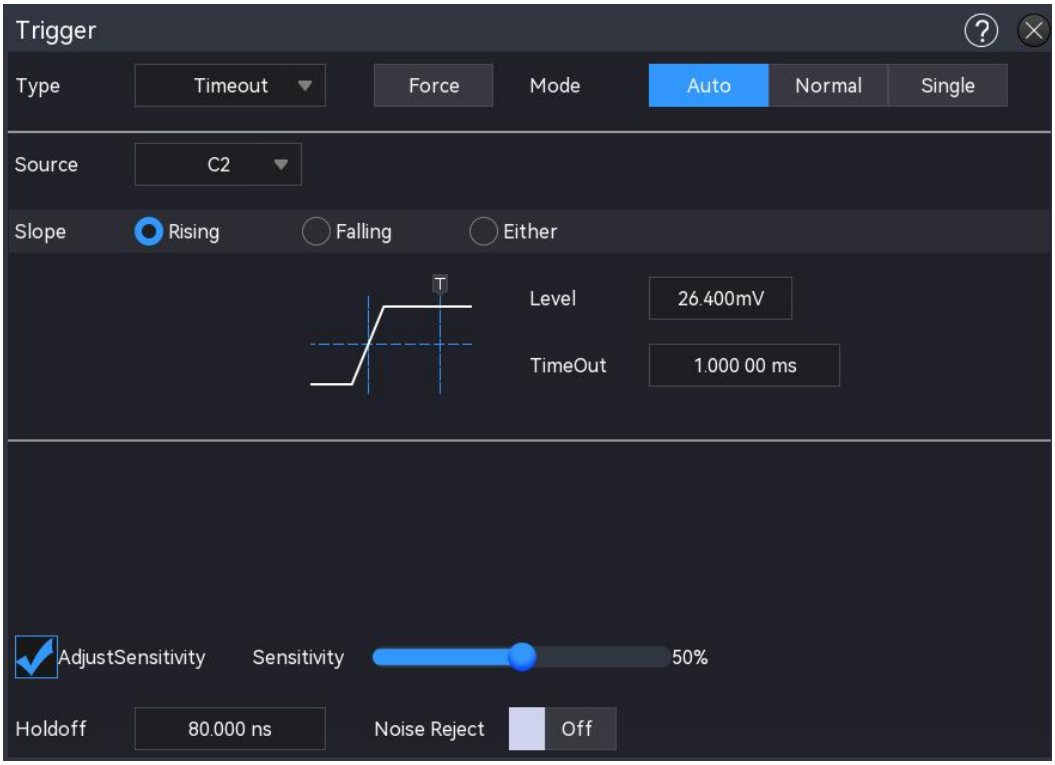


of the input signal across the trigger level and to the adjacent falling edge (rising edge) across the trigger level is greater than the set timeout time, as shown in the following figure.



(1) Trigger Type

Tap the T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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) Slope 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

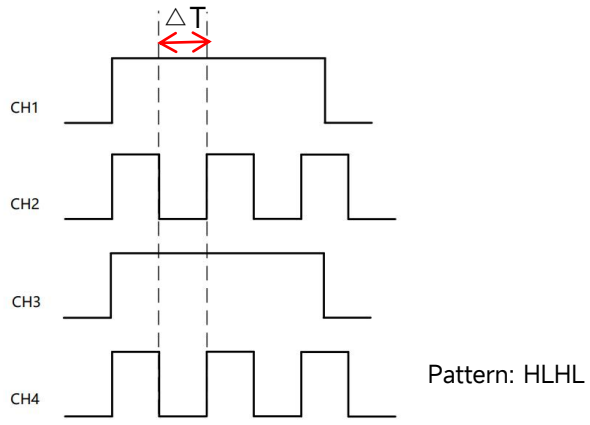
Timeout is used to set the maximum hold time after the input signal across the trigger level. The oscilloscope will be generated when timeout < △T. Click Timeout input field to open the numeric keypad and set timeout. The timeout can also be set using the Multipurpose rotary knob. The timeout can be set from 3.2 ns to 10 s.

(6) Level

Tap Level input field, then use Multipurpose rotary knob, trigger Position rotary knob, or numeric keypad to adjust the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

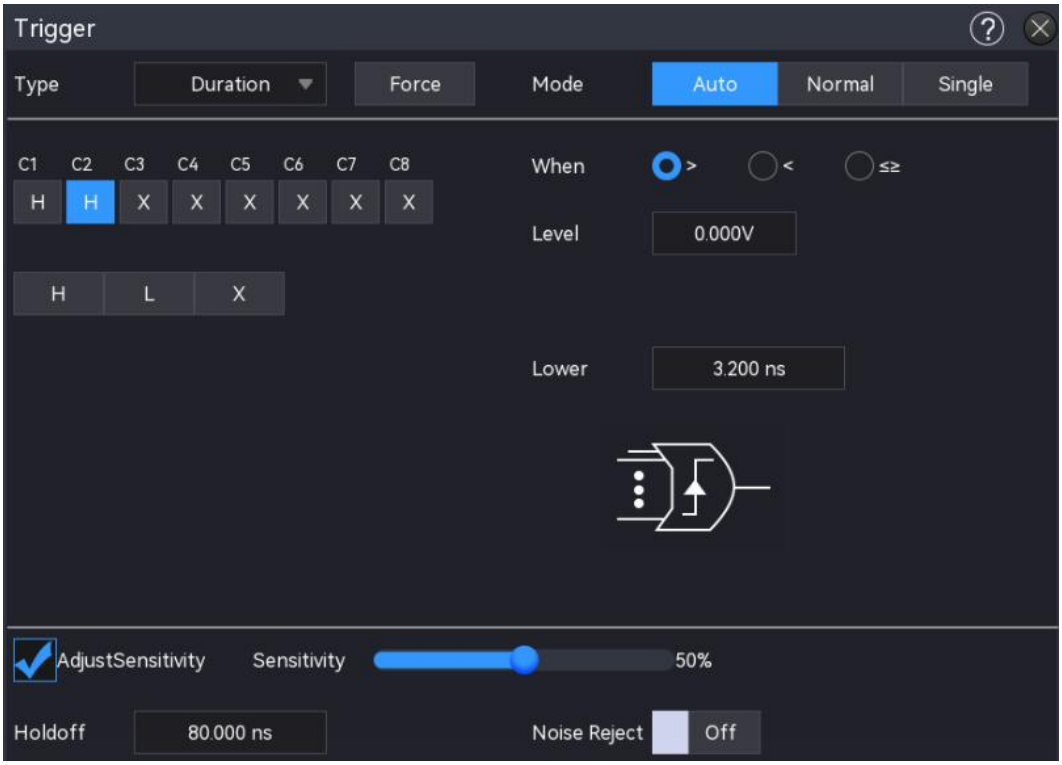
8.10. Duration Triggering

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 T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 - C8, For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 (omitted), 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. >: When the pattern duration is greater than the set pattern duration, the lower time limit can be set.

b. <: When the pattern duration is less than the set pattern duration, the lower time limit can be set.

c. ⩽ ⩾: When the pattern duration is less than or equal to greater than the set pattern duration, and the pattern duration is greater than or equal to greater than the set pattern duration, both the upper and lower limit can be set.

(6) Upper/Lower Time Limit

- When the trigger condition is set to > or <, click Upper or Lower input field to open the numeric keypad and enter the corresponding value. The upper/lower time limits can also be set using the Multipurpose rotary knob.
- When the trigger condition is set to ⩽ ⩾, click Upper or Lower input field to open the numeric keypad and enter the corresponding value. The upper/lower time limits can also be set using the Multipurpose rotary knob.

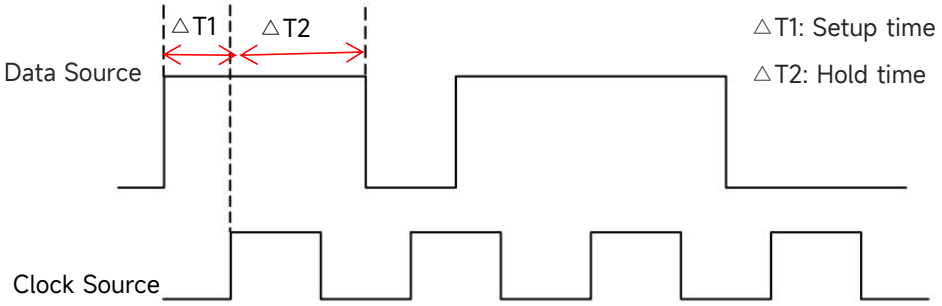
  The lower time limit must be less than or equal to the upper time limit.

(7) Trigger Level

Tap Level input field, then use Multipurpose rotary knob, trigger Position rotary knob, or numeric keypad to adjust the trigger level. 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.

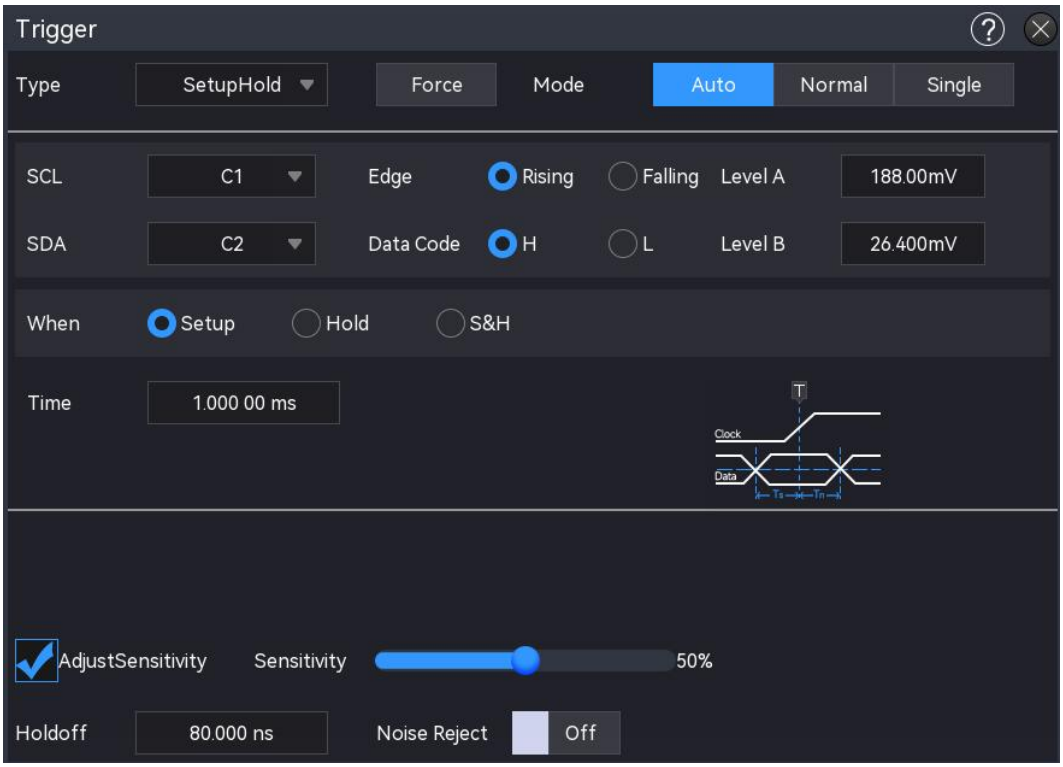


Data Pattern: H

Edge Type: Rising edge

(1) Trigger Type

Tap the T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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) SDA (Data Source)

Click SDA to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 Code

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) SCL (Clock Source)

Click SDA to select C1 – C8 For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 can also be set using the Multipurpose rotary knob. The time can be set from 3.2 ns to 10 s.

(9) Data Level, Clock Level

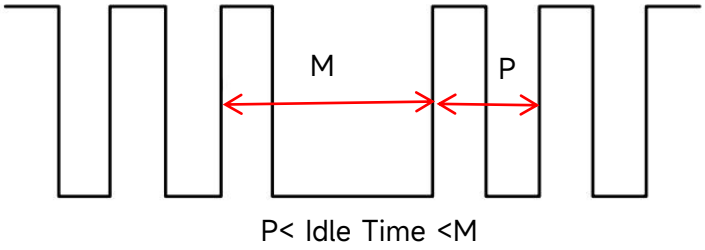
The setup & hold trigger requires both the data level and clock level to be set. It can trigger stably only when all conditions are met.

Tap Data or Clock level input field, then use Multipurpose rotary knob, or numeric keypad to adjust the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

8.12. Nth Edge Triggering

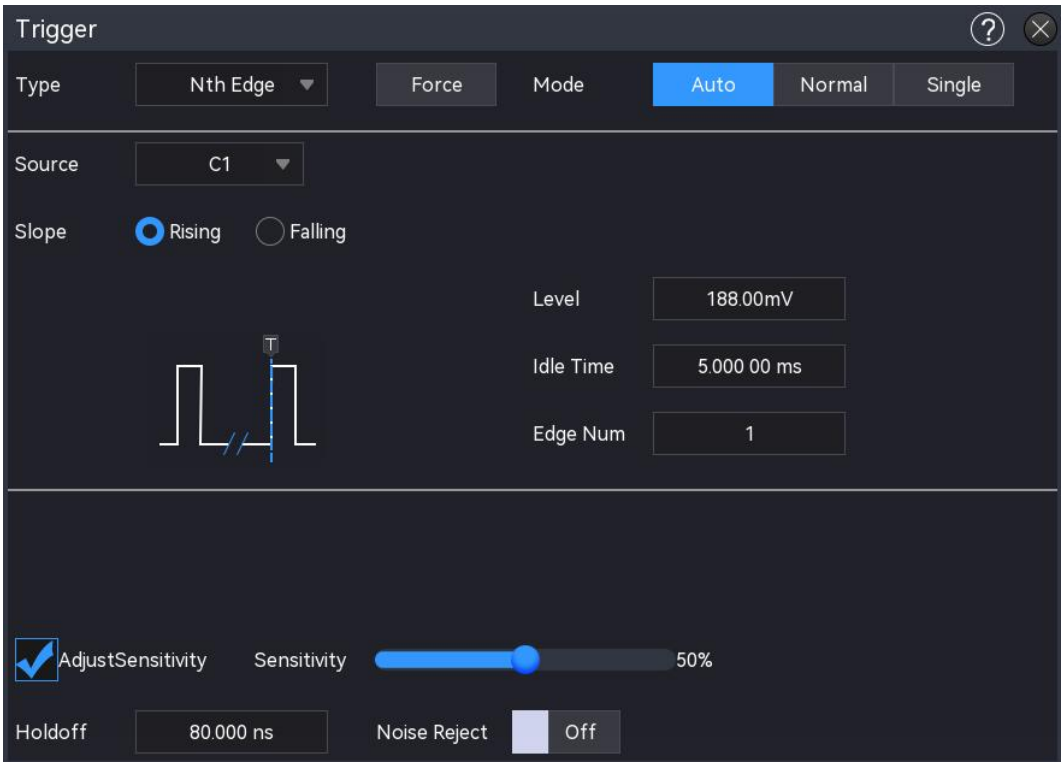
The Nth edge triggering refers to be triggers on the Nth edge after the specified idle time. For example, as shown in the following figure, if the oscilloscope is set to trigger on the 2nd rising edge following the specified idle time (the time between two adjacent rising edges), the idle time should 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.



(1) Trigger Type

Tap the T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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) Slope 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 can also be set using the Multipurpose rotary knob. The idle time can be set from 3.2 ns to 10 s.

(6) Edge Number

The edge count represents the Nth edge value. Click Edge Number input field to open the numeric keypad and set the edge count. The edge number can also be set using the Multipurpose rotary knob. The edge count range can be set from 1 to 65535.

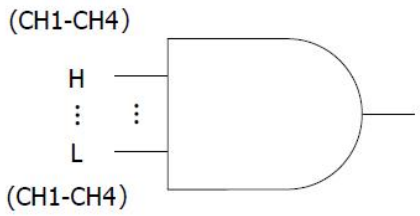
(7) Trigger Level

Tap Level input field, then use Multipurpose rotary knob, trigger Position rotary knob, or numeric keypad to adjust the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

8.13. Pattern Triggering

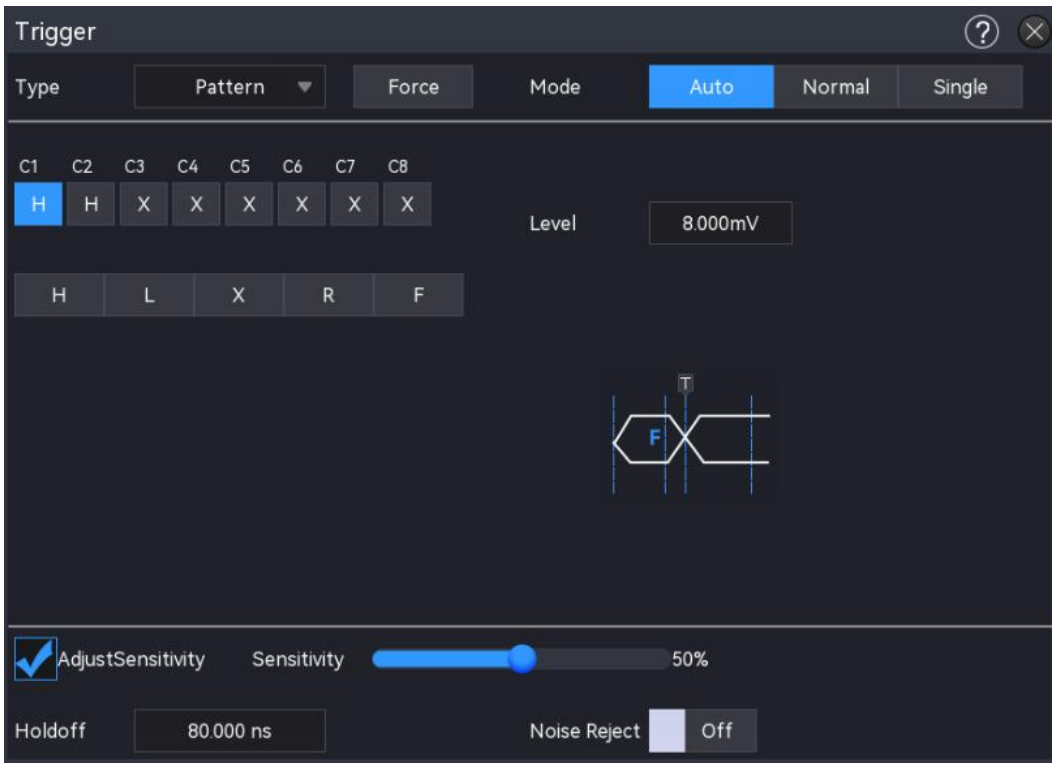
The pattern triggering identifies the trigger condition by searching for the specified patterns. This pattern is a logical "AND" combination of channels. Each channel can be set to H (high), L (low), or X (ignore).

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 does not trigger.



(1) Trigger Type

Tap the T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list 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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 (omitted), i.e. the channel is not part of the code pattern, the oscilloscope will not be triggered if all channels 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

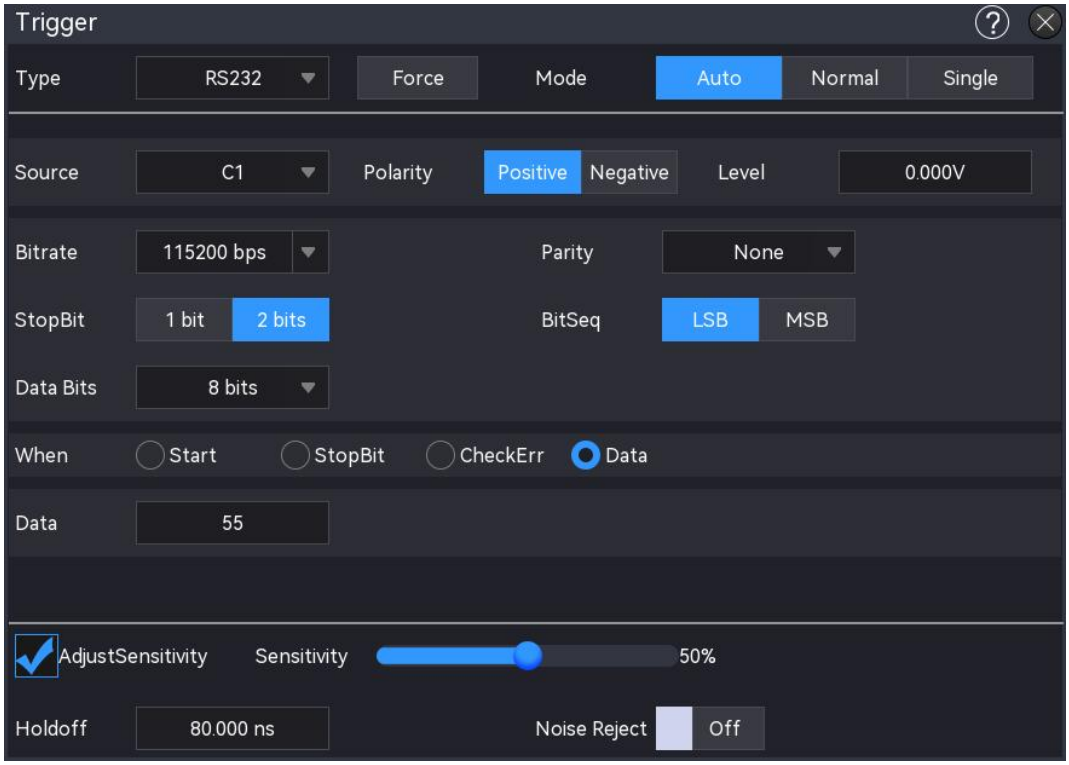
Tap Level input field, then use Multipurpose rotary knob, trigger Position rotary knob, or numeric keypad to adjust the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System.

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 T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list and select RS232 as the trigger type.



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob. 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 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.

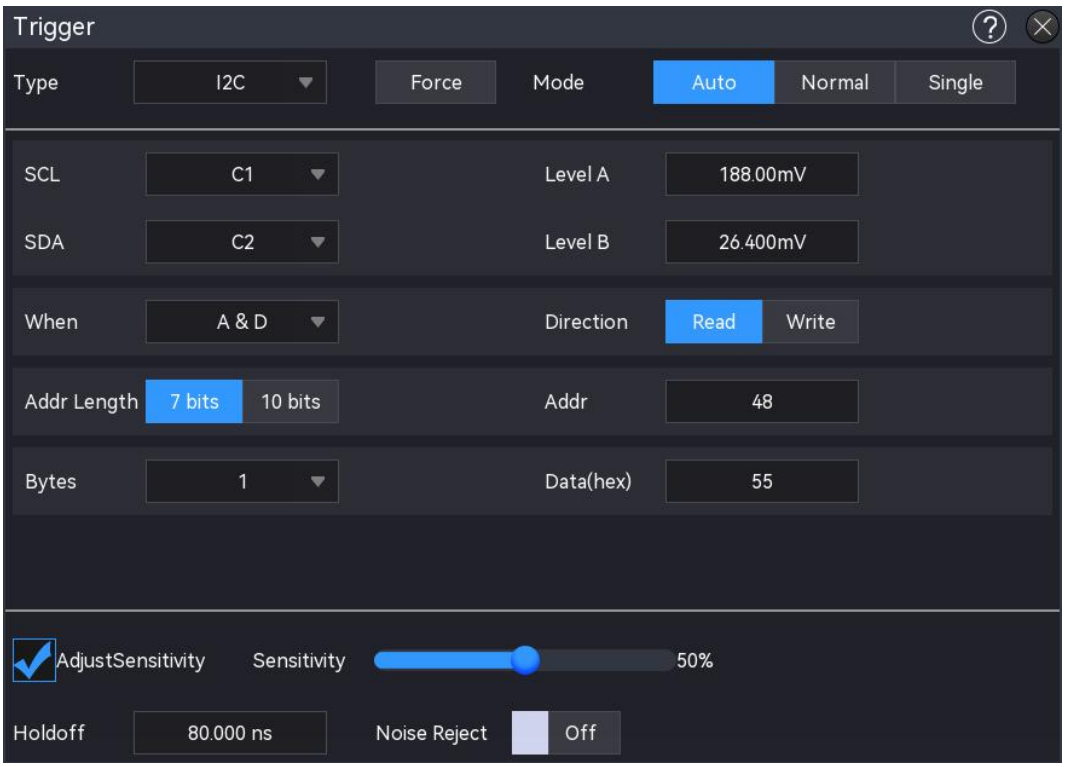
- Data: The data depends on the frame length. Double-click Data input field to open the numeric keypad and set the data. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data. Range: 00 – 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 T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list and select I²C as the trigger type.



(2) Trigger Source

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

a. SCL

Click SCL to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on the top of the screen.

b. SDA

Click SDA to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on the top of the screen.

(3) Trigger Level

Click Level A or Level B input field to open the numeric keypad and enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob. The selected trigger level is displayed as a solid line.

(4) Operating Direction

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 5.8 Parameter Setting. Alternatively, select the Address, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the address. Range: 00 - 7F, 000 - 3FF.
- 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 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data. 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.

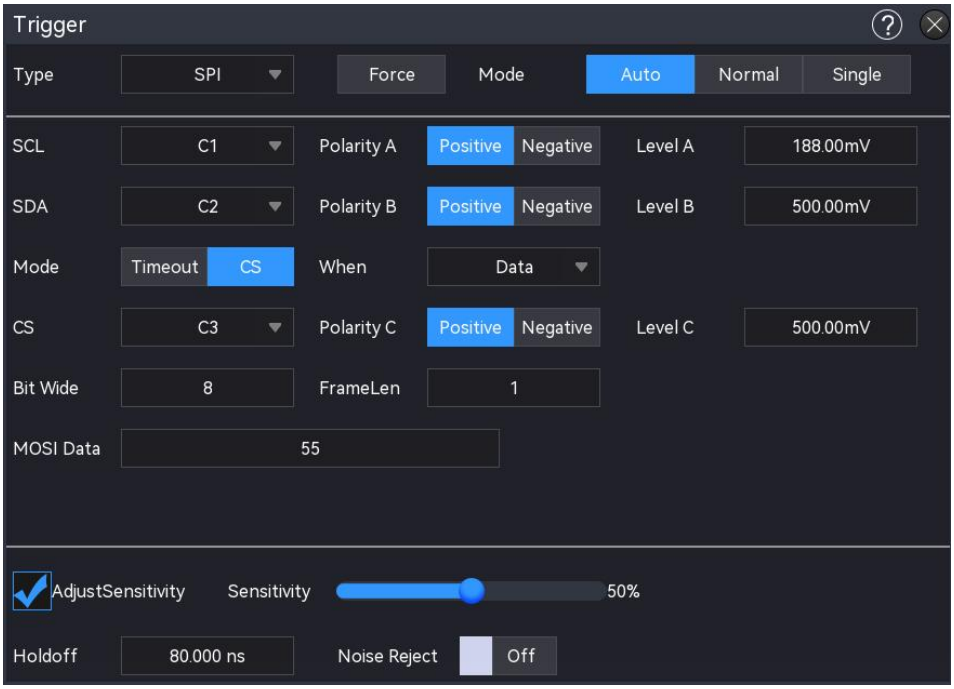
When the address length, address, byte length, or data is selected, the corresponding

menus can be configured. For the setting of each parameter, refer to *Address and Data* under Trigger Condition described above.

8.16. SPI Triggering

(1) Trigger Type

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



(2) Trigger Source

Set both the clock source (SCL), the data source (SDA), and chip select source (CS). The oscilloscope will only trigger stably if the selected channel has a connected signal and is set as the trigger source.

a. SCL

Click SCL to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on the top of the screen.

b. SDA

Click SDA to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on the top of the screen.

c. CS

Click CS to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun

Explanation of Triggering System.

The current source is displayed in the trigger label on the top of the screen.

(3) Polarity 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

Click Level A, Level B, or Level C input field to open the numeric keypad and enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob. The selected trigger level is displayed as a solid line.

(5) Mode

- 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 select: 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

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 5.8 Parameter Setting. Alternatively, select the Timeout, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the timeout. Range: 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 5.8 Parameter Setting. Alternatively, select the Bit Width, and then use the Multipurpose rotary knob to adjust the timeout. Range: 4 – 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 5.8 Parameter Setting. Alternatively, select the Frame Length, and then use the Multipurpose rotary knob to adjust the frame length. Range: 1 – 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 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data. The setting range is from 0 – FFFFFFFFFFFFFFFFFFFFFFFFFFFFFFFF (32 Fs).

8.17. CAN Triggering

(1) Trigger Type

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



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob. 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) 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 5.8 Parameter Setting. Alternatively, select the Identifier, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the identifier. 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. Range: 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 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data.

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 *ID* and *Data* under Trigger Condition described 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.

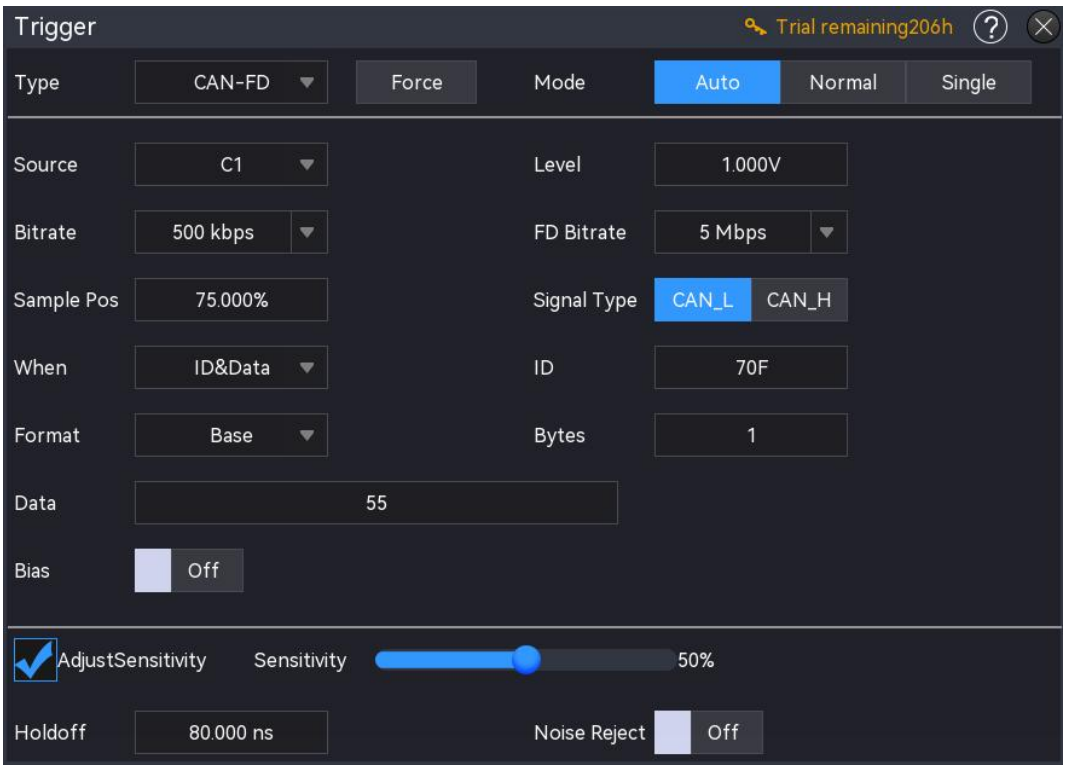
l. CRC error: Triggers when a CRC error occurs.

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

8.18. CAN-FD Triggering

(1) Trigger Type

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



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob. 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) 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 custom.

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

(7) Sample 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 Sample Position input field to open the numeric keypad and set the sampling position. The sample position can also be adjusted using the Multipurpose rotary knob. Range: 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.

- 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 5.8 Parameter Setting. Alternatively, select the ID, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the ID. 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 and Extended formats are suitable for CAN signals. The FD Standard and FD Extended formats are suitable for CAN-FD signals.

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 5.8 Parameter Setting. Alternatively, select the Byte Length, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the byte length. Range: 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 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data.
- Offset: Sets the data offset of the byte data for the 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 5.8 Parameter Setting. Alternatively, select the Byte Offset, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the byte offset. Range: 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 *ID* and *Data* under Trigger Condition described 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.

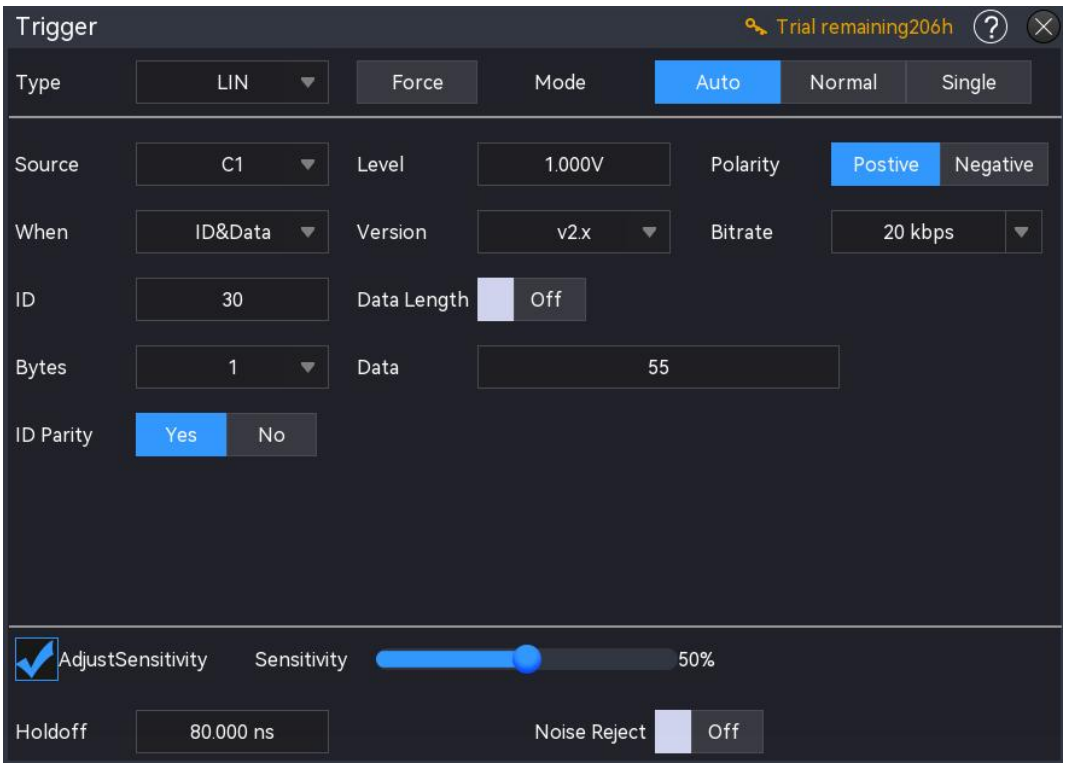
l. CRC error: Triggers when a CRC error occurs.

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

8.19. LIN Triggering

(1) Trigger Type

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



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob. 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.

(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

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 5.8 Parameter Setting. Alternatively, select the Data Length, then use the Multipurpose rotary knob to adjust the data length. Range: 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 5.8 Parameter Setting. Alternatively, select the ID, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the ID.

  When ID Parity Check is set to ON: 00 – FF.

  When ID Parity Check is set to OFF: 00 - 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. Range: 1 to 8.
- 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 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data. 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 *ID* and *Data* under Trigger Condition described

above.

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

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

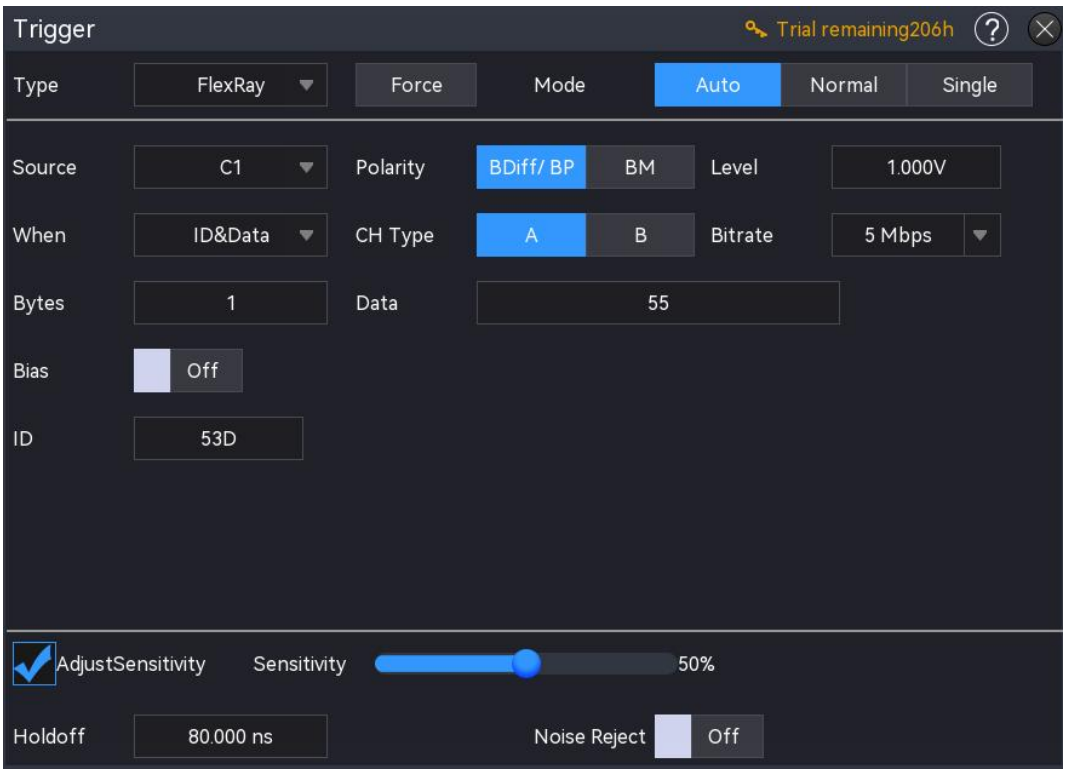
g. Error: Triggers on the LIN error frame. When the error type is selected, the corresponding menus can be configured.

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

8.20. FlexRay Triggering

(1) Trigger Type

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



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob.

(5) Channel Type

Click CH 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 Indicating bit is selected, the corresponding 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 corresponding 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 5.8 Parameter Setting. Alternatively, select the ID, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the ID. 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 corresponding 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 5.8 Parameter Setting. Alternatively, select the ID, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the period number. Range: 00 - 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.

- Indicating bit: Double-click Indicating Bit to open the numeric keypad and set the

indicating bit. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Indicating Bit, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the indicating bit. Range: 00 - 1F.

- ID: For setting ID, refer to *ID* under Trigger Condition described above.
- Static load length: 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 5.8 Parameter Setting. Alternatively, select the Static Load Length, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the static load length. Range: 00 - 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 5.8 Parameter Setting. Alternatively, select the Header CRC, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the header CRC. Range: 000 - 7FF.
- Period number: For setting the period number, refer to *Period Number* under Trigger Condition described 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 5.8 Parameter Setting. Alternatively, select the Byte Length, then use the Multipurpose rotary knob to adjust the byte length. Range: 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 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data. 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 5.8 Parameter Setting.

Alternatively, select the Byte Offset, then use the Multipurpose rotary knob to adjust the byte offset. Range: 0 to 253.

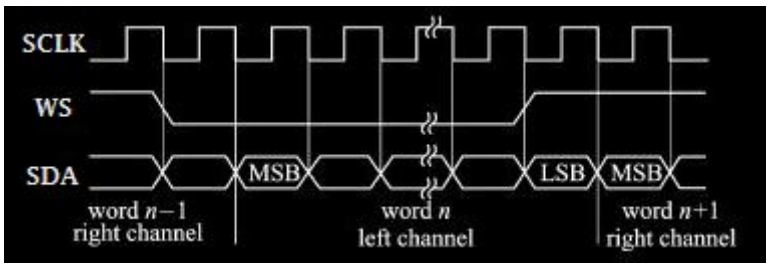
g. ID and Data: Triggers on the data frame that matches the specified ID and data.
When the ID, byte length, bias, data, or byte offset is selected, the corresponding menus can be configured. For the setting of each parameter, refer to *ID* and *Data* under Trigger Condition described above.

h. End of frame: Triggers at the end of frame of the oscilloscope.
When the frame type is selected, the corresponding 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.
- 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.

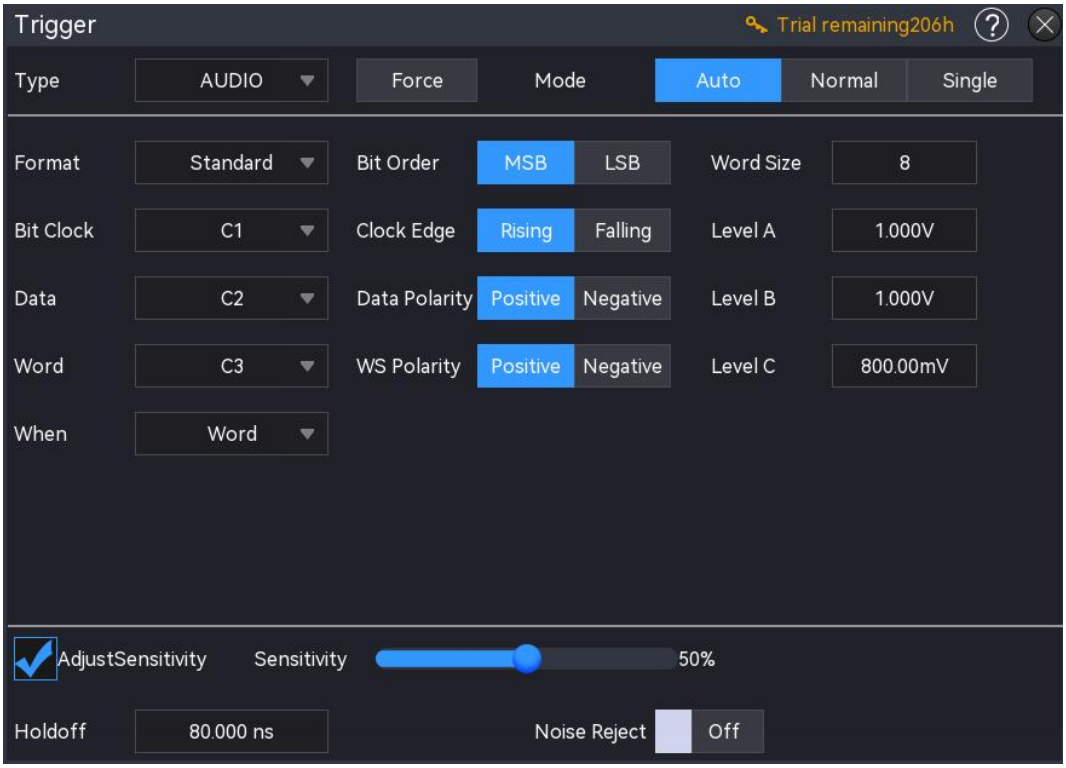
8.21. Audio Triggering

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

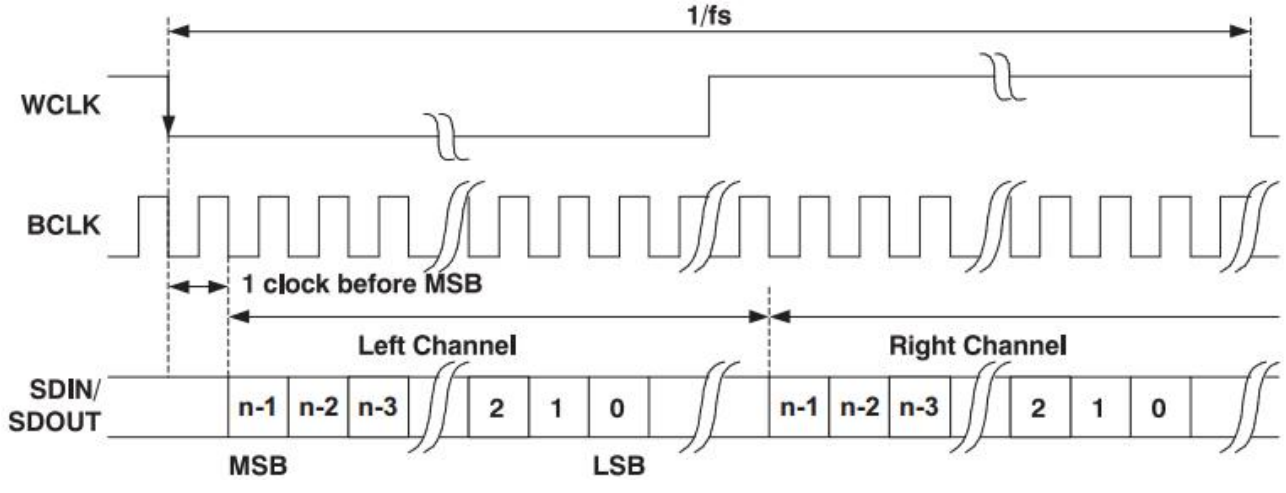
Tap the T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list and select Audio as the trigger type.



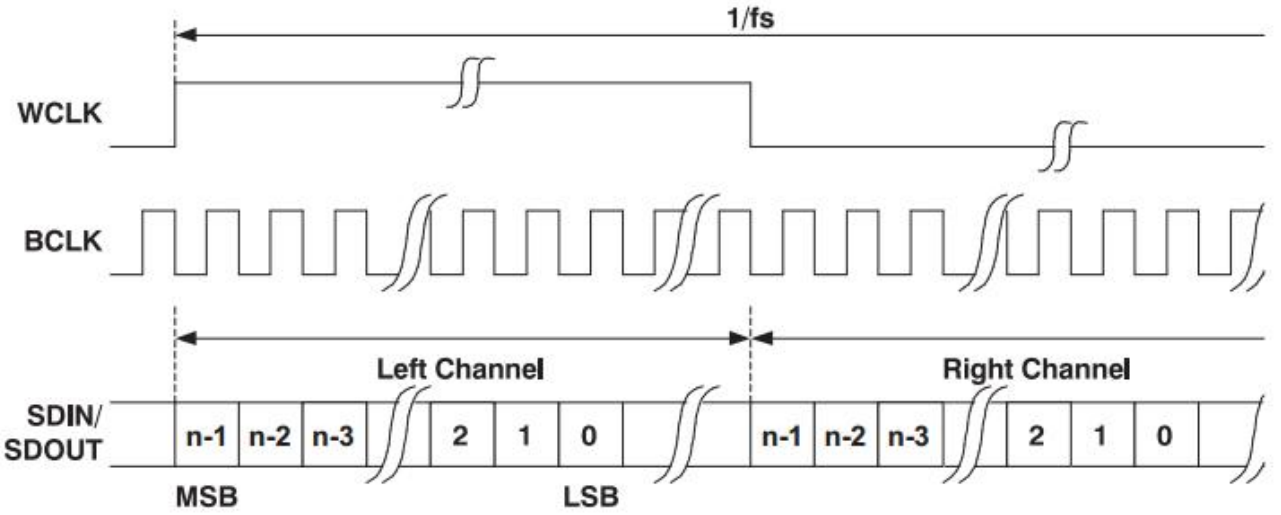
(2) Format

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

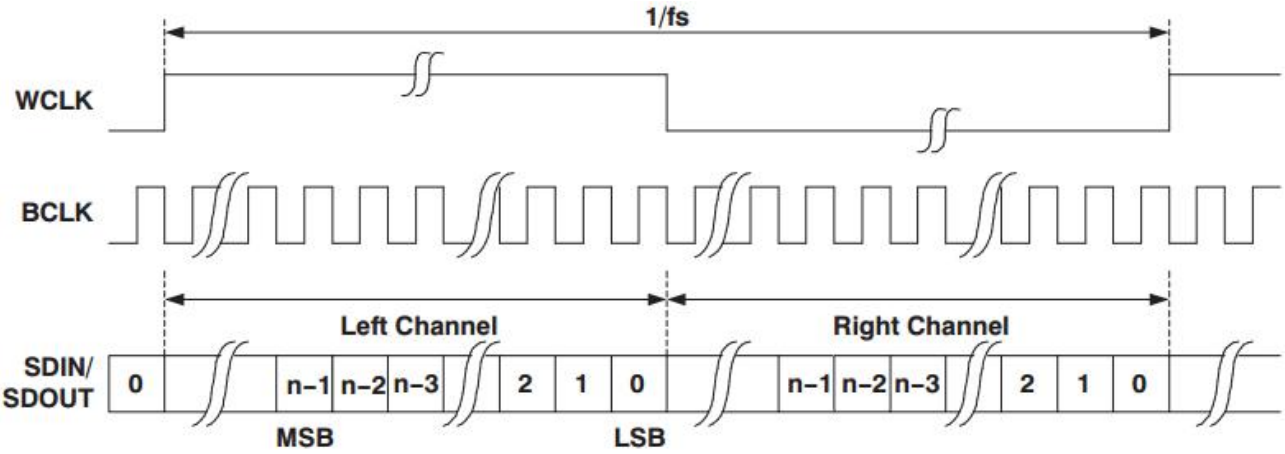
- Standard: The MSB of each sampled data is sent first, followed by the LSB. The MSB is displayed on the SDATA line one clock bit after the WS transition edge.



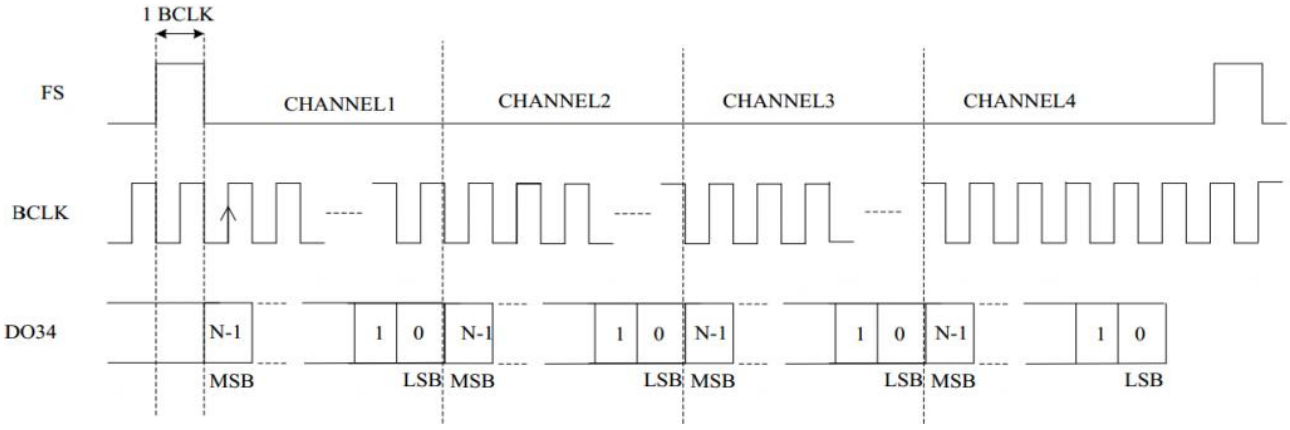
- Left justifying: Data transmission (MSB first) starts at the WS transition edge, without the one-bit delay used in the standard format.



- 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 setting 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 - C8 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. Word Select

Click Word to select C1 - C8 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. Word select indicates whether the currently transmitted audio data belongs to the left channel or the right channel.

c. Data

Click Data to select C1 - C8 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 Data to select C1 - C8 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.

(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

Click Level A, Level B, Level C, or Level D input field to open the numeric keypad and enter the

trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob. The selected trigger level is displayed as a solid line.

(7) TDM Data Format (do not select TDM)

When the data format is set to standard, left justifying, or right justifying is selected, the corresponding 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 5.8 Parameter Setting. Alternatively, select the Bit Size Length, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the bit size. Range: 4 to 32.

b. Trigger Type

- Word select: Triggers on the word select signal.
- 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 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data.

(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 5.8 Parameter Setting. Alternatively, select the Data Bit per Channel, then use the Multipurpose rotary knob to adjust the data bit. Range: 4 to 32. Data bits per channel ≤ Clock bits per channel.

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 5.8 Parameter Setting. Alternatively, select the Clock Bit per Channel, then use the Multipurpose rotary knob to adjust this value. Range: 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 5.8 Parameter Setting. Alternatively, select the Channel Number per Frame, then use the Multipurpose rotary knob to adjust this value. Range: 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 5.8 Parameter Setting. Alternatively, select the Bit Delay, then use the Multipurpose rotary knob to adjust the data bit. Range: 0 to 31. Bit delay setting < Clock bits per channel.

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 corresponding 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 5.8 Parameter Setting. Alternatively, select the Data, then use the Multipurpose rotary knob to adjust the data.
- Channel and data: Triggers when the channel and data meet the setting value.

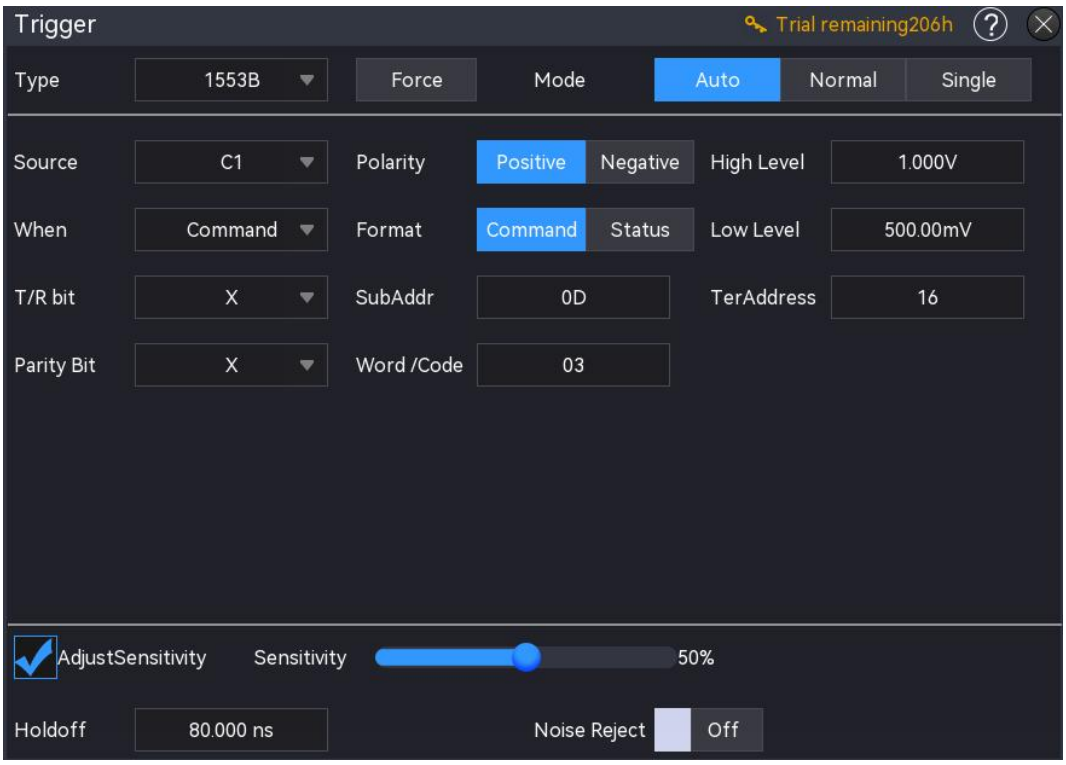
  When the channel number or data is selected, the corresponding menus can be configured.

  - Channel number: Double-click Channel Number input field to open the numeric keypad and set the channel number. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Channel, then use the Multipurpose rotary knob to adjust the channel number.
  - 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 5.8 Parameter Setting. Alternatively, select the Data, then use the Multipurpose rotary knob to adjust the data.

8.22. 1553B Triggering

(1) Trigger Type

Tap the T (Trigger) label at the top of the screen to open the trigger menu, then tap Type drop-down list and select 1553B as the trigger type.



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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) High/Low Level

Double-click High or Low input field, then use the Multipurpose rotary knob, or numeric keypad to set the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System. The selected trigger level is displayed as a solid line on the screen.

(5) Format

Click Format to set command word or state word.

When the format is set to Command Word, the Status trigger condition is hidden. When the format is set to Status Word, the Command trigger condition is 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 Terminal Address input field to open the numeric keypad to set the terminal address. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Terminal Address, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the terminal address.
- T/R bit: Click T/R Bit to select X, 0 (R), or 1 (T). The default setting is X.
- Sub-address/mode: Set the sub-address for a command word. Double-click Sub-address/Mode input field to open the numeric keypad to set the sub-address. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Sub-address, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the sub-address.
- Word count/code: Set the word count/code for a command word. Double-click Word Count/Code input field to open the numeric keypad to set the word count/code. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Word Count/Code, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the word count/code.
- Parity check: Click Parity Check to select X, 0, or 1. The default setting 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 command word. Double-click Terminal Address input field to open the numeric keypad to set the terminal address. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Terminal Address, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the terminal address.
- Error message (9): Click Error message (9) to select X, 0, or 1. The default setting is X.
- Instr (10): Click Instr (10) to select X, 0, or 1. The default setting is X.
- Service request (11): Click Service request (11) to select X, 0, or 1. The default setting is X.
- BCR (15): Click BCR (15) to select X, 0, or 1. The default setting is X.
- Busy (16): Click Busy (16) to select X, 0, or 1. The default setting is X.
- System flag (17): Click System flag (17) to select X, 0, or 1. The default setting is X.
- DBCA (18): Click DBCA (18) to select X, 0, or 1. The default setting is X.

- Terminal flag (19): Click Terminal flag (19) to select X, 0, or 1. The default setting is X.
- Parity check: Click Parity check to select X, 0, or 1. The default setting 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 Data input field to open the numeric keypad to set the data. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, →, to move the cursor, and then use the Multipurpose rotary knob to adjust the data.
- Parity check: Click Parity check to select X, 0, or 1. The default setting is X.

e. Error: Triggered on the specified error type.

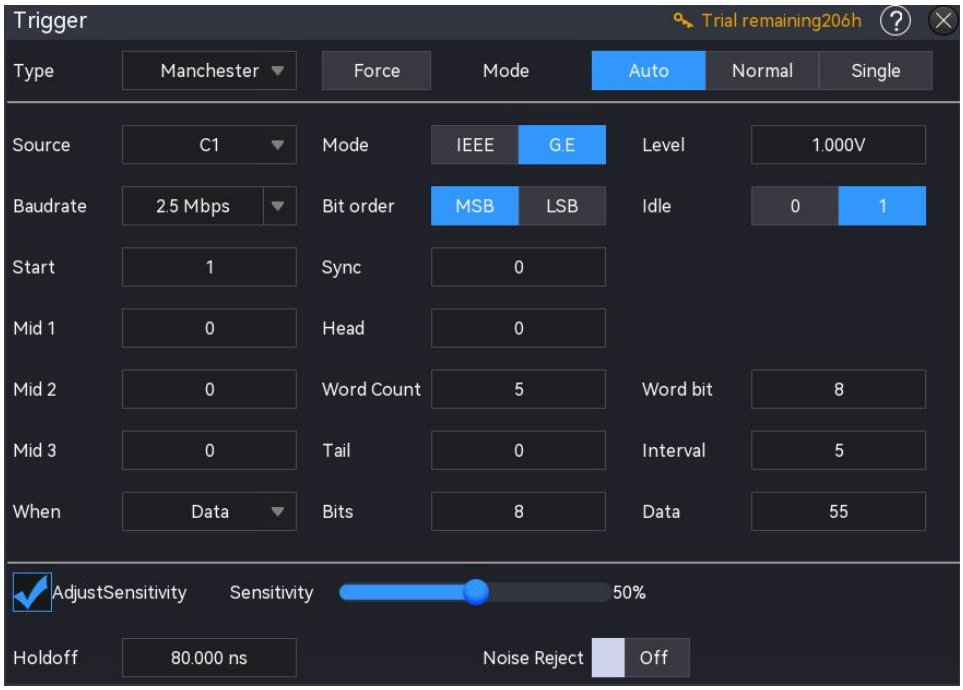
When the error type (Parity check, synchronization, Manchester, and non-continuous data) is selected, the corresponding menus can be configured.

- 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.

8.23. Manchester Triggering

(1) Trigger Type

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



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob.

(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) Baud Rate

Click Baudrate 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 corresponding 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 5.8 Parameter Setting. Alternatively, select the Header Field, use the arrow

keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the header field length.

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

When the data field is selected, the corresponding 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 using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Bitrate, then use the Multipurpose rotary knob to adjust the bitrate value.
- 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 5.8 Parameter Setting. Alternatively, select the Data Field, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data field.

d. Tail field: Triggers on the tail field when the condition is met.

- Tail field: Sets the trigger data for the tail field. The data length that can be set is limited by the tail field. Double-click Tail field input field to open the numeric keypad and set the tail field. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Tail Field, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the tail field.

e. Error field: Triggers on the error field.

(9) Start Frame Bit

Click Start Frame Bit input field to open the numeric keypad and enter the start frame bit. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Start Frame Bit, then use the Multipurpose rotary knob to adjust the start frame bit. Range: 1 to 32.

(10) Synchronization Field

Click Synchronization Field input field to open the numeric keypad and enter the synchronization field. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Synchronization Field, then use the Multipurpose rotary knob to adjust the synchronization field. Range: 0 to 32.

(11) Middle Field 1

Click Middle Field 1 input field to open the numeric keypad and enter the middle field 1. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select

the Middle Field 1, then use the Multipurpose rotary knob to adjust the middle field 1. Range: 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 5.8 Parameter Setting. Alternatively, select the Header Field, then use the Multipurpose rotary knob to adjust the header field. Range: 0 to 32.

(13) Middle Field 2

Click Middle Field 2 input field to open the numeric keypad and enter the middle field 1. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Middle Field 2 then use the Multipurpose rotary knob to adjust the middle field 2. Range: 0 to 32.

(14) Word Count

Click Word Count input field to open the numeric keypad and set the word value. or details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Word Count, then use the Multipurpose rotary knob to adjust the word value. Range: 1 to 255.

(15) Word Bit

Click Word Bit input field to open the numeric keypad and set the word bit. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Word Bit, then use the Multipurpose rotary knob to adjust the word bit. Range: 1 to 8.

(16) Middle Field 3

Click Middle Field 3 input field to open the numeric keypad and enter the middle field 1. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Middle Field 3 then use the Multipurpose rotary knob to adjust the middle field 3. Range: 0 to 32.

(17) Tail Field

When the tail field is triggered, the setting of this parameter is applied. Click Tail Field input field to open the numeric keypad and set the tail field. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Tail Field, then use the Multipurpose rotary knob to adjust the tail field. Range: 0 to 32.

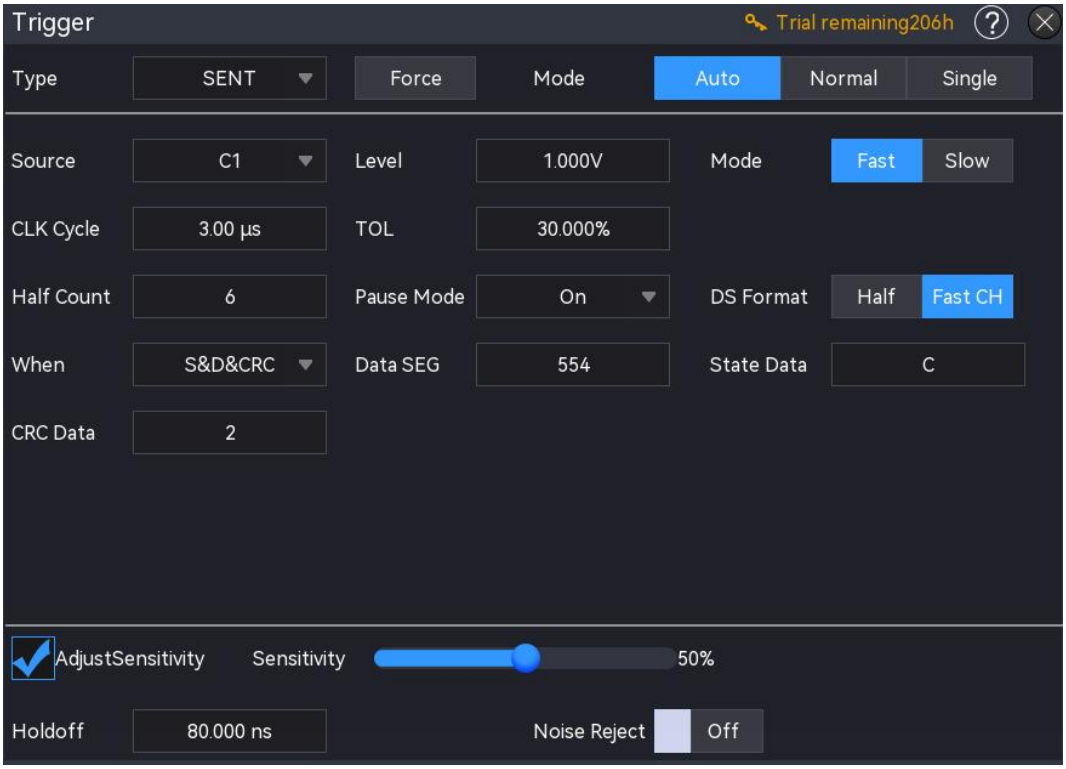
(18) Inter-frame Space

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 5.8 Parameter Setting. Alternatively, select the Inter-frame Space, then use the Multipurpose rotary knob to adjust the inter-frame space. Range: 0 to 32.

8.24. SENT Triggering

(1) Trigger Type

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



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 enter the trigger level. The trigger level can also be adjusted using Multipurpose or trigger Position rotary knob.

(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 5.8 Parameter Setting. Alternatively, select the Clock Period, then use the Multipurpose rotary knob to adjust the clock period. Range: 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 5.8 Parameter Setting. Alternatively, select the Half Byte, then use the Multipurpose rotary knob to adjust the half byte. Range: 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 corresponding 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 5.8 Parameter Setting. Alternatively, select the State Data, then use the Multipurpose rotary knob to adjust the state data.

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 5.8 Parameter Setting. Alternatively, select the Half Byte, then use the Multipurpose rotary knob to adjust the half byte. Range: 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 5.8 Parameter Setting. Alternatively, select the Data Field, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data field.
- 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 corresponding 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 5.8 Parameter Setting. Alternatively, select the CRC Data, then use the Multipurpose rotary knob to adjust the CRC data.

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 *State* and *Data* under Trigger Condition described 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, For the setting of each parameter, refer to *State, Data,* and *CRC Data* under Trigger Condition described above.

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

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

(10) Frame Type

When the mode is set to Slow Mode and the slow trigger condition is set to Enhanced, the Frame Type can be configured.

Click Frame Type to select 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 corresponding 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 5.8 Parameter Setting. Alternatively, select the Short ID, then use the Multipurpose rotary knob to adjust the short ID.

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

When the short data is selected, the corresponding 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 5.8 Parameter Setting. Alternatively, select the Short Data, then use the Multipurpose rotary knob to adjust the short data.

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

When the short CRC is selected, the corresponding 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 5.8 Parameter Setting. Alternatively, select the Short CRC, then use the Multipurpose rotary knob to adjust the short CRC.

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 *Short ID* and *Short Data* under Trigger Condition described above.

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

When the enhanced ID is selected, the corresponding 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 5.8 Parameter Setting. Alternatively, select the Enhanced ID, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the enhanced ID.

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

When the enhanced data is selected, the corresponding 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 5.8 Parameter Setting. Alternatively, select the Enhanced Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the enhanced data.

h. Enhanced CRC: Triggers when the enhanced data condition is met.

When the enhanced CRC is selected, the corresponding 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 5.8 Parameter Setting. Alternatively, select the Enhanced CRC, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the enhanced CRC.

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.

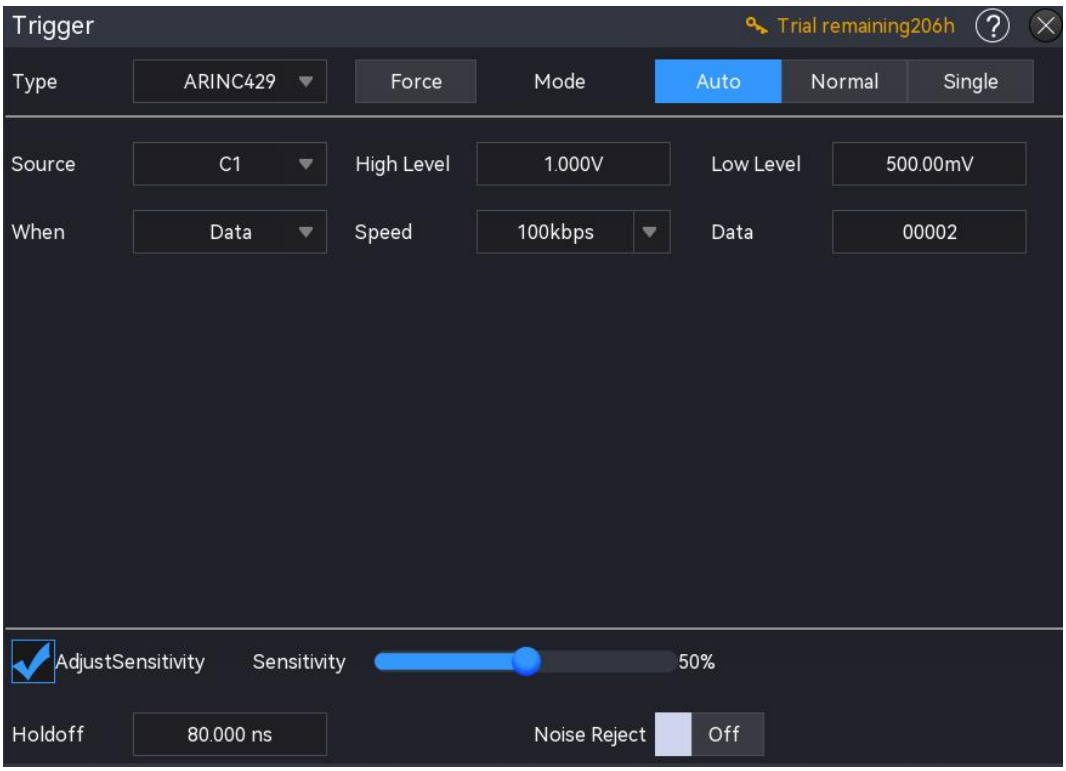
For the setting of each parameter, refer to *Enhanced ID* and *Enhanced data* under Trigger Condition described above.

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

8.25. ARINC429 Triggering

(1) Trigger Type

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



(2) Trigger Source

Click Source to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System.

The current source is displayed in the trigger label on 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 or Low input field, then use the Multipurpose rotary knob, or numeric keypad to set the trigger level. For more details on *Trigger Level,* refer to the section Noun Explanation of Triggering System. The selected trigger level is displayed as a solid line on the screen.

(4) Speed

Click Speed to set the transmission rate to high (100 kb/s), low (12.5 kb/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 corresponding menus 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 5.8 Parameter Setting.

  Alternatively, select the Label, use the arrow keys ←, → to move the cursor, and then

use the Multipurpose rotary knob to adjust the label. Range: 00 – FF.

d. SDI: Triggers when the specified SDI occurs.

When the SDI is selected, the corresponding menus 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 5.8 Parameter Setting. Alternatively, select the SDI, then use the Multipurpose rotary knob to adjust the SDI. Range: 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 corresponding menus 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 5.8 Parameter Setting. Alternatively, select the Data, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data. Range: 00000 - 7FFFF.

f. SSM: Triggers when the Symbol State Matrix matches the custom Symbol State Matrix.

When the SSM is selected, the corresponding menus 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 5.8 Parameter Setting. Alternatively, select the SSM, then use the Multipurpose rotary knob to adjust the SSM. Range: 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 *Address, SDI, Data*, and *SSM* under Trigger Condition described 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. Area 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 UPO6000HD series adds an area 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.



Area 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 - C8.

(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 - C8.

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 - C8.

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, select the corresponding boundary using Multipurpose rotary knob to adjust its values.

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

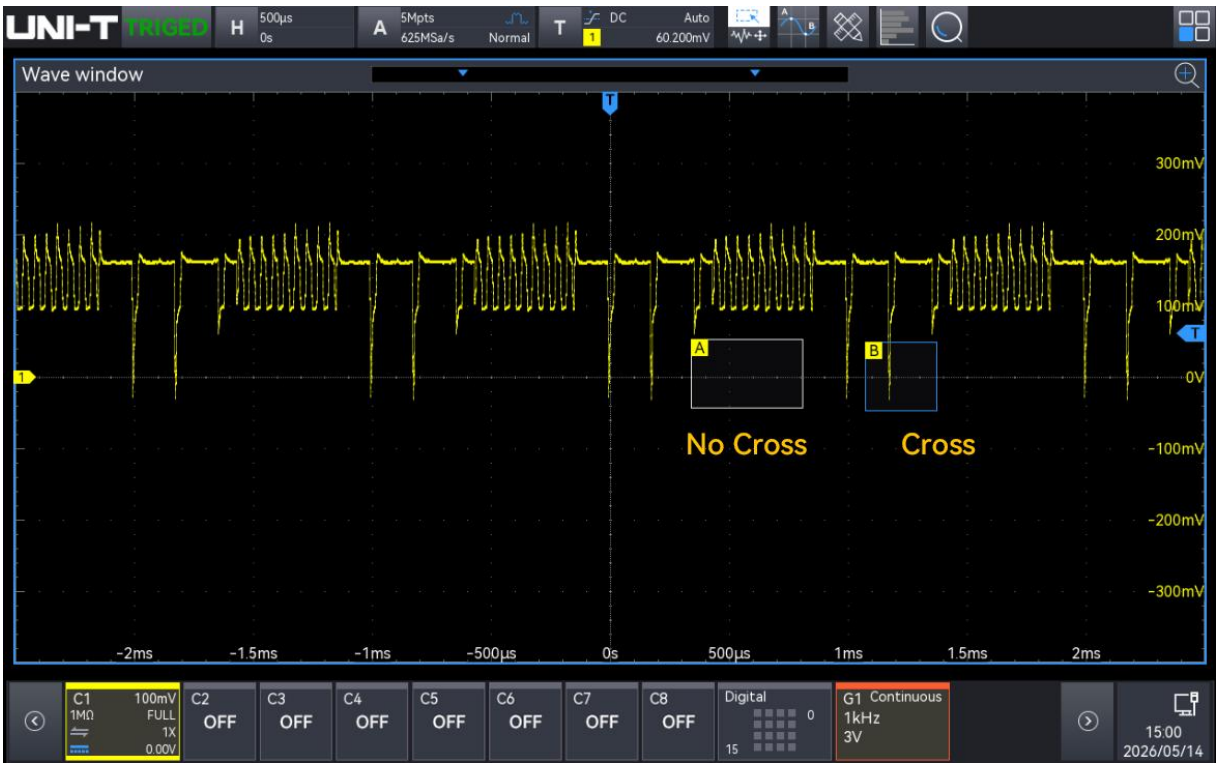
b. Bx0, Bx1, By0, By1 represent the left, right, top, and bottom boundaries of Area B, respectively.

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 area trigger boxes expand or compress with the waveform accordingly.

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

- RS232 Decoding
- I²C Decoding
- SPI Decoding
- CAN Decoding
- CAN-FD Decoding
- LIN Decoding
- FlexRay Decoding
- Audio Decoding
- 1553B Decoding
- Manchester Decoding
- SENT Decoding
- ARINC429 Decoding

Users can efficiently locate errors, debug hardware, and accelerate development by using protocol decoding, ensuring high-speed and high-quality project completion. The UPO6000HD provides four bus decoder modules (Decode 1, Decode 2, Decode 3, and Decode 4) for decoding common protocols for analog channel input signals.

The UPO6000HD series supports protocol decoding for RS232, I^2C, SPI, and optionally for CAN, CAN-FD, LIN, FlexRay, Audio, 1553B, Manchester, SENT, 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 methods.

- Press the Bus button on the front panel to open the decoding setting menu.
- Click Home icon in the top-right corner of the screen, then 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.

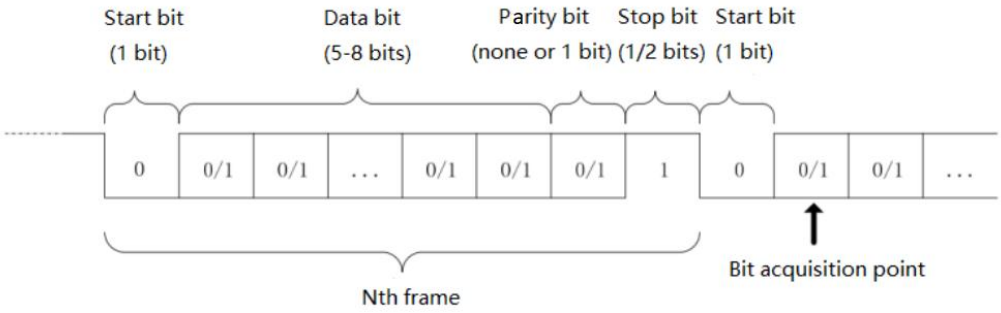
9.1. RS232 Decoding

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.

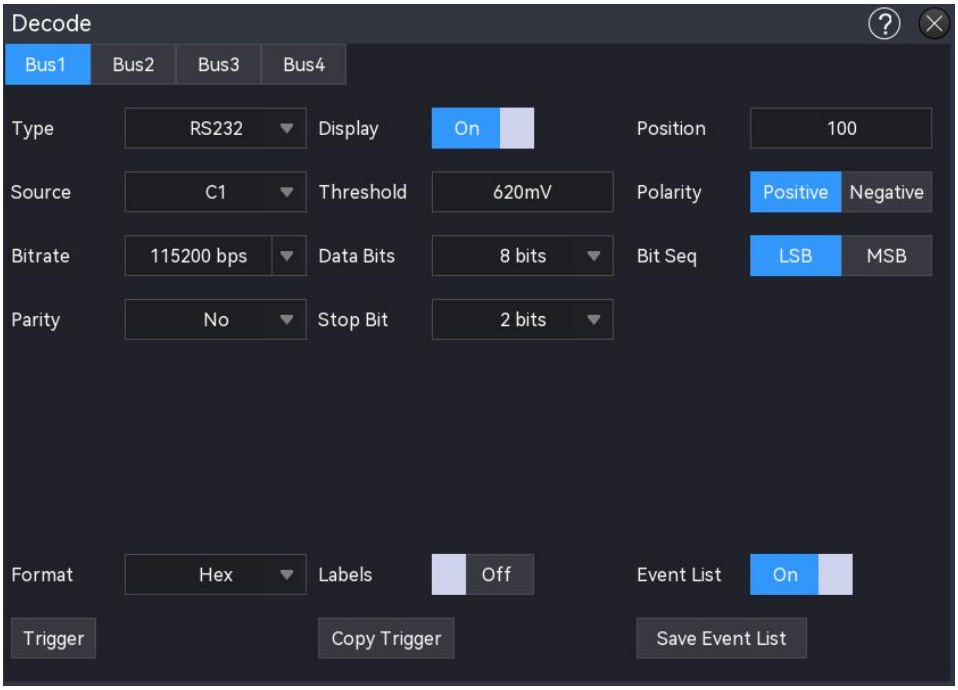
Send one start bit first, and send 5-8 data bits, and send optional parity check bit, and send one or two stop bits at last. The number of data bits is agreed by both communicating parties, it can be 5-8 bits, with no parity check bit, odd parity check bit, or even parity check bit. The stop bit can 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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has a connected signal.

c. Threshold

Click Threshold input field, then use the Multipurpose rotary knob to adjust the threshold. Alternatively, double-click Threshold input field to open the virtual numeric keypad and enter the threshold. The threshold range depends on the vertical scale and vertical offset of the signal source.

d. Polarity

- 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 for data transmission. Click Parity to select none, even, or odd.

f. 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.

g. Bit Sequence

Set the data bit sequence for RS232 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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.

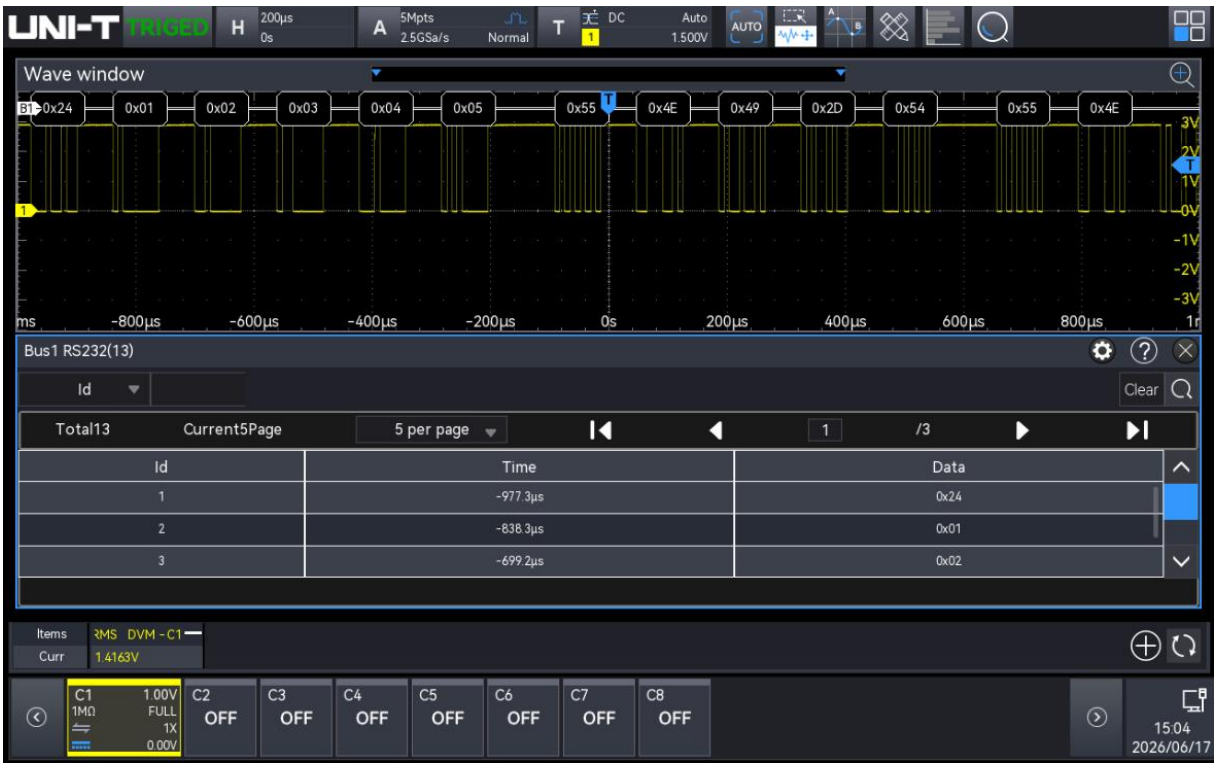
OFF: The decoding bus label is hidden.

(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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(5) Jump to Trigger Menu

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

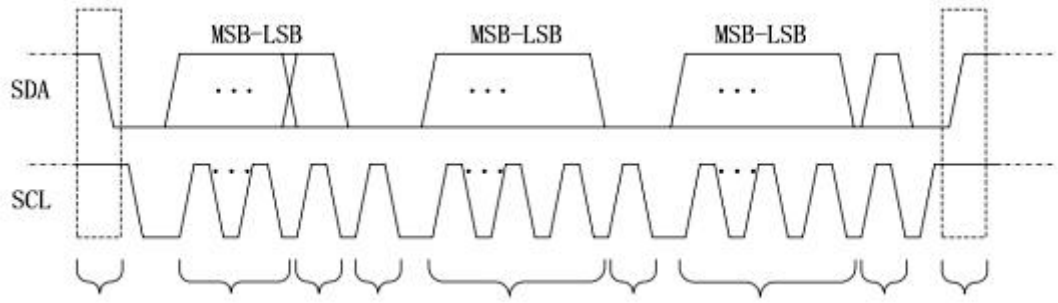
(6) Copy Trigger Settings

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.

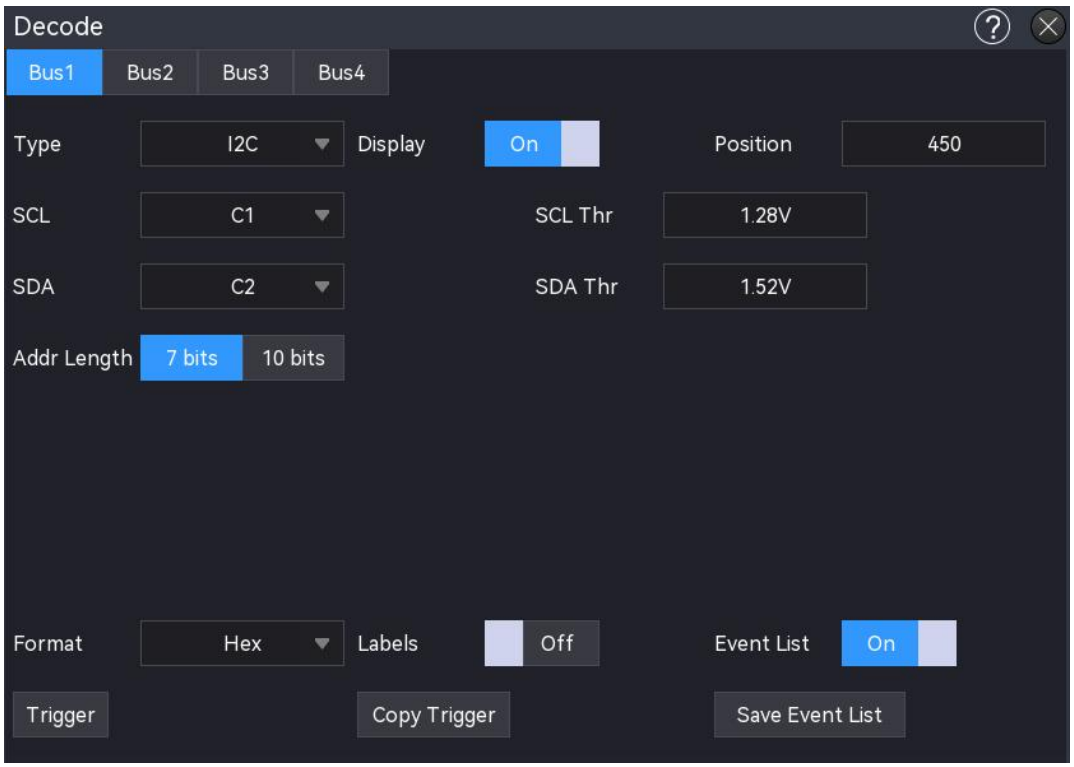
This bus is the bus of multiple masters, preventing data corruption through conflict demodulation and arbitration mechanisms. It should be noted that the I^2C bus supports two address bit width: 7 bits and 10 bits. 10 bits and 7 bits address are compatible and can be used together. SCL and SDA on the 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 I^2C and configure the decoding settings.



b. Source Setting

Set both the clock source and the data source. Decoded data is available only when the signal source is set to a channel that has a connected signal.

- Clock Source

  Click SCL to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

- Data Source

  Click SDA to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

c. Threshold

Double-click SCL Threshold or SDA Threshold input field to open the numeric keypad and set the threshold. Alternatively, click SCL Threshold or SDA Threshold input field, then use the Multipurpose rotary knob to adjust 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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.

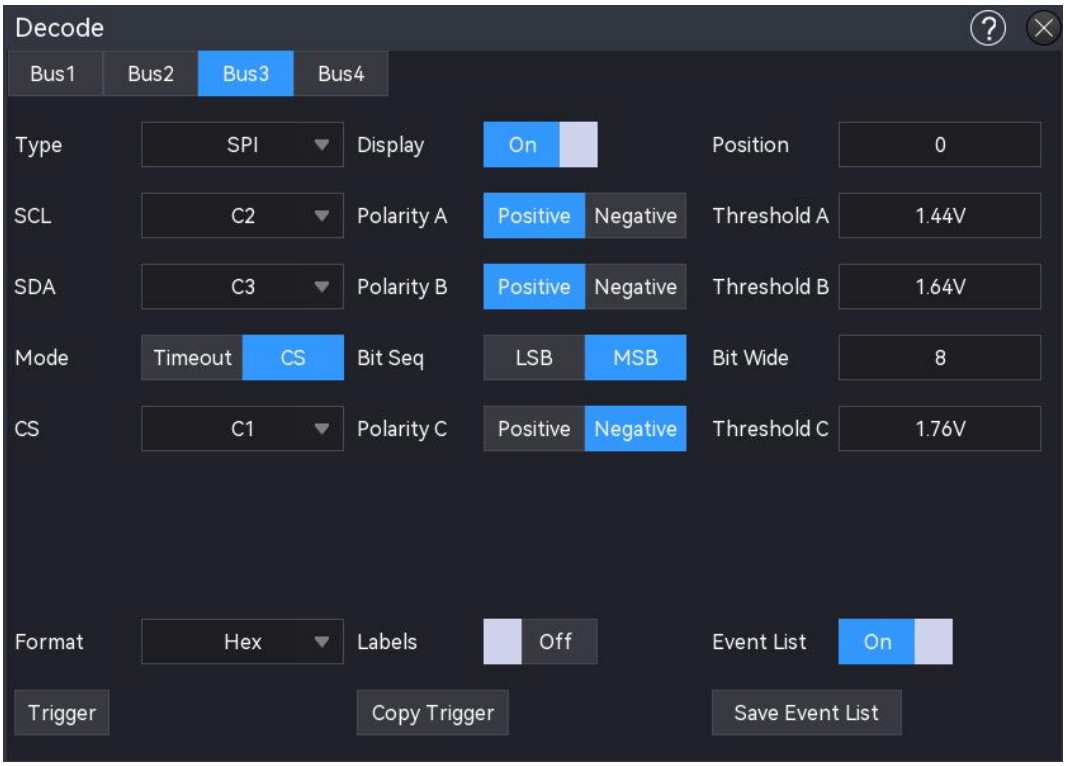
CS: Chip select used to enable the slave device.

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 (Chip Select) source. Decoded data is available only when the signal source is set to a channel that has a connected signal.

- Clock Source

  Click SCL to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

- Data Source

  Click SDA to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the

top-right corner of the screen.

- Chip Select Source

  Click CS to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

c. Edge Setting

- 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.

- CS Polarity

  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.

- 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.

d. Threshold

Double-click Threshold A, Threshold B, or Threshold C input field to open the numeric keypad and set the threshold. Alternatively, click Threshold A, Threshold B, or Threshold C input field, then use the Multipurpose rotary knob to adjust the threshold.

e. Mode

Click Mode to select SPI mode.

- 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.

f. Bit Sequence

Set the data bit sequence for SPI 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.

g. Data 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 5.8 Parameter Setting. Alternatively, select the Bit Width, then use the Multipurpose rotary knob to adjust the bit width. The setting range is from 4 to 32.

h. 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 5.8 Parameter Setting. Alternatively, select the Timeout, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the timeout. The setting range is from 3.2ns 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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.

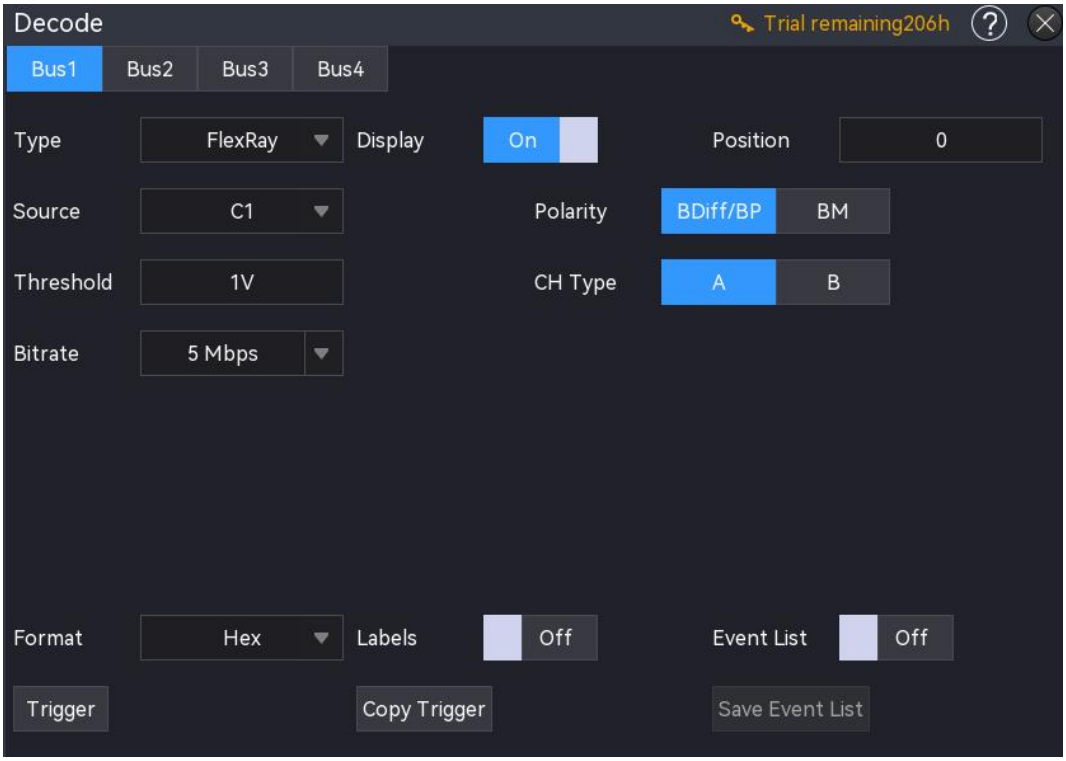
9.4. FlexRay Decoding

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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has 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.

Alternatively, click Threshold input field, then use the Multipurpose rotary knob to adjust 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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

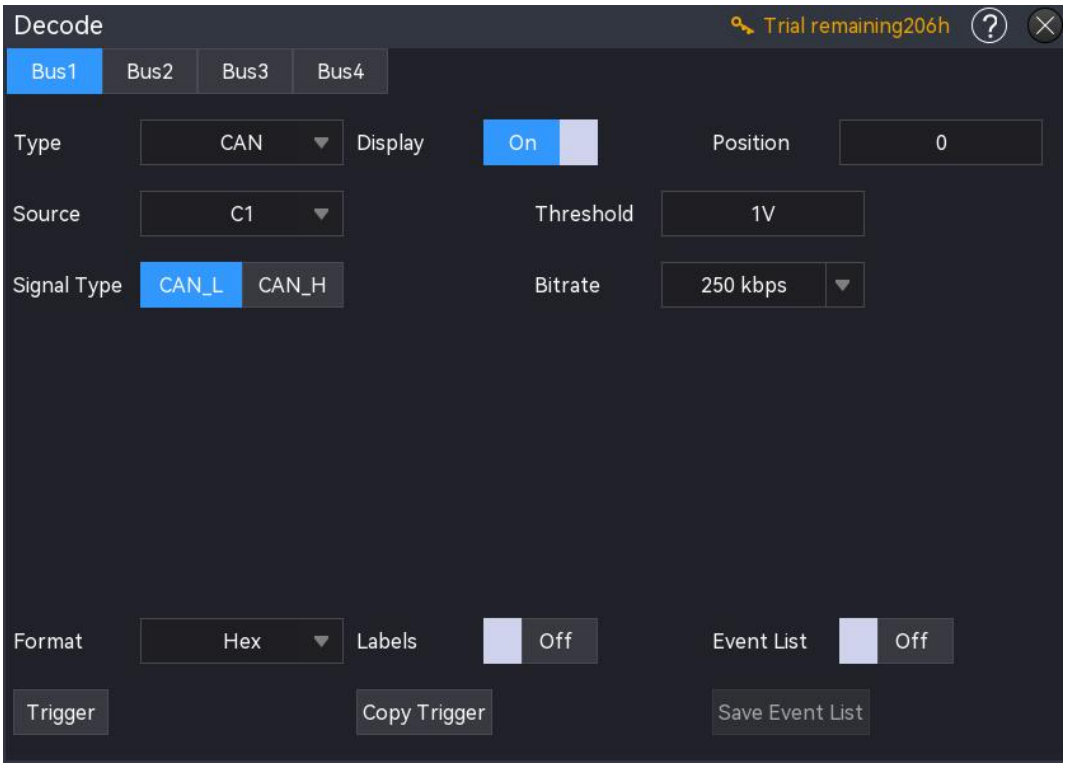
When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has a connected signal.

c. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold. Alternatively, click Threshold input field, then use the Multipurpose rotary knob to adjust 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, 115.2kbps, 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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

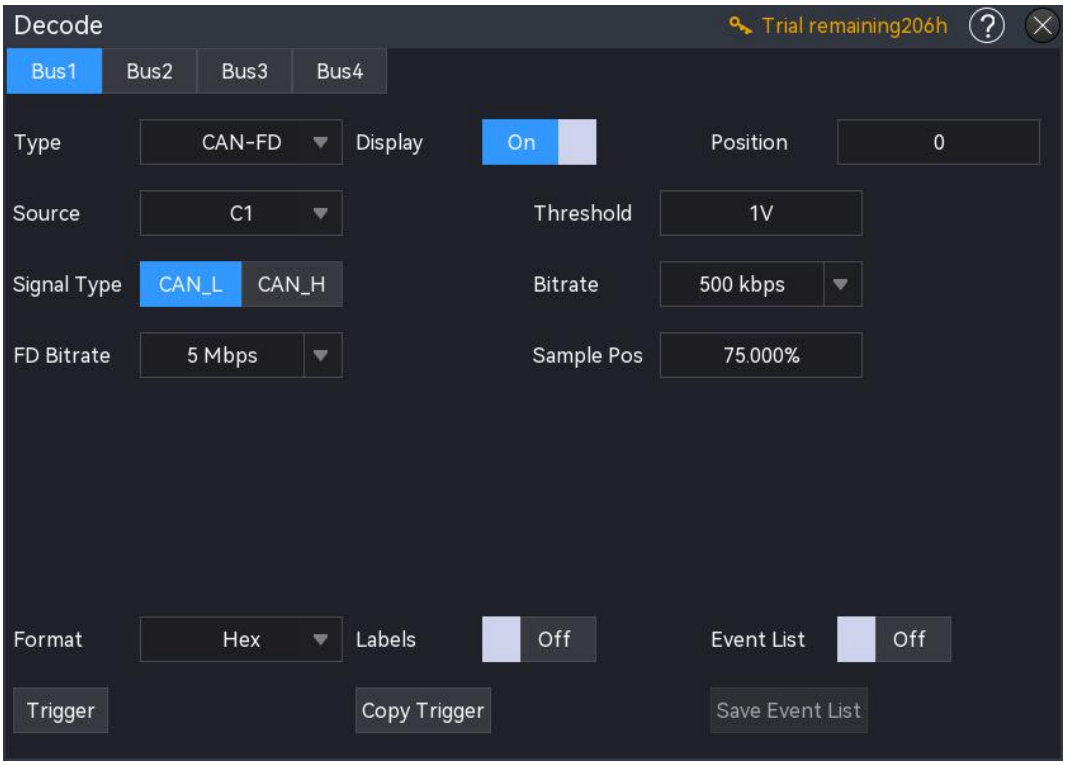
When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has a connected signal.

c. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold. Alternatively, click Threshold input field, then use the Multipurpose rotary knob to adjust 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.

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

g. Sample 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 Sample Position input field to open the numeric keypad and set the sampling position. The sample position can also be adjusted using the Multipurpose rotary knob. Range: 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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

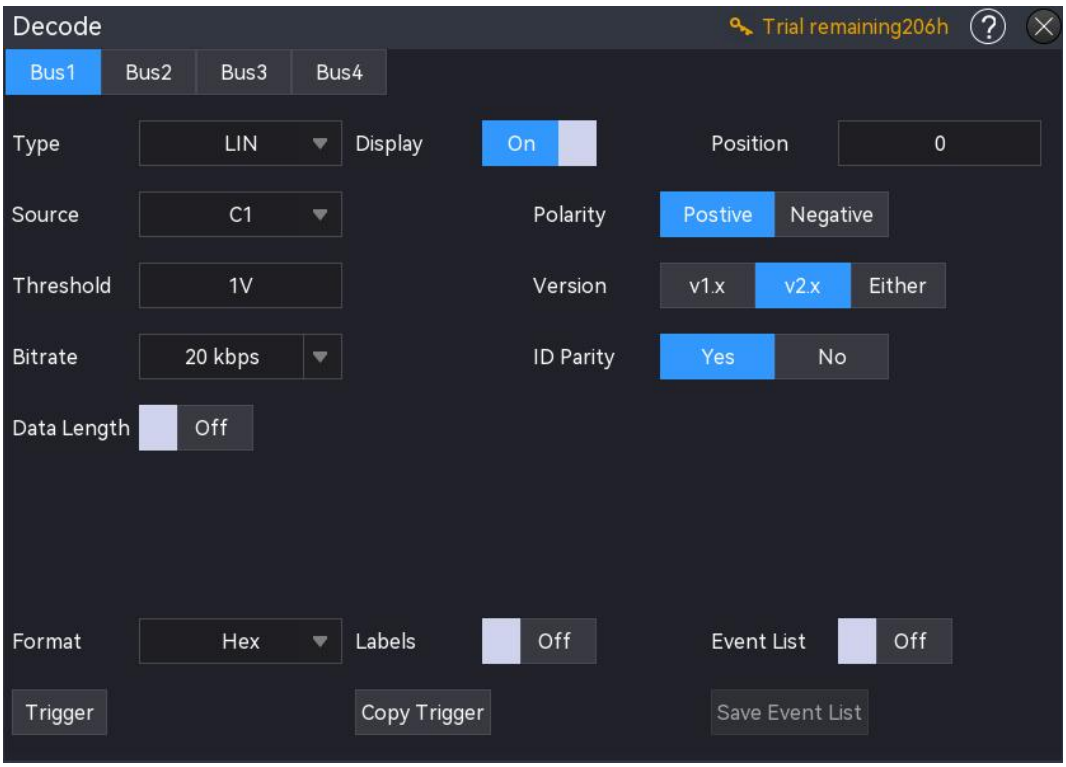
When the trigger type and parameter settings are completed, and the decoding type is the same, click Copy Trigger on the decoding page to apply the trigger parameter settings to the decoding menu.

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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has 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. Alternatively, click Threshold input field, then use the Multipurpose rotary knob to adjust 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.

e. Version

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

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 5.8 Parameter Setting. Alternatively, select the Data Length, use the arrow keys ←, → to move the cursor, and then use the Multipurpose rotary knob to adjust the data length. 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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.

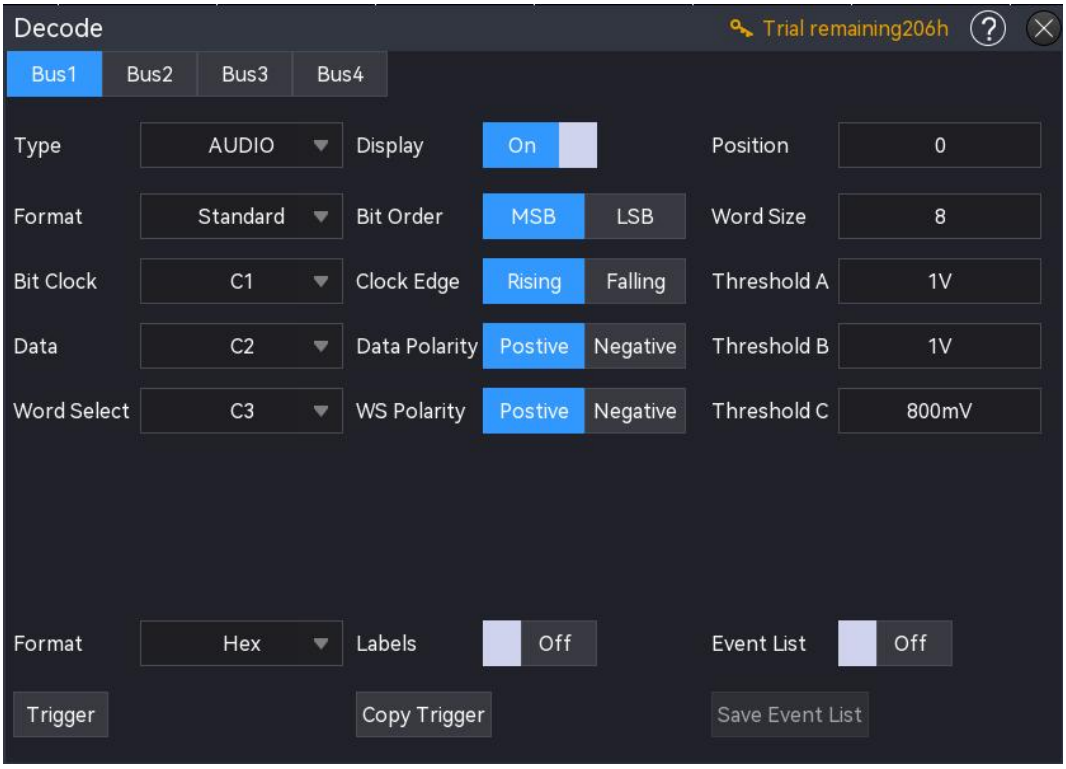
9.8. Audio Decoding

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

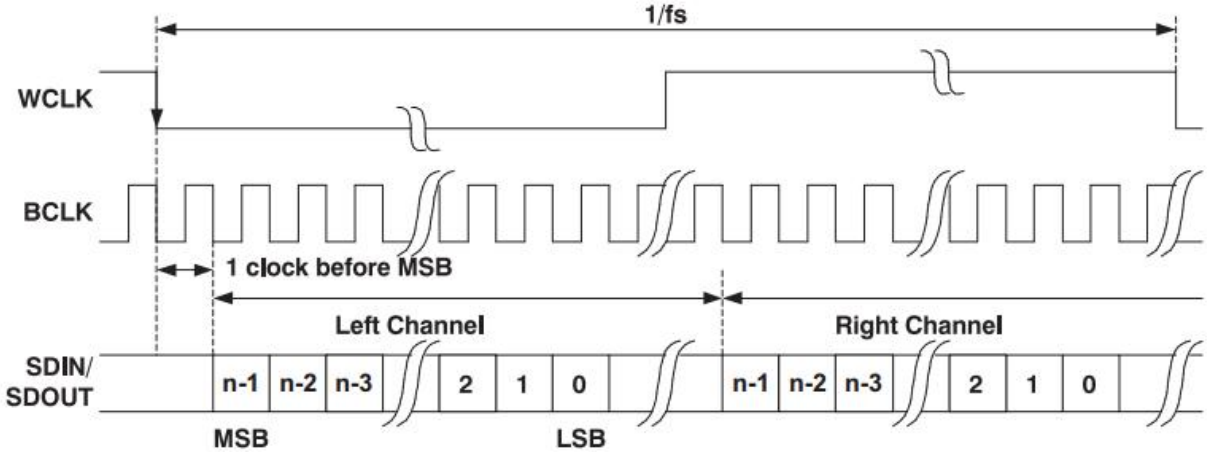
Click Protocol Type to select Audio and configure the decoding settings.



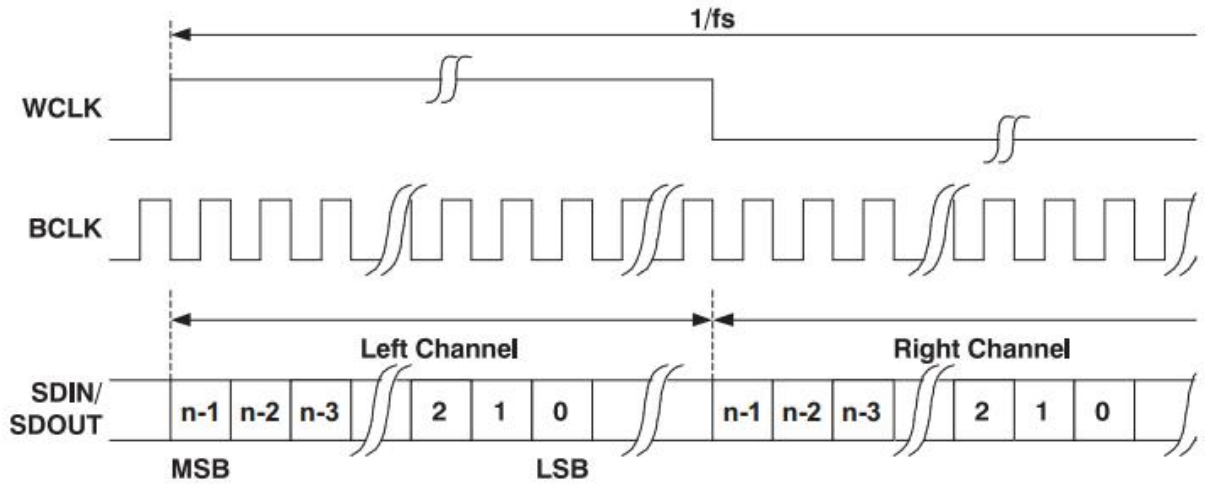
b. Format

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

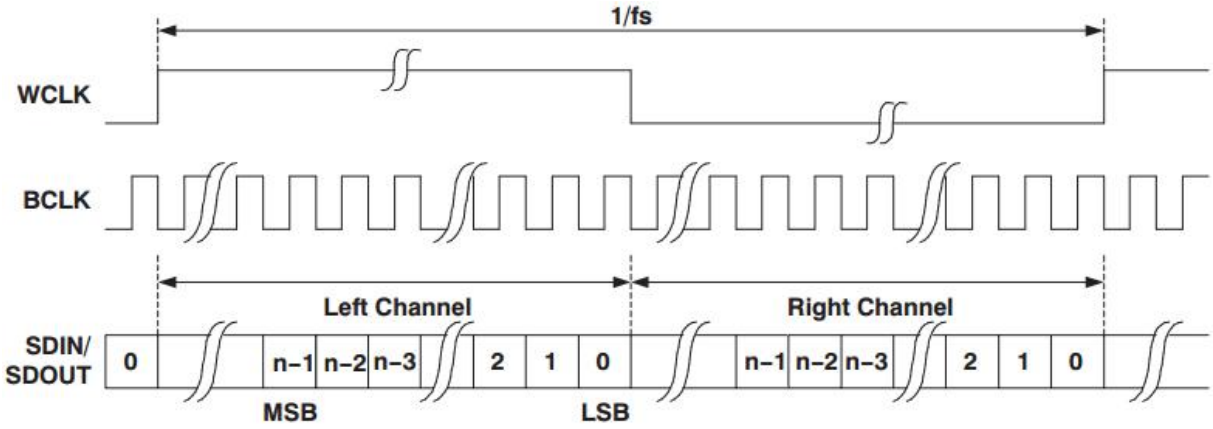
- Standard: MSB of each sampled data is sent first and the LSB is sent last. MSB is displayed on the SDATA line, which at one clock bit clock after the edge of WS transition.



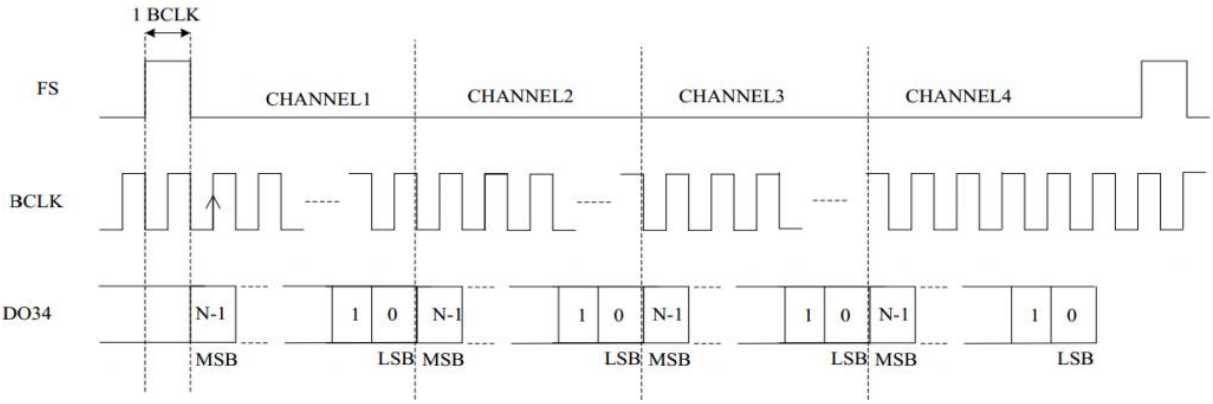
- Left justifying: Data transmission (MSB first) starts at the edge of the WS conversion (without the one-bit delay used by the standard format).



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



- TDM: Time Division Multiplexing mode can transmit multi-channel data.



c. Bit Sequence

Click Bit Sequence to select MSB or LSB. The default setting is MSB.

- 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.

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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

  The clock line (SCLK) provides the clock signal for synchronizing audio data transmission.

- Word Select

  Click Word Select to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen. The bit select indicates the audio data of the current transmission is left channel or right channel.

- Data

  Click Data select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen. The data line is used to transmit the actual audio data.

- Frame Synchronization

  Click Frame Synchronization to select C1 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner 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-D input field to open the numeric keypad and set the threshold.

Alternatively, click Threshold A-D input field, then use the Multipurpose rotary knob to adjust the threshold.

g. Word Size

Double-click Word Size to open the numeric keypad and set the word size. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Word Size, then use the Multipurpose rotary knob to adjust the word size. The setting range is from 4 to 32.

h. 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 5.8 Parameter Setting. Alternatively, select the Word Size, then use the Multipurpose rotary knob to adjust this value. The setting range is from 4 to 32.

Data bits per channel ≤ Clock bits per channel.

i. 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 5.8 Parameter Setting. Alternatively, select the Clock Bit per Channel, then use the Multipurpose rotary knob to adjust this value. The setting range is from 4 to 32.

j. 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 5.8 Parameter Setting. Alternatively, select the Channel Number per Frame, then use the Multipurpose rotary knob to adjust this value. The setting range is from 2 to 64.

k. 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 5.8 Parameter Setting. Alternatively, select the Bit Delay, then use the Multipurpose rotary knob to adjust the data bit. The setting range is from 0 to 31.

Bit delay setting < Clock bits per 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line

to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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.

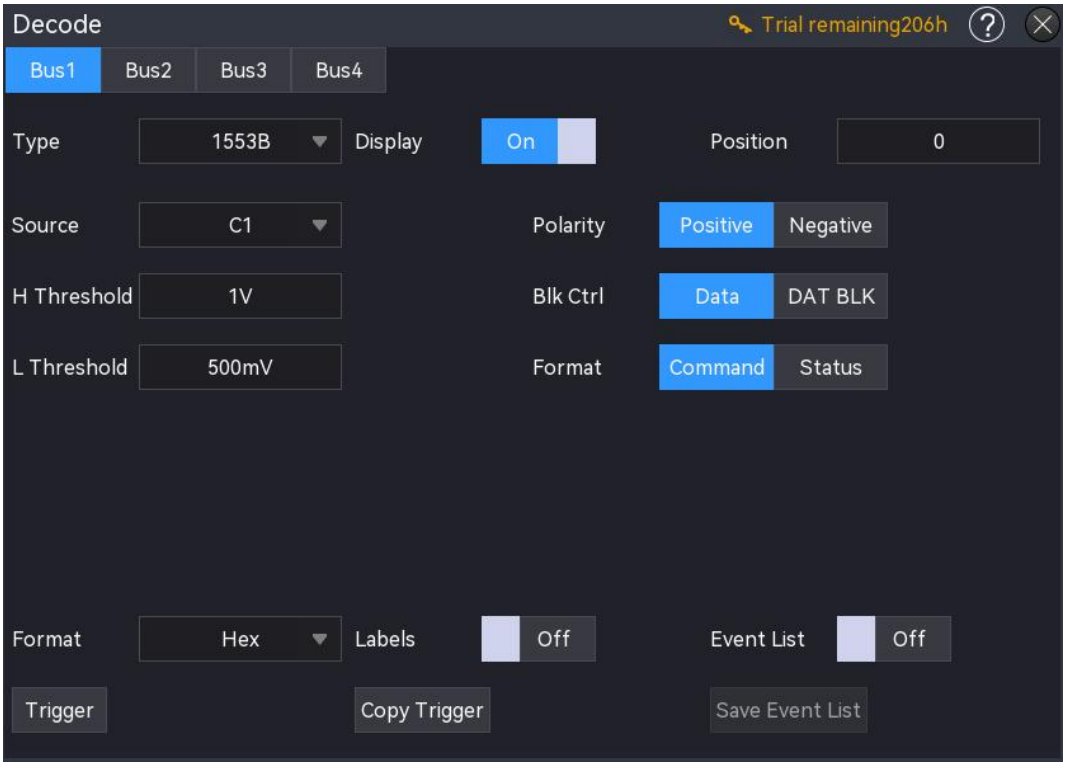
9.9. 1553B Decoding

The oscilloscope samples the 1553B signal and determines each data point as logic 1 or logic 0 based on the set threshold level. 1553B decoding requires specifying the data channel source and the threshold level.

(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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has a connected signal.

c. Polarity

Click Polarity to select the positive or negative. The default setting is positive.

d. High/Low Level

Click High or Low input field, then use the Multipurpose rotary knob to adjust the threshold. Alternatively, double-click Threshold input field to open the virtual numeric keypad and enter the threshold. The threshold range depends on the vertical scale and vertical offset of the signal source.

e. Block Control

Click Block Control to select data or data block for decoding. The default setting is data.

f. Format

Click Format to set command word or state word.

If the format is set to command word, it decodes the command word.

If the format is set to state word, it decodes the state word.

(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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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

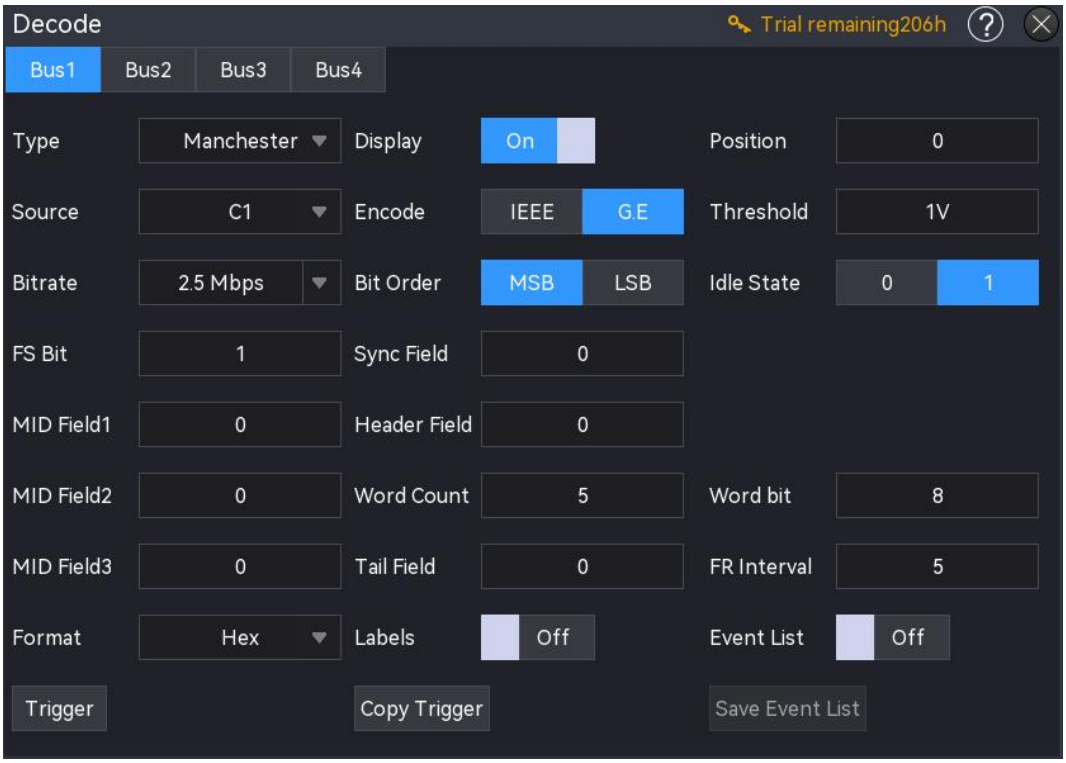
cycles and without conversion levels, thus preventing loss of clock synchronization or analogue link bit errors due to low frequency shifts with poor compensation. In this technique, the actual binary data transmitted through the cable is not sent as a sequence of logical 1's or 0's (technically known 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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has a connected signal.

c. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold. Alternatively, click Threshold input field, then use the Multipurpose rotary knob to adjust 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, 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 bitrate must match with the DUT. The default bitrate is 1.2 kbps.

f. Bit Sequence

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.

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 5.8 Parameter Setting. Alternatively, select the Start Frame Bit, then use the Multipurpose rotary knob to adjust the start frame bit. The setting range is 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 5.8 Parameter Setting. Alternatively, select the Synchronization Field, then use the Multipurpose rotary knob to adjust the synchronization field. 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 5.8 Parameter Setting. Alternatively, select the Middle Field 1, then use the Multipurpose rotary knob to adjust the middle field 1. 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 5.8 Parameter Setting. Alternatively, select the Header Field, then use the Multipurpose rotary knob to adjust the header field header field. 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 5.8 Parameter Setting. Alternatively, select Middle Field 2, then use the Multipurpose rotary knob to adjust the middle field 2. The setting range is from 0 to 32.

m. Word Count

Click Word Count input field to open the numeric keypad and set the word count. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Word Count, then use the Multipurpose rotary knob to adjust the word count. The setting range is from 1 to 255.

n. Word Bit

Click Word Bit input field to open the numeric keypad and set the word bit. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Word Bit, then use the Multipurpose rotary knob to adjust the word bit. The setting range is from 1 to 8.

o. Middle Field 3

Click Middle Field 33 input field to open the numeric keypad and set the middle field 3. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select Middle Field 3, then use the Multipurpose rotary knob to adjust the middle field 2. The setting range is from 0 to 32.

p. Tail Field

Click Tail Field input field to open the numeric keypad and set the tail field. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Tail Field, then use the Multipurpose rotary knob to adjust the tail field.

q. Inter-frame Space

Click Inter-frame Space input field to open the numeric keypad and enter the inter-frame space. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Inter-frame Space, then use the Multipurpose rotary knob to adjust the inter-frame space. 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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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

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 (only

when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(5) Jump to Trigger Menu

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

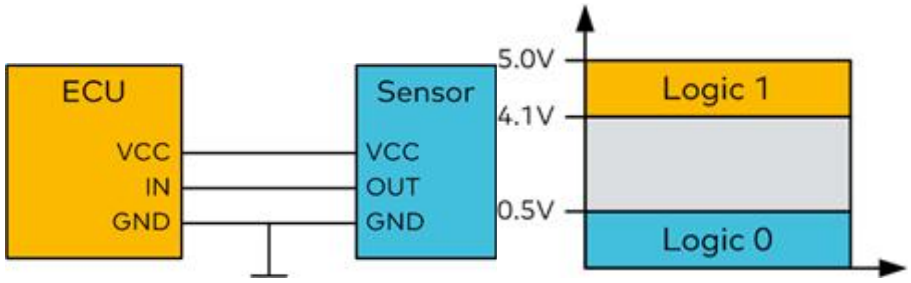
(6) Copy Trigger Settings

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.

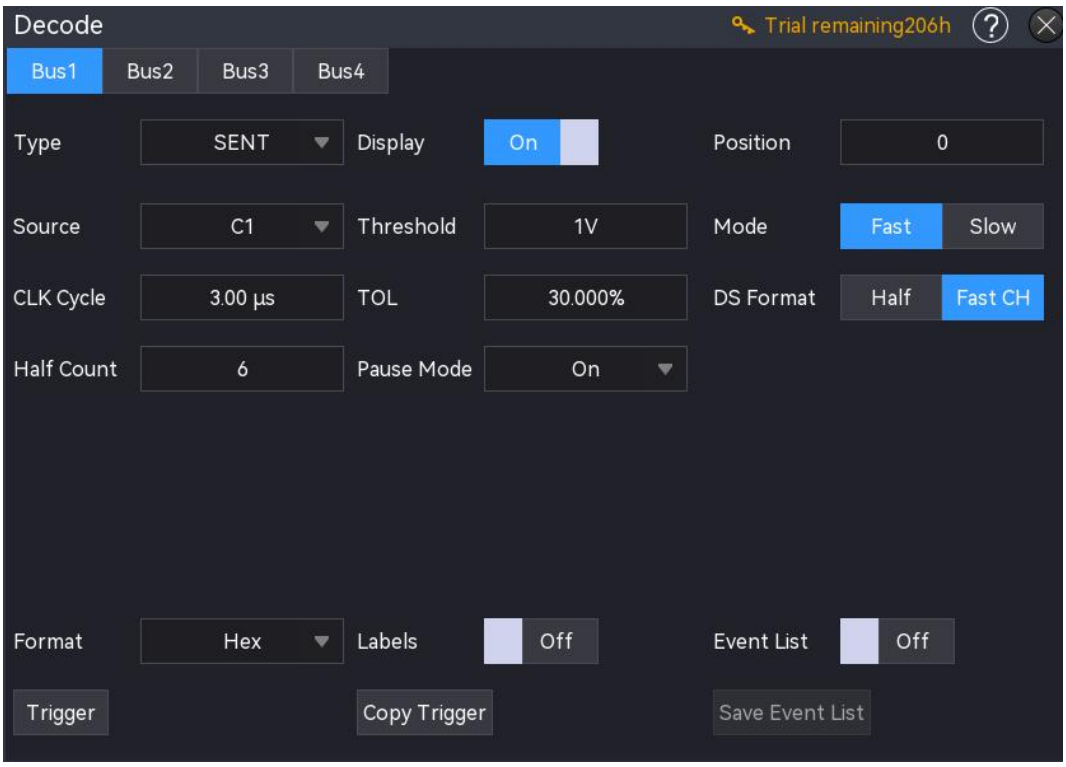


The data of the SENT protocol is coded using a half-byte nibble, i.e. 4 bits, and a half-byte nibble is defined by the time difference between two falling edges.

(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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has a connected signal.

c. Threshold

Double-click Threshold input field to open the numeric keypad and set the threshold. Alternatively, click Threshold input field, then use the Multipurpose rotary knob to adjust 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. Alternatively, select the Clock Period, then use the Multipurpose rotary knob to adjust the clock period. 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. Alternatively, select the Half Byte, then use Multipurpose rotary knob to adjust the half byte.

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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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.

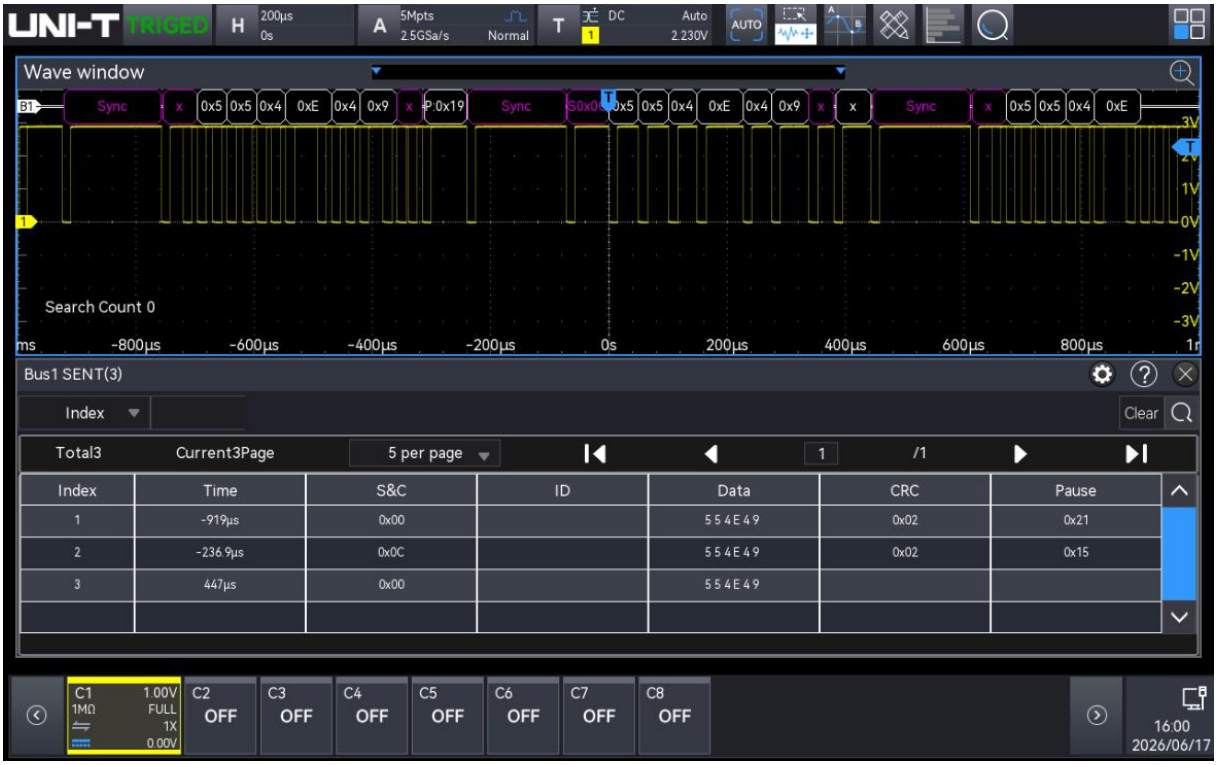
OFF: The decoding bus label is hidden.

(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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(5) Jump to Trigger Menu

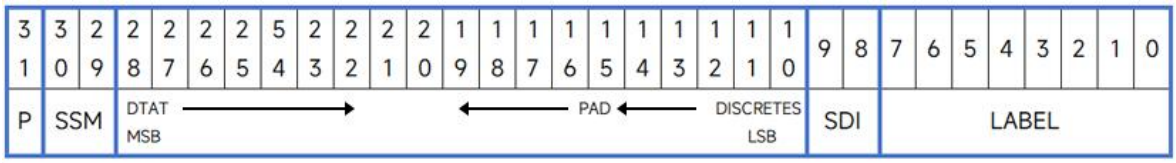
Click Trigger to directly navigate to the trigger menu, where the trigger type is same as the decoding type.

(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

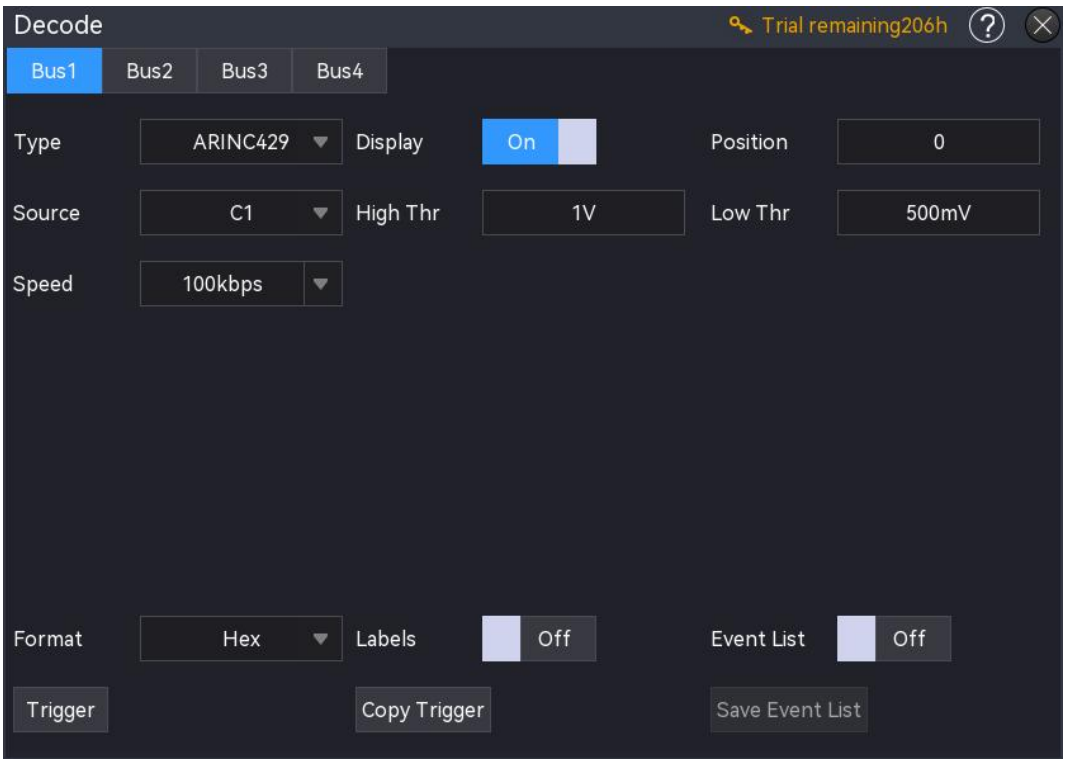
The ARINC 429 bus is a serial standard, interface-oriented, unidirectional broadcast transmission bus.



(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 - C8. For more details on *Trigger Source,* refer to the section Noun Explanation of Triggering System. The current source is displayed in the top-right corner of the screen.

Decoded data is available only when the signal source is set to a channel that has a connected signal.

c. High/Low Threshold

Double-click High or Low Threshold input field to open the virtual numeric keypad and enter the threshold. The threshold range depends on the vertical scale and vertical offset of the signal source.

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 the decoding bus on the screen. Use the Multipurpose rotary knob to adjust the position of the decoding line. Alternatively, double-click Decoding Line to open the numeric keypad and enter the position value. The range can be set from 0 to 546.

c. Format

Set the display format of the decoding bus and the format of the decoded content in the event list. 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.

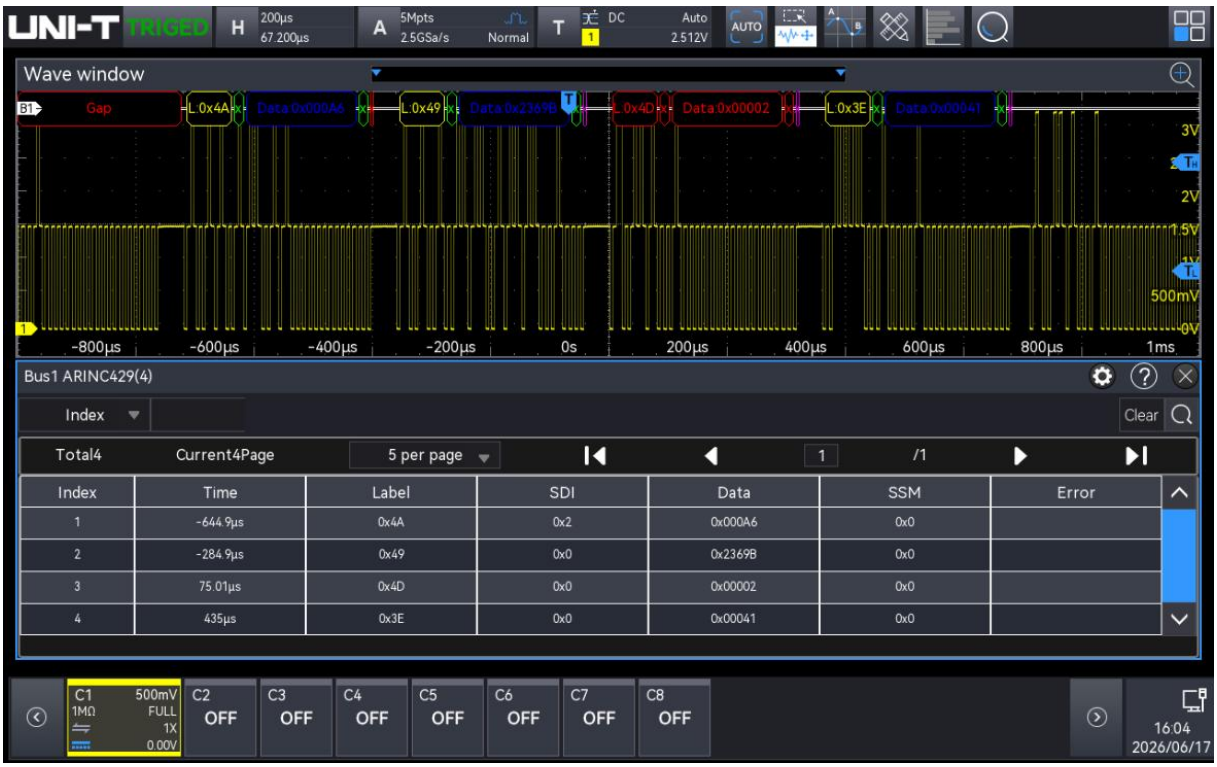
OFF: The decoding bus label is hidden.

(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

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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

(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

- Parameter Measurement Overview
- Frequency Counter
- Digital Voltmeter
- Parameter Snapshot
- Parameter Measurement
- Measurement Statistics
- Add Parameter
- Clear Added Measurement
- Global Setting

The UPO6000HD 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.

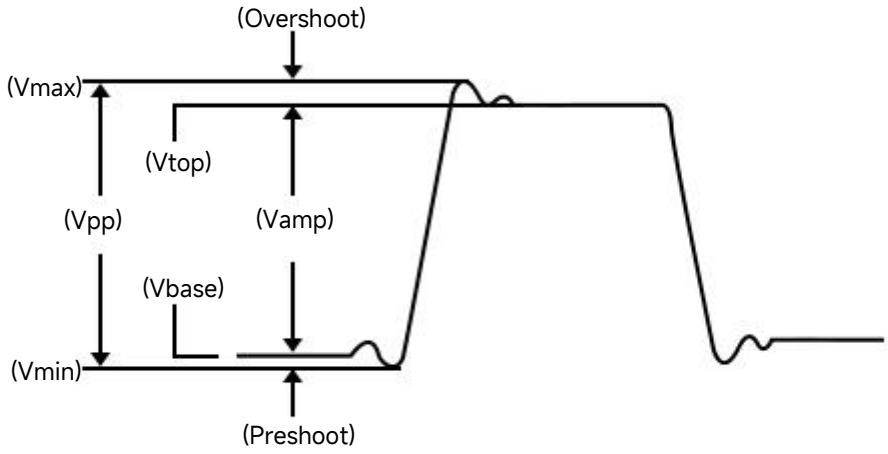
Access the Measure menu using the following methods.

- Press the Meas button on the front panel to open the measurement menu.
- Click Home icon in the top-right corner of the screen, then click measurement icon to enter the measurement menu.
- If the measurement function is added in the toolbar, click measurement icon in the top-right corner of the screen to enter the measurement menu.

10.1. Parameter Measurement Overview

UPO6000HD series oscilloscope can automatically measure 56 types of parameters, such as voltage, time, other parameters, and power analysis.

■ Voltage Parameters



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: The RMS value of the waveform after the DC component has been removed.

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.

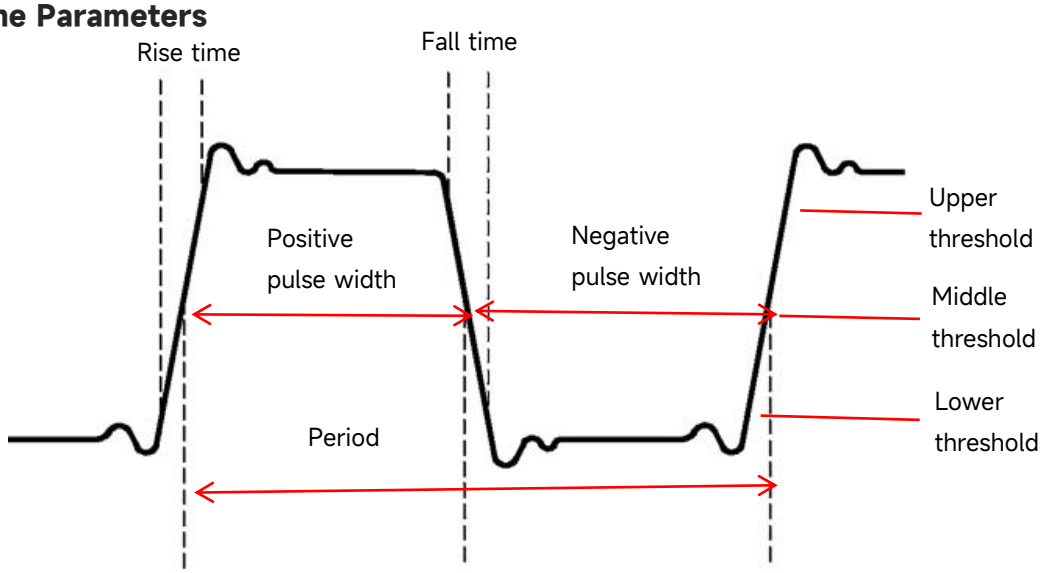
Positive overshoot: The nearest extreme point after the signal along the rising edge of the waveform crosses the upper threshold limit.

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 Parameters



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

  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.

- Turn On/Off Time

  Turn on time: The time it takes for the output voltage to rise to 90% of its steady-state value after the power supply is turned on.

  Turn off 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 analog channel.

Access the Counter menu using the following methods.

- Press the Meas button on the front panel, then click Counter in the measurement menu to open the counter menu.
- Press the Analyze button on the front panel, then click Counter in the analyze menu to open the counter menu.
- Click Home icon in the top-right corner of the screen, then click counter icon to open the counter menu.
- If the counter function is added in the toolbar, click counter icon in the top-right corner of the screen to open the counter menu.

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 - C8 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

Click Refresh Time input field, then use the Multipurpose rotary knob to set the time value. Alternatively, double-click Refresh Time input field to open the numeric keypad and enter the time value. The setting range is from 200 ms to 10 s.

e. Resolution

The display resolution of the counter measurement results can be set when the parameter type is Frequency or Period. Click Resolution input field, then use the Multipurpose rotary knob to set the resolution. Alternatively, double-click Resolution input field to open the numeric keypad and enter the resolution. The setting range 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 analog channel. DVM measurements are asynchronous to the oscilloscope's acquisition system and are always acquired.

Access the digital voltmeter measurement menu using the following methods.

- Press the Meas button on the front panel, then click Voltmeter in the measurement menu to open the digital voltmeter menu.
- Press the Analyze button on the front panel, then click Voltmeter in the analyze menu to open the digital voltmeter menu.
- Click Home icon in the top-right corner of the screen, then click voltmeter icon to open the digital voltmeter menu.
- If the counter function is added in the toolbar, click voltmeter icon in the top-right corner of the screen to open the digital voltmeter menu.

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 - C8 can be used as the trigger source. DVM measurement can be performed even if C1 - C8 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

Click Refresh Time input field, then use the Multipurpose rotary knob to set the time value. Alternatively, double-click Refresh Time input field to open the numeric keypad and enter the time value. The setting 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 Utility 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 Utility 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 5.8 Parameter Setting. Alternatively, select the Output Pulse Width, then use the Multipurpose rotary knob to adjust the output pulse width. The setting range is from 200 ns to 1.5 s.

c. Output Polarity

Click Output Polarity to select 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 trigger an alarm if the test result meets the condition, otherwise, the alarm will not be sound.

b. Limit Condition

- >: The oscilloscope will trigger an alarm if the DVM value is greater than the set lower limit, and the lower limit can be set.
- <: The oscilloscope will trigger an alarm if the DVM value is less than the set upper limit, and the upper limit can be set.
- < >: The oscilloscope will trigger 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 trigger 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.

- 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. Alternatively, use the Multipurpose rotary knob to adjust 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. Alternatively, use the Multipurpose rotary knob to adjust 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.

Access the parameter snapshot menu using the following method.

Press the Meas button on the front panel, then click Parameter Snapshot in the measurement menu to open the parameter snapshot menu.



In the parameter snapshot window, click Source to select the test signal (C1 - C8, M1 - M8). The color of the measured results matches the color of the corresponding source.

10.5. Parameter Measurement

Parameter Measurement: Toggles the display of the parameter measurement information bar. If unchecked, the information bar is hidden.

Access the parameter measurement menu using the following method.

- Press the Meas button on the front panel, then click Parameter Measurement in the measurement menu to open the parameter measurement.



Parameter measurements are displayed above the Volt/Div information box, showing two items: the measurement item and its current value.

In parameter measurement mode, the Counter and Voltmeter are displayed on the far left by default, followed by custom parameters. A maximum of 27 parameters can be configured for parameter measurement.

To remove a custom parameter from the parameter measurement bar, click the “–” icon at the upper right of the parameter.

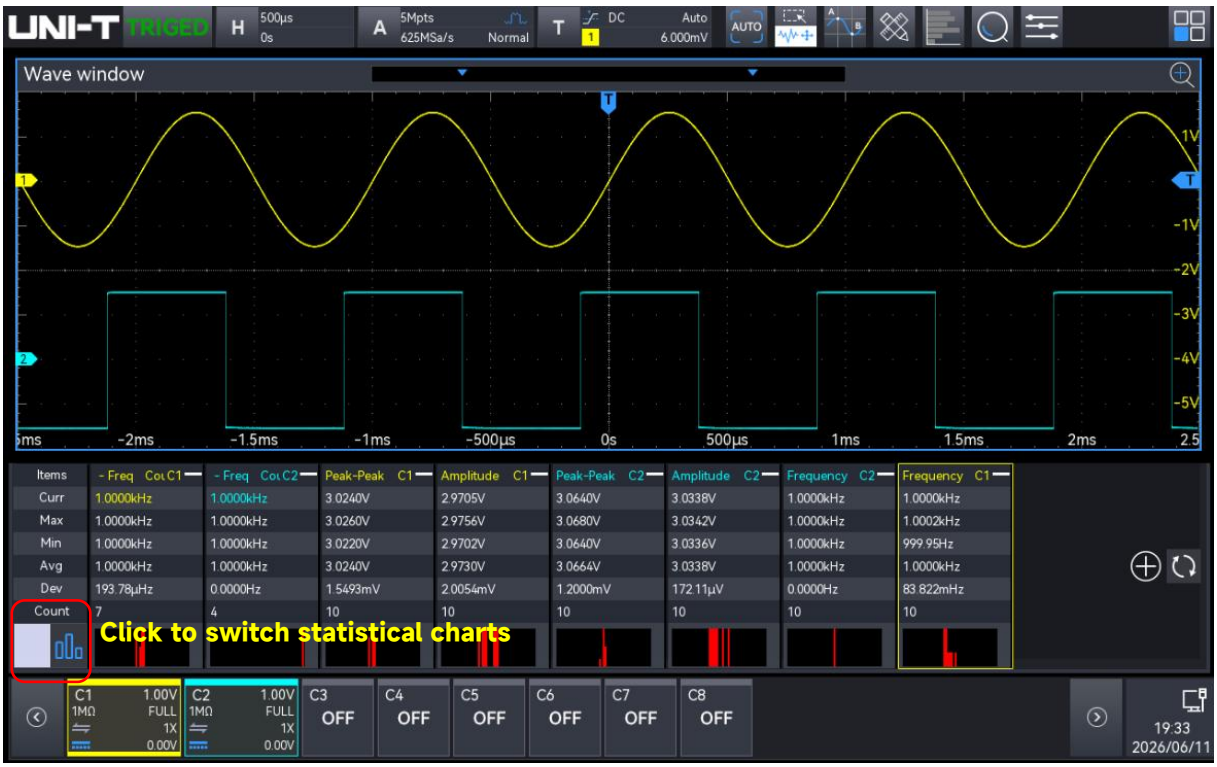
10.6. Measurement Statistics

Press the Meas button on the front panel, then click Measurement Statistics in the measurement 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 the following methods.

- Press the Meas button on the front panel, then click Add Parameter in the measurement menu to open the add parameter menu.
- Click the icon ⊕ in the parameter measurement info label to enter the add parameter menu.

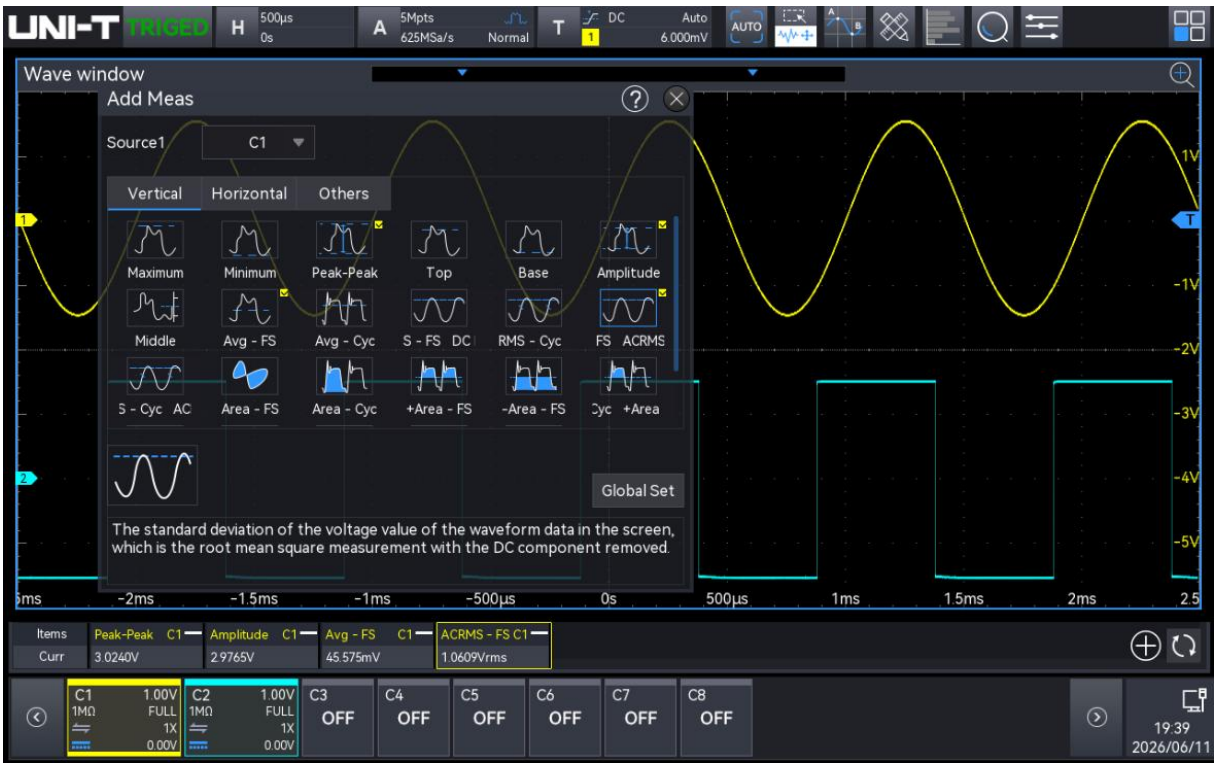
In the Add Parameter menu, click Vertical, Horizontal, Other, or Power, or swipe left or right to browse the menu, then select a parameter to start the corresponding measurement. This oscilloscope supports up to 27 parameter measurements to be opened at the same time.

a. Source

Click Source 1 or Source 2 to select C1 - C8, M1 - M8, as the measurement source.

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.
- Turn on/off: Turn on/off time.
- Transient response: Transient response time
- Efficiency: Input power, output power, and efficiency

10.8. Clear Added Measurement

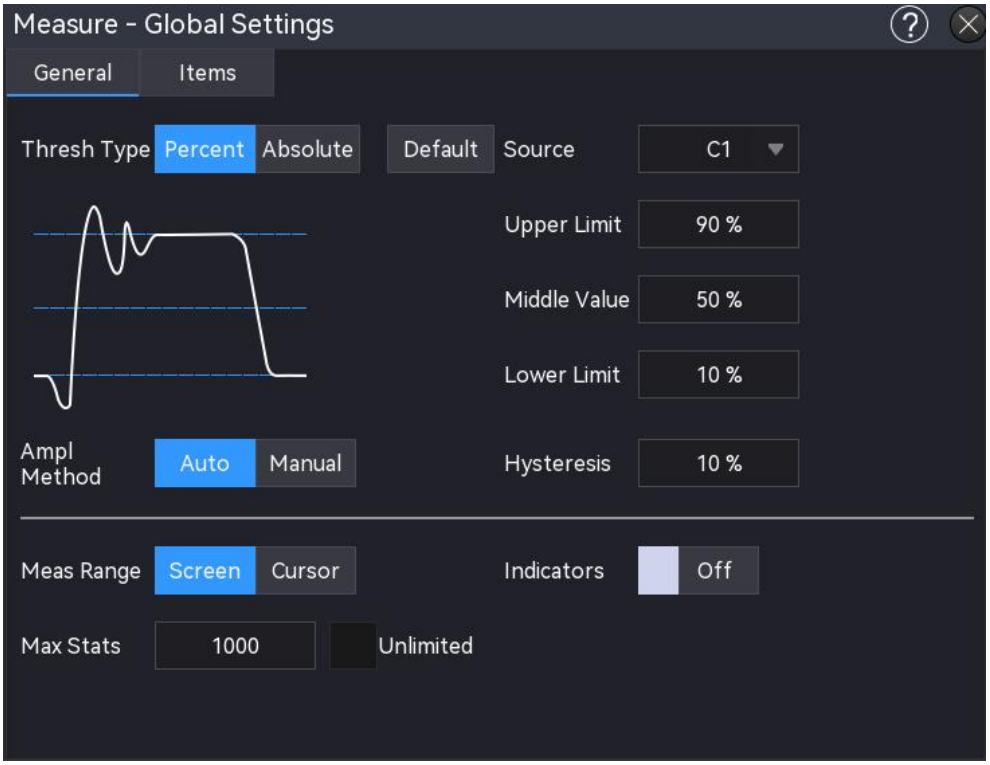
All added measurements can be deleted.

- 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.

- Press the Meas button on the front panel, in the Measure menu, click Clear to delete all added measurements.

10.9. Global Setting

Press the Meas button on the front panel, in the Measure menu, click Global Settings to enter 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 - C8, M1 - M8 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. Alternatively, use the Multipurpose rotary knob to adjust the upper limit. 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. Alternatively, use the Multipurpose rotary knob to adjust the middle limit. The middle value is determined by the upper and lower limits. By default, the middle value is 50%, and it can be set within a range of 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. Alternatively, use the Multipurpose rotary knob to adjust the lower limit. 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.

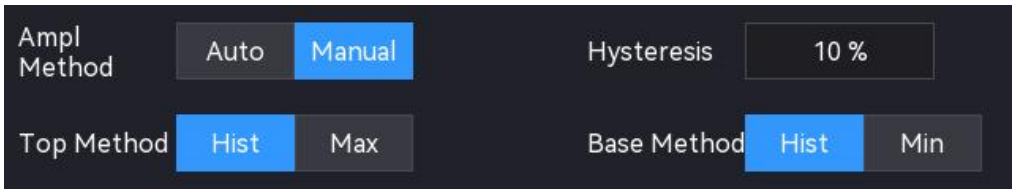
c. Maximum Count

This is a custom parameter. When measurement statistics are enabled, the number of statistical samples can be set from 10 to 10000, or 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

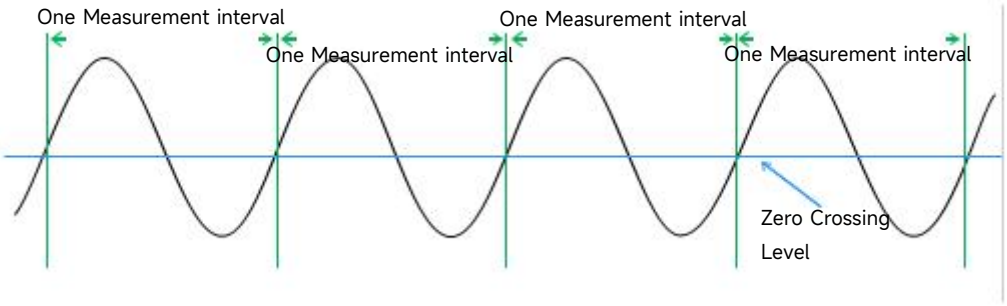
Click Indicator to turn the indicator on or off.

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.

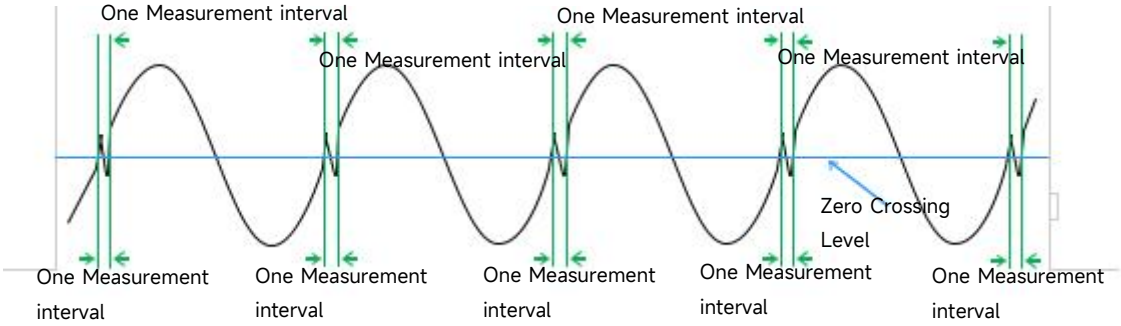
h. Hysteresis

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.

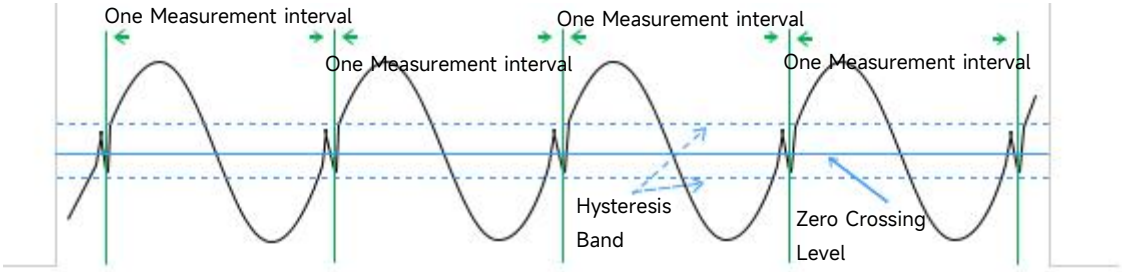
To illustrate the operating principle of hysteresis band settings, consider the perfect sine wave example shown in the figure below. In this case, the zero-crossing point or the 50% crossing point is clearly defined, allowing the oscilloscope to accurately detect the crossing and determine the measurement 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
- Inrush Current
- RDS(ON)
- Switching Loss
- Slew Rate
- SOA
- Modulation
- Ripple
- Turn On/Off Time
- Transient Response
- Efficiency

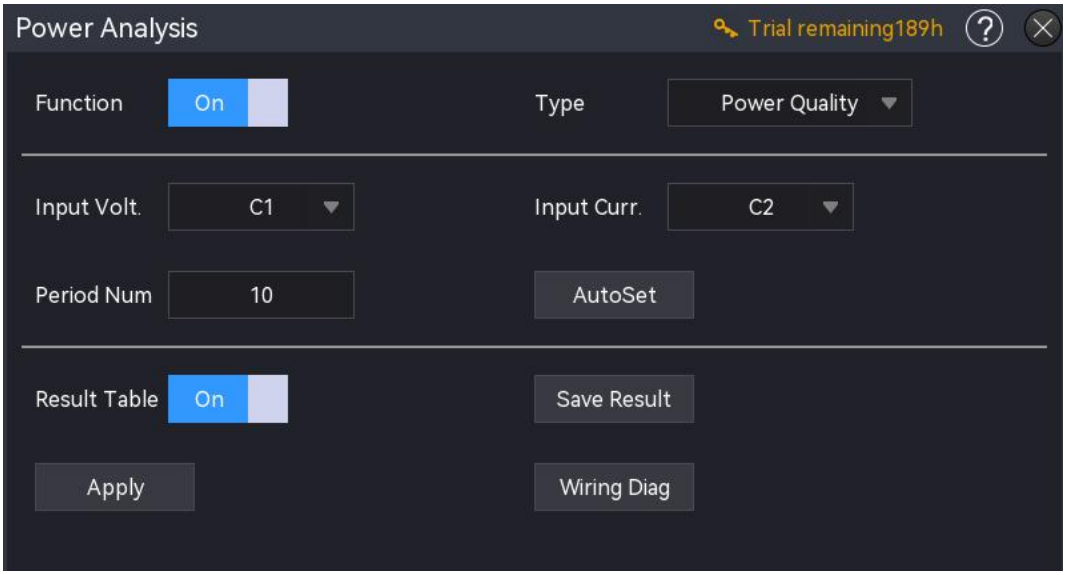
This oscilloscope supports power analysis function (option), it can help the user to quickly analyze the efficiency and reliability of switch power. With this function, the user can analyze the power quality, harmonics analysis, surge current of the input power, RDS(ON), switching loss, slew rate, safe operating area, and modulation analysis, ripple analysis, turn on/off time, transient response, and efficiency.

Access the power analysis menu using the following methods.

- Press the Analyze button on the front panel, then click Analysis in the analyze menu to open the power analysis menu.
- Click Home icon in the top-right corner of the screen, then click power analysis icon to enter the power analysis menu.
- If the power analysis is added in the toolbar, click power analysis icon in the top-right corner of the screen 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

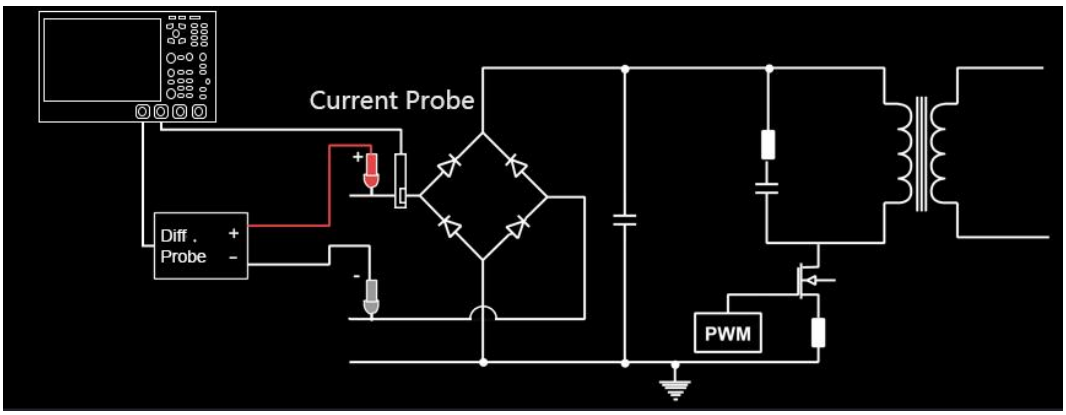
Click Wiring Diagram to display the wiring diagram of the power quality analysis. Follow the instructions in the figure below to make the wiring connections.

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 - C8). 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 - C8). 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 5.8 Parameter Setting. Alternatively, click Period Number input field, then use the Multipurpose rotary knob to adjust the period number. 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

Frequency	Frequency of voltage waveform
Voltage peak	Maximum voltage waveform
RMS voltage	Effective voltage value $\mathrm{Vrms} = \frac{1}{N} * \sqrt{\sum_{i=0}^{n-1} Vi^2}$
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

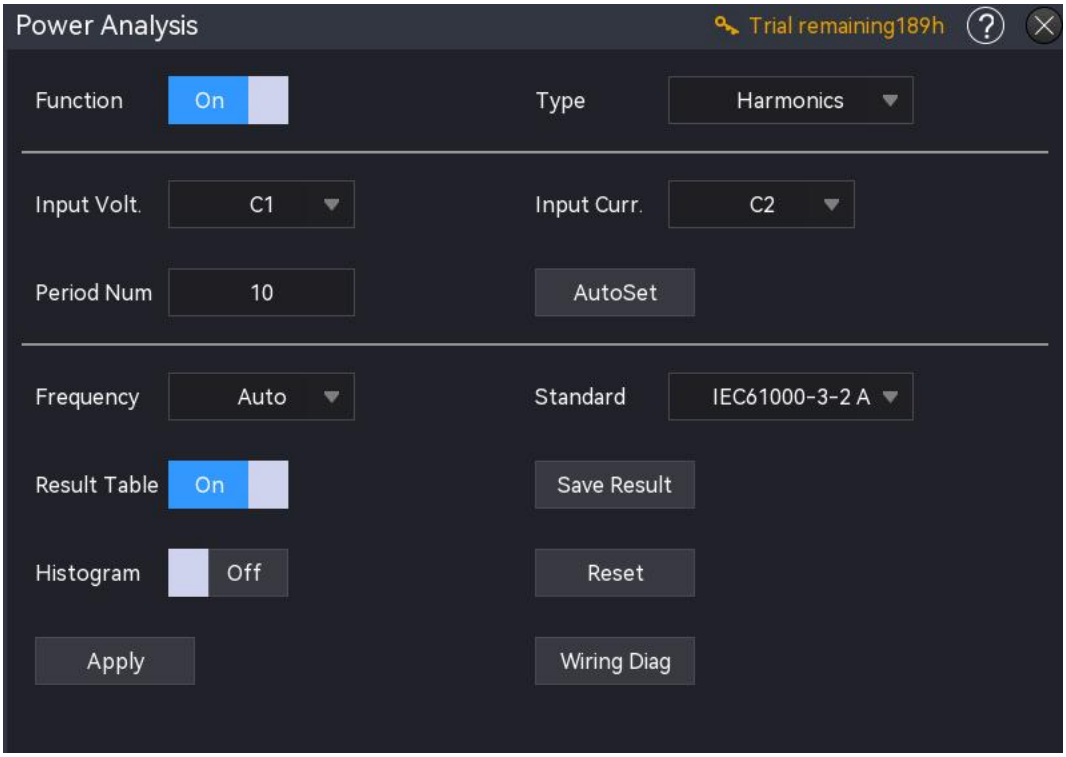
After opening the result table, click Save Result to open the export setting menu, the data can

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 Storage 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

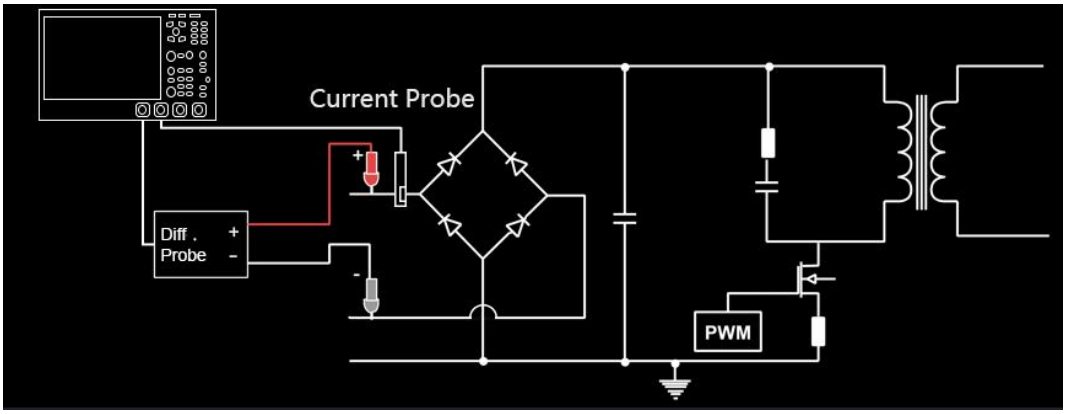
Click Wiring Diagram to display the wiring diagram of the current harmonics analysis. Follow the instructions in the figure below to make the wiring connections.

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 - C8). 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 - C8). 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 5.8 Parameter Setting. Alternatively, click Period Number input field, then use the Multipurpose rotary knob to adjust the period number. 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.

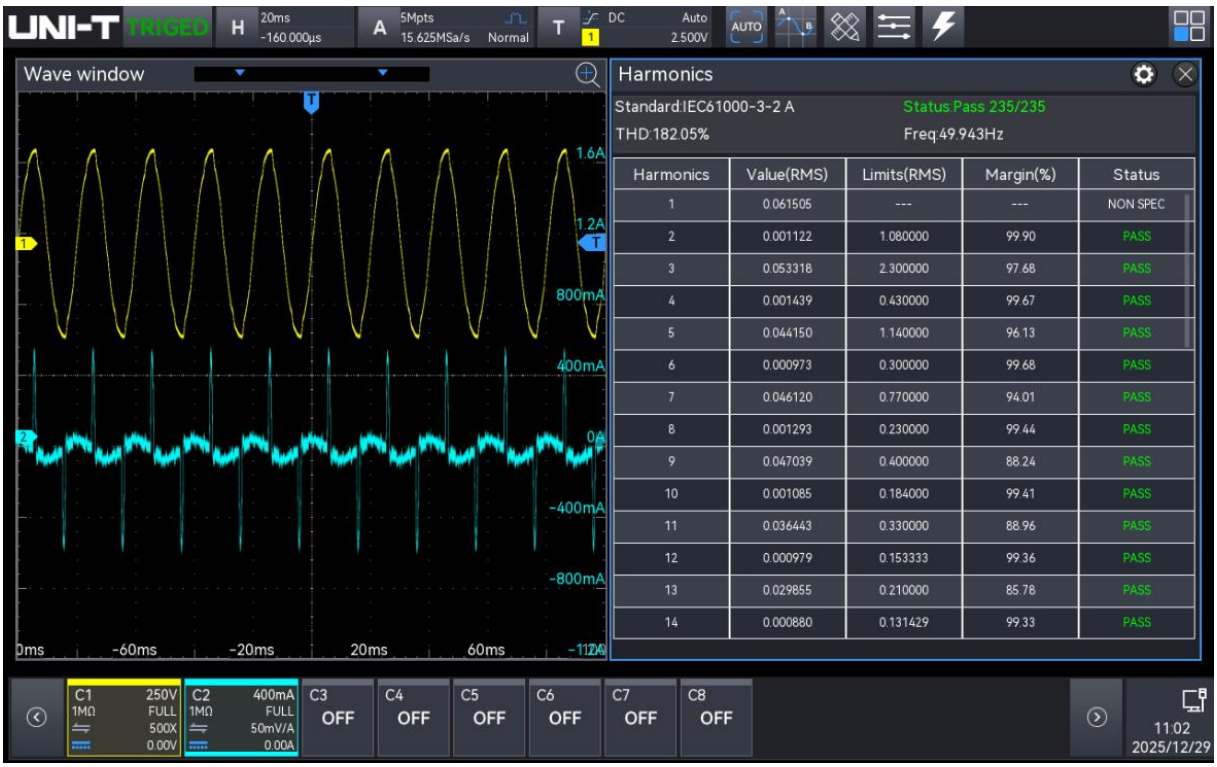
(11) Histogram

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. Actual value (RMS): The measured values displayed in the unit specified by the Harmonic Unit. Limit (RMS): The limit defined by the selected harmonic analysis standard. Margin: The margin defined by the selected harmonic analysis standard. Pass/Fail: Indicates whether the value passes or fails according to the selected harmonic analysis standard. 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)	$\mathrm{THD} = 100 \times \frac{\sqrt{X_2{}^2+X_3{}^2+..+X_n{}^2.}}{X_1}$ X_n = current of each harmonic 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 Storage and Load.

11.3. Inrush Current

The inrush 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 Inrush Current.

(2) Function Switch

Click Function to turn the power analysis on or off.

(3) Wiring Diagram

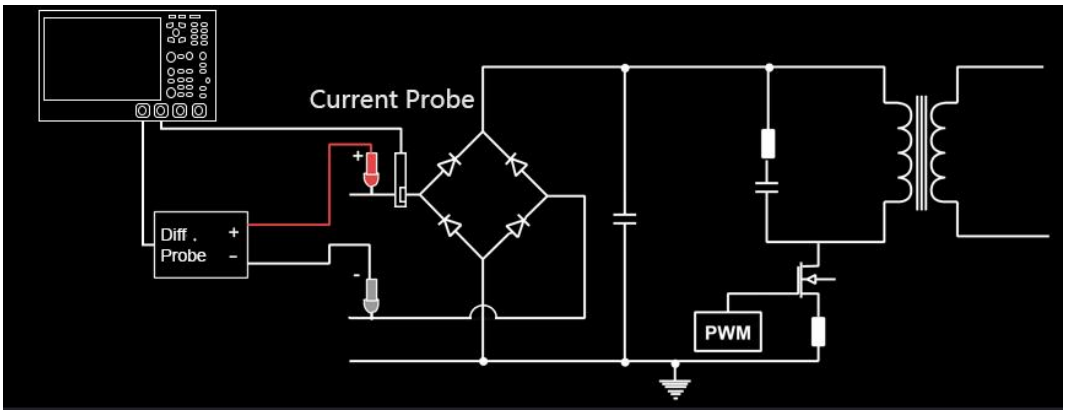
Click Wiring Diagram to display the wiring diagram of the power quality analysis. Follow the instructions in the figure below to make the wiring connections.

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 - C8). 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 - C8). 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 5.8 Parameter Setting. Alternatively, click Maximum Input Voltage (RMS) input field, then use the Multipurpose rotary knob to adjust the maximum input voltage. 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.

Double-click Prospective Current input field to open the numeric keypad and set the current. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, click Prospective Current input field, then use the Multipurpose rotary knob to adjust the current. The setting range is from 100 mA to 500 A.

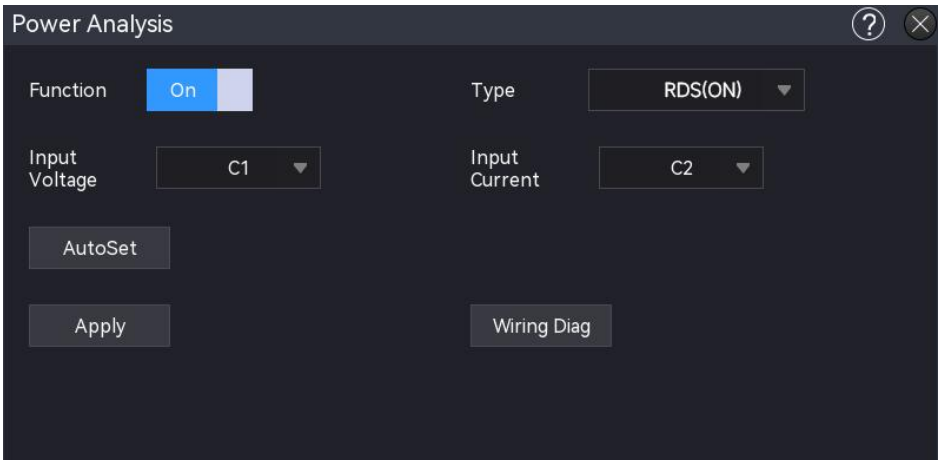
(8) Apply

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

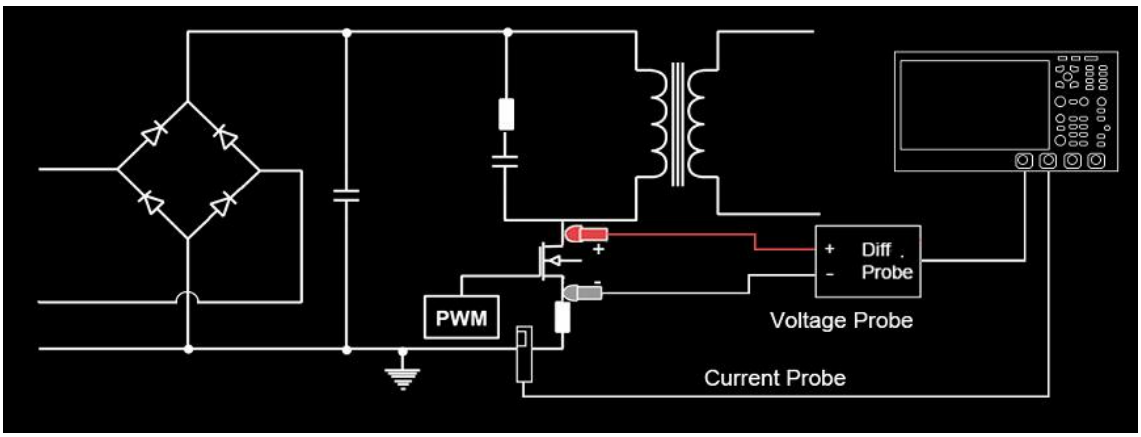
Click Wiring Diagram to display the wiring diagram of RDS(ON). Follow the instructions in the figure below to make the wiring connections. Tap the icon in the top-right corner to close the wiring diagram.

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 - C8). 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 - C8). 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.

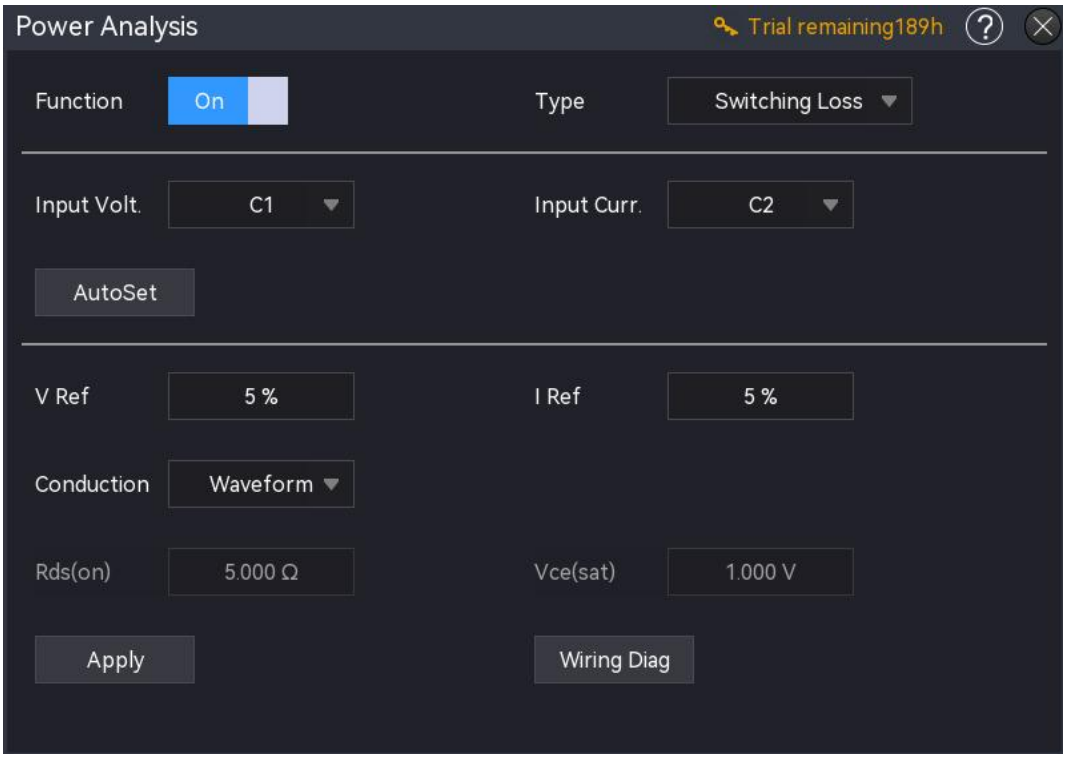


(8) Slew Rate Measurement Results

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.

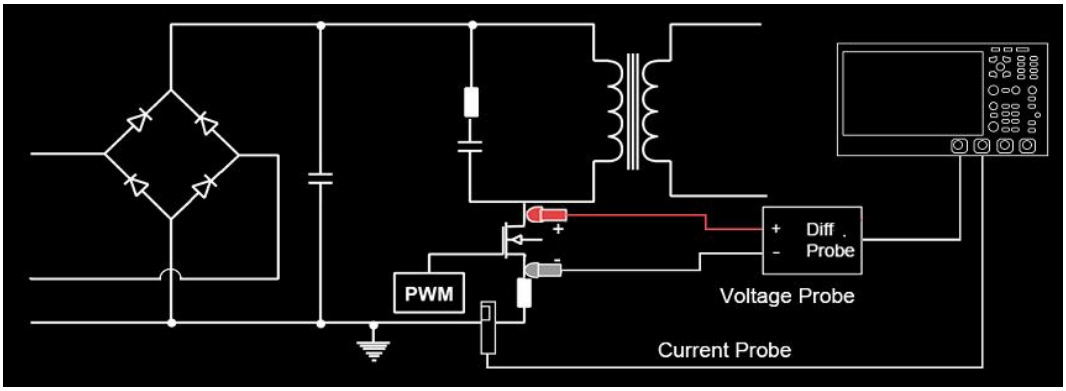
a. Voltage Probe

- 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 - C8). 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 - C8). 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.



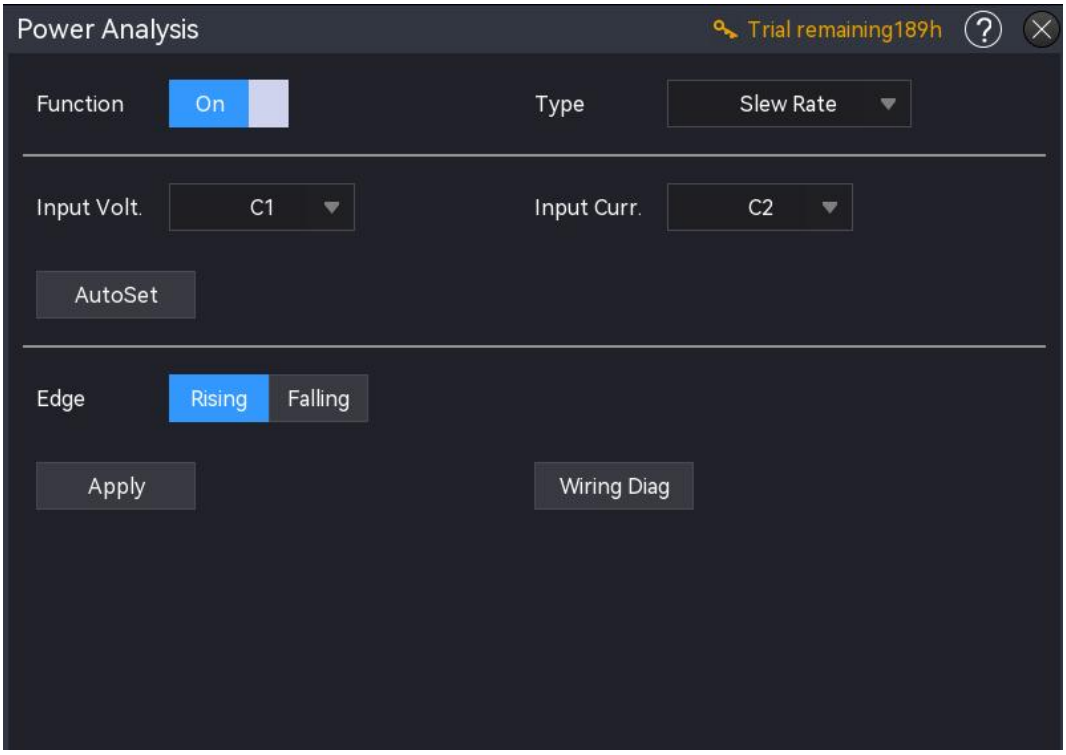
(13) Switching Loss Measurement Results

Switching period	Meets the required number of switching periods.

number	
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

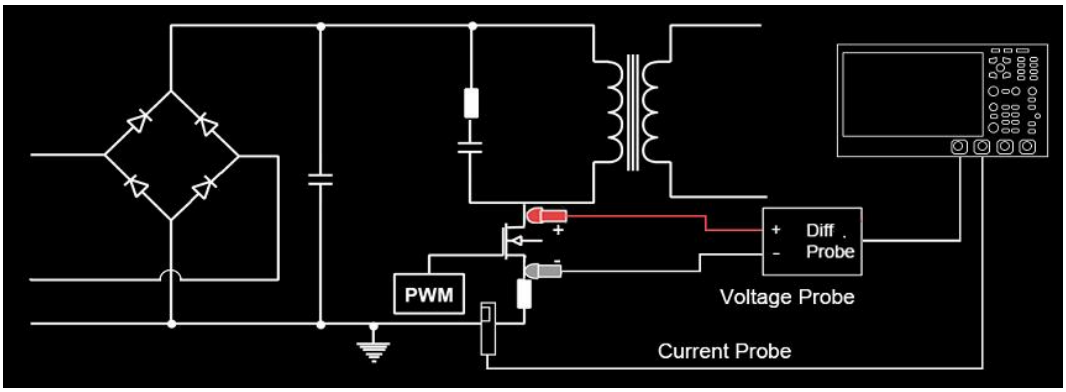
Click Wiring Diagram to display the wiring diagram of slew rate. Follow the instructions in the figure below to make the wiring connections. Tap the icon in the top-right corner to close the wiring diagram.

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 - C8). 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 - C8). 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.



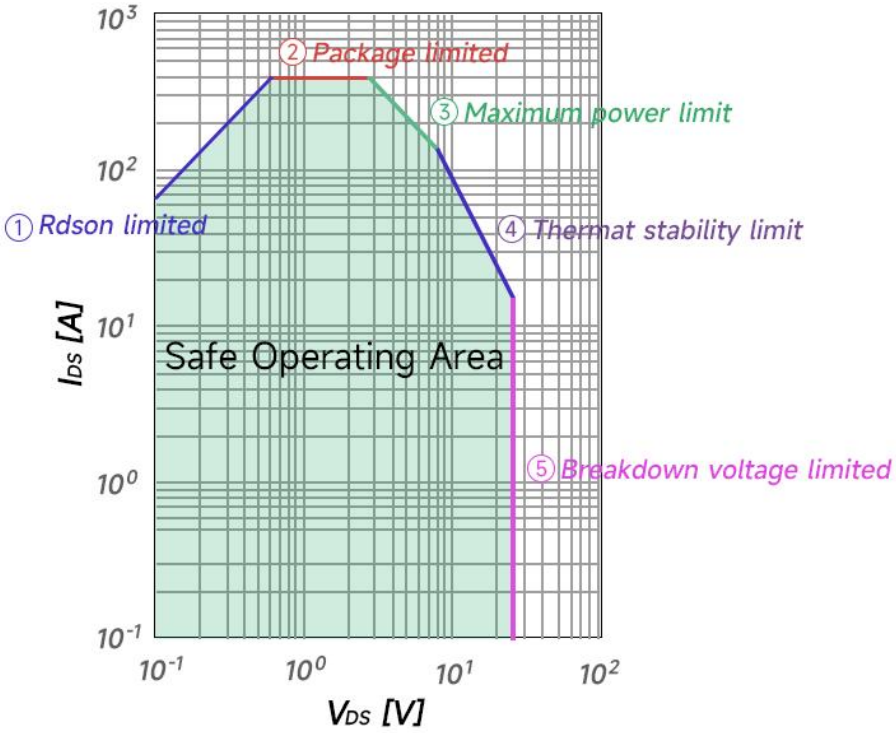
(9) Slew Rate Measurement Results

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. SOA

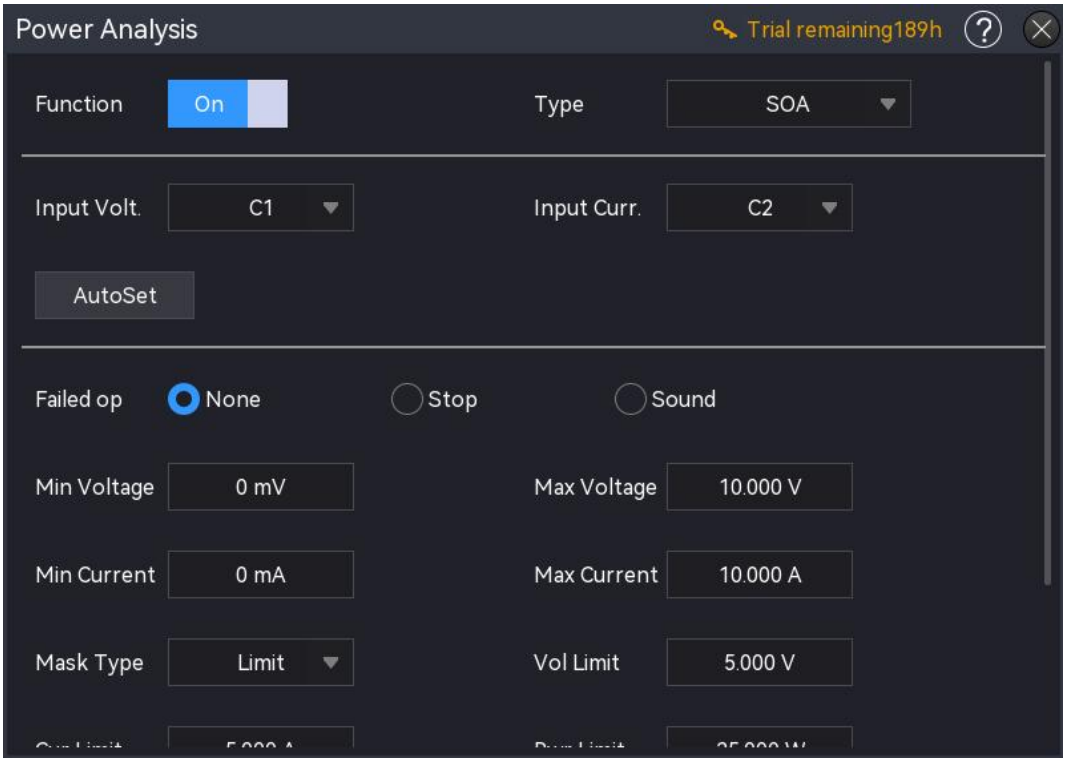
The safe operating area (SOA) of a MOSFET defines the limiting conditions for voltage, current, and power dissipation, under which the device can operate without sustaining damage.

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

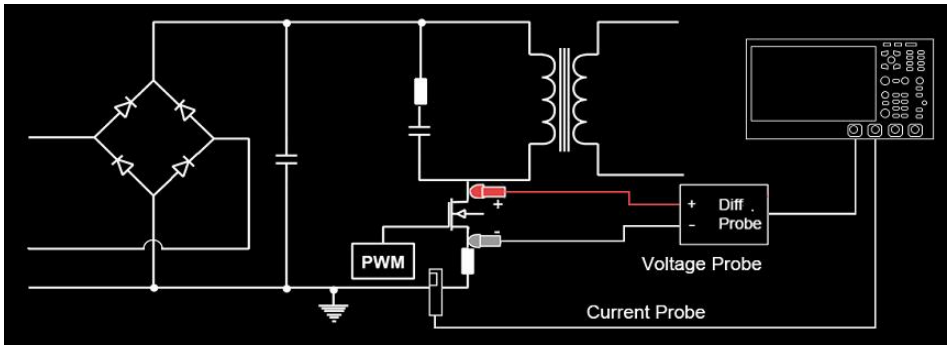
corner to close the wiring diagram.

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 - C8). 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 - C8). 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.

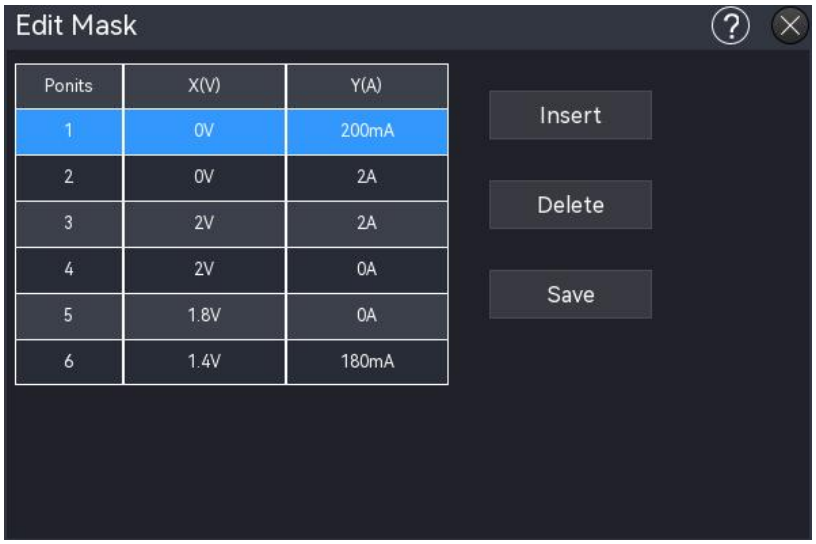
- Minimum voltage: Sets the display minimum value for SOA voltage axis, with a default value of 0 mV.
- Maximum voltage: Sets the display maximum value for SOA voltage axis, with a default value of 10 V.
- Minimum current: Sets the display minimum current for SOA voltage axis, with a default value of 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 test.
- 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 the setting steps, refer to the section of Storage and Load.

(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.

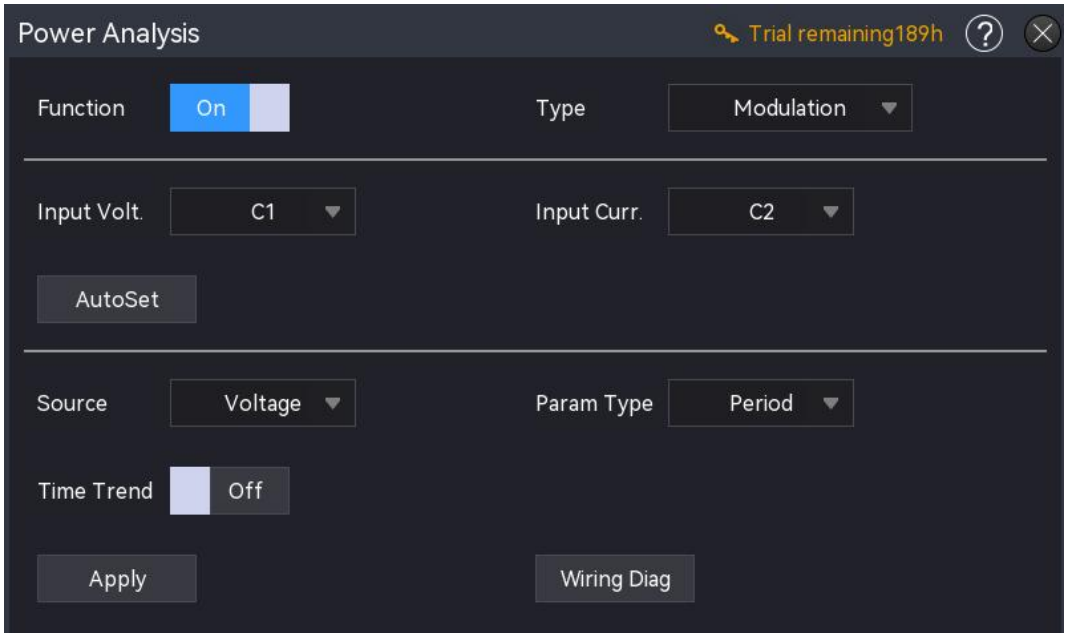
(12) Apply

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

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

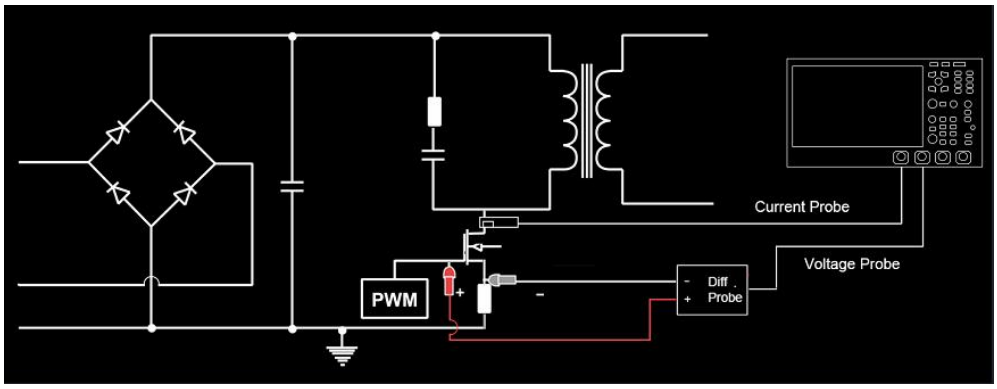
Click Wiring Diagram to display the wiring diagram of modulation analysis. Follow the instructions in the figure below to make the wiring connections. Tap the icon in the top-right corner to close the wiring diagram.

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 - C8). 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 - C8). 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

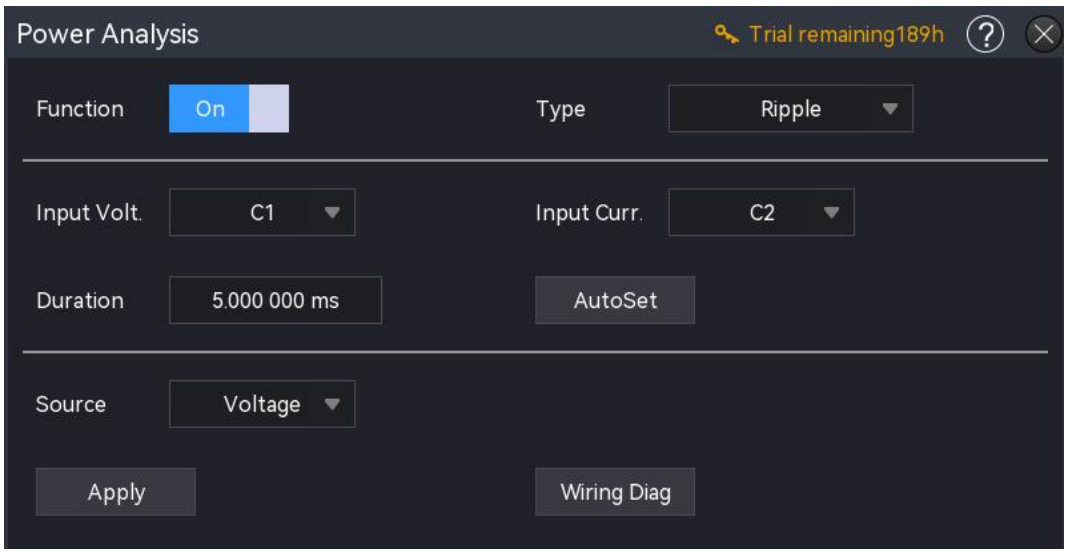
Click Apply to enable the oscilloscope to perform the power analysis function based on the input voltage, input current, source reference, and parameter type. The time trend chart will automatically display after the analysis is complete.



The time trend graph visualizes the measured changes in each cycle of the modulation waveform using a mathematical waveform.

11.9. Ripple

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.

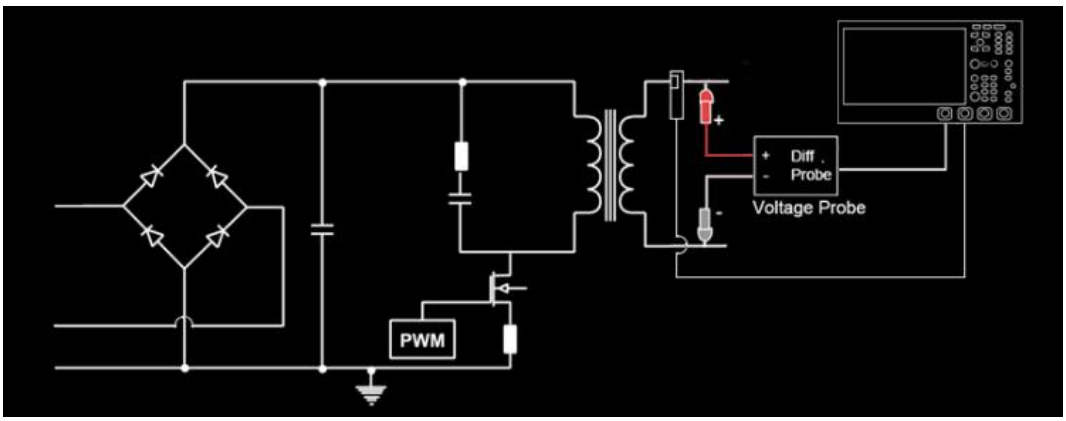
a. Voltage Probe

- 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 - C8). 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 - C8). 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

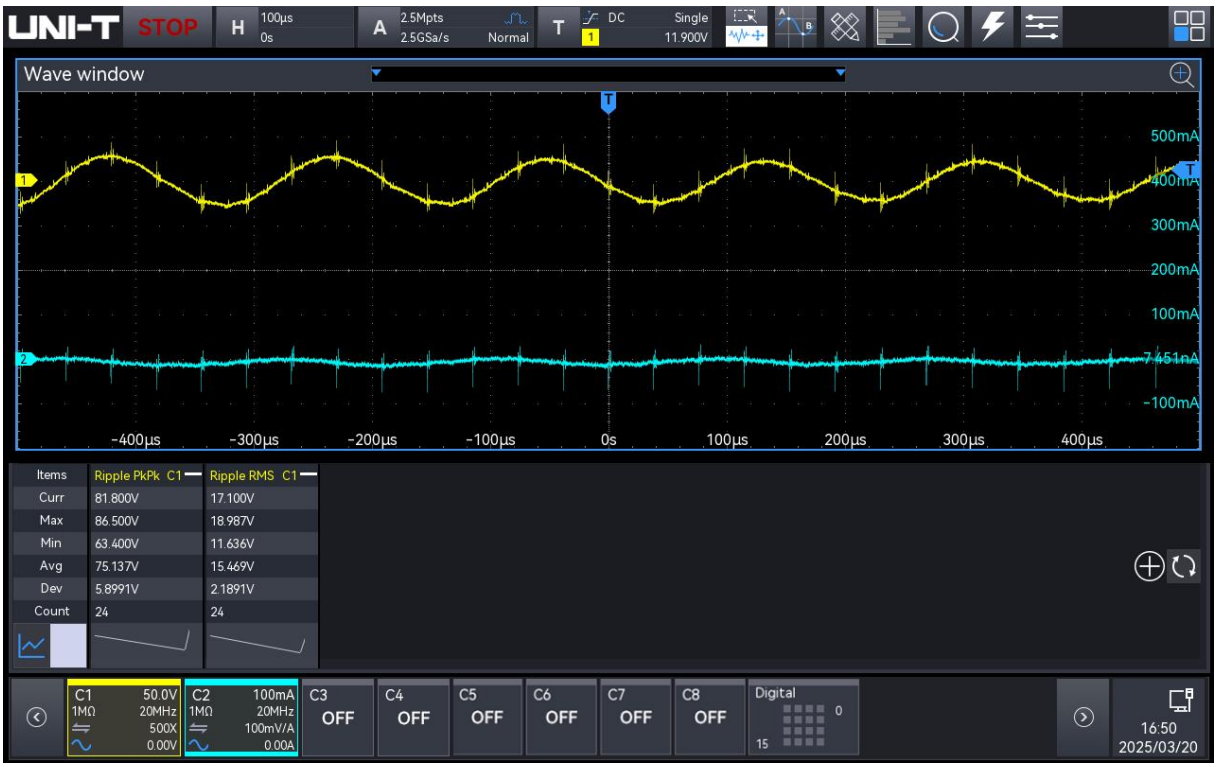
Set the time span for the entire waveform display. The valid range is from 10 ns to 20 s. The oscilloscope automatically adjusts the time base range using the formula: Time Base = Duration / 10

(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.

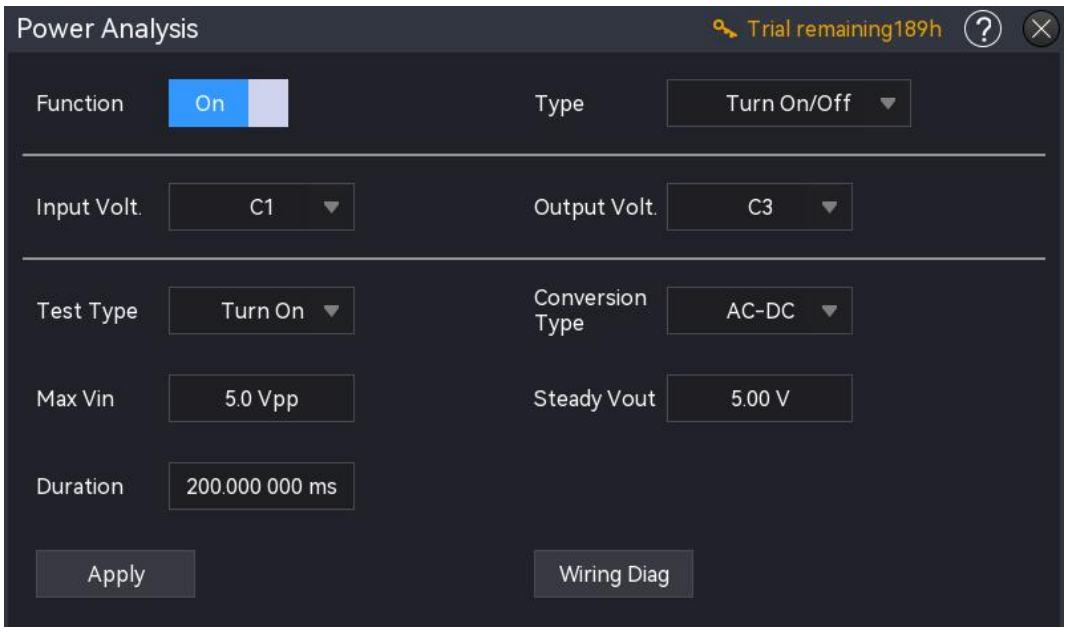
11.10. Turn On/Off Time

During turn on 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 turn off 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

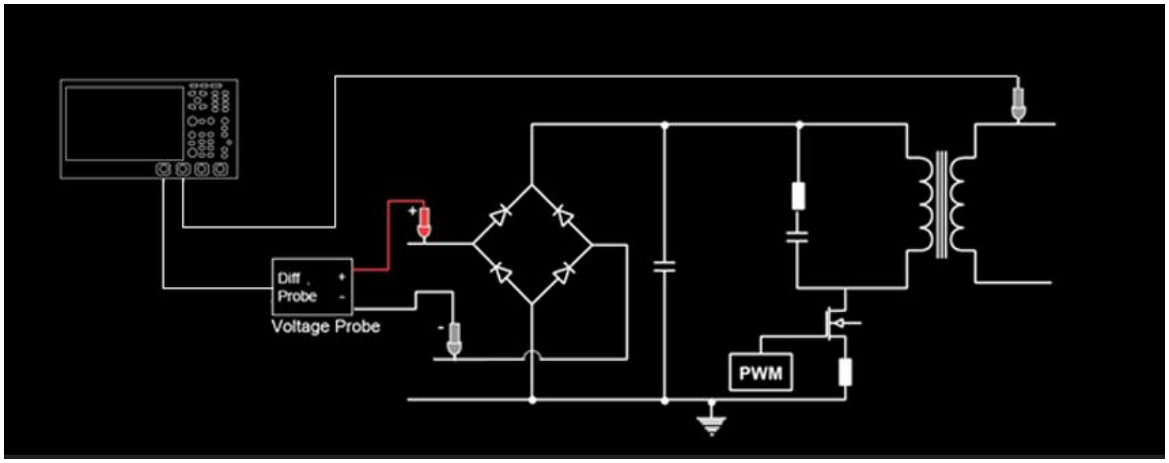
Click Wiring Diagram to display the wiring diagram of the startup/shutdown time. Follow the instructions in the figure below to make the wiring connections. To close the diagram, click close icon in the upper-right corner.

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 - C8). 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 - C8). 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) Turn On/Off Time Measurement Results

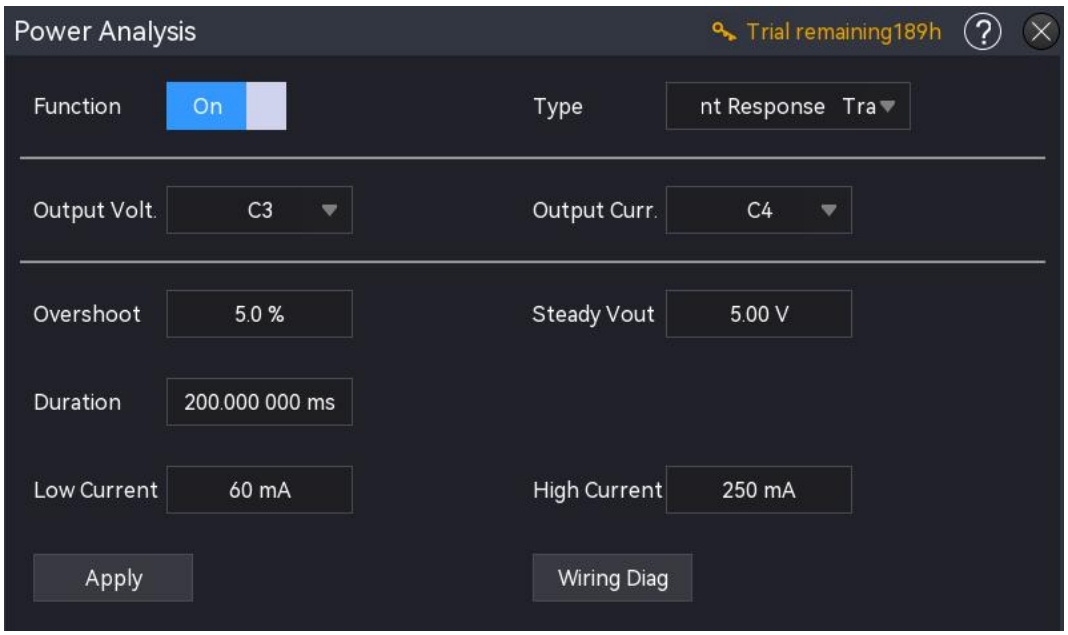
Startup Time	The time it takes for the output voltage to rise to 90% of its steady-state value after the power supply is powered on.
Shutdown Time	The time it takes for the output voltage to fall to 10% of its maximum value after the power supply is turned off.

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

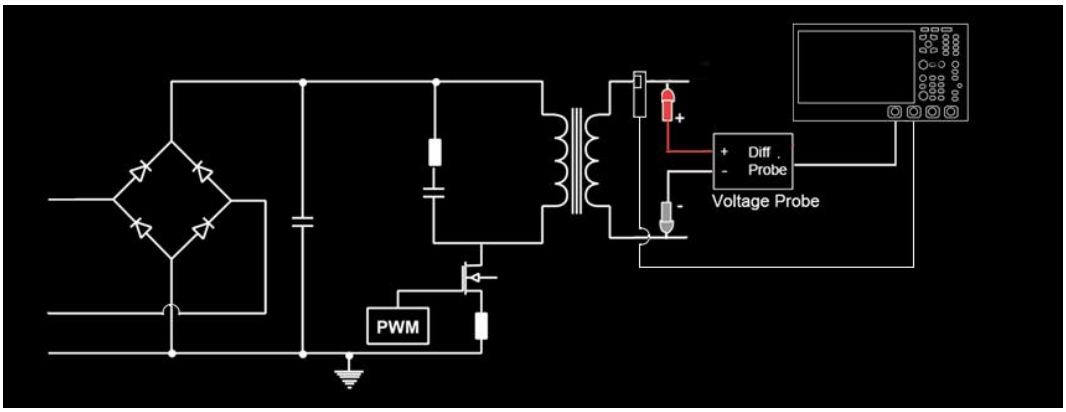
Click Wiring Diagram to display the wiring diagram of the transient response, please follow the instructions to make the wiring connection, as shown in the following figure. To close the diagram, click close icon in the upper-right corner.

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 - C8). 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 - C8). 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

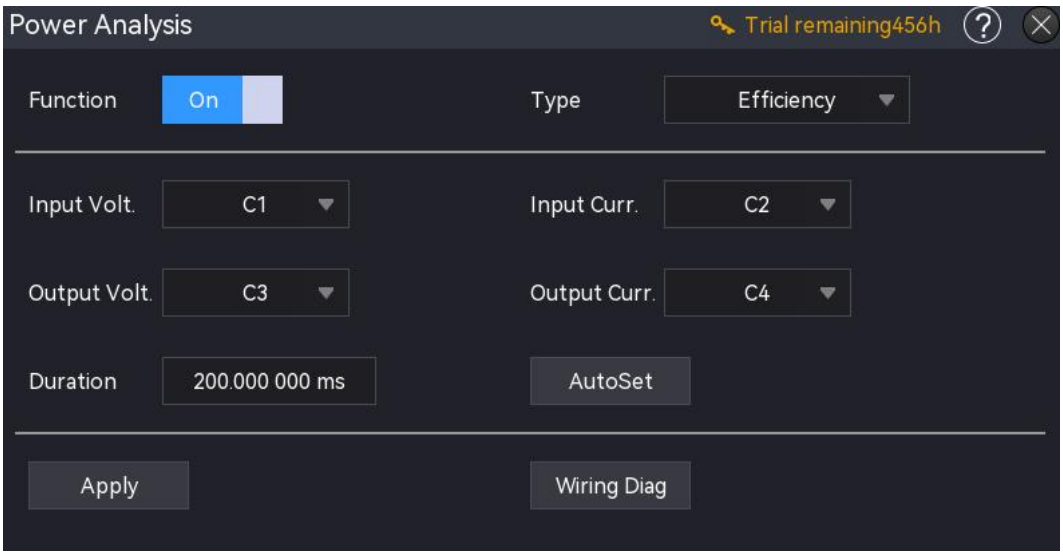
Transient Response Time	The time required for the output voltage to recover and stabilize within the allowable error range, measured from the moment it deviates from the steady-state value due to a sudden change in the load current at the power supply output.

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

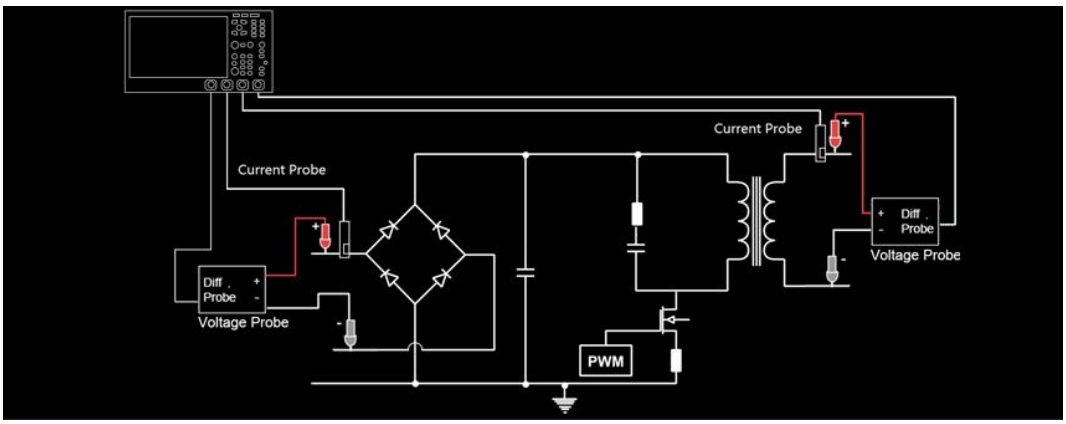
- Connect the current probe to the power output. The arrow indicates the direction of current flow.

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 iutput voltage source (C1 - C8). 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 iutput current source (C1 - C8). 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 voltage source (C1 - C8). 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 - C8). The current channel should be configured with the appropriate unit and probe attenuation ratio according to the connected current probe.

(8) 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.

(9) Apply

Click Apply key and follow the on-screen instructions. Once the analysis is complete, the results are displayed.



(10) 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

Cursors can be used to measure the X-axis (time) and Y-axis (voltage) values of the selected waveform. The cursor measurement supports simultaneous measurement of multiple channels, as well as Math waves.

Configurable options: source, measurement type, synchronous movement, and horizontal cursor.

The cursor measurement can be entered using the following steps.

- Press the Cursors button on the front panel to open the cursor measurement menu.
- Click Home icon in the top-right corner of the screen, then click cursor icon to enter the cursor measurement menu.
- If the cursor measurement is added in the toolbar, click cursor icon to enter the cursor measurement menu.
- If cursor measurement results appear, click icon to open the cursor measurement.

(1) Source: Sets the source for the cursor measurement.
Available options: C1 - C8, M1 - M8.

(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.

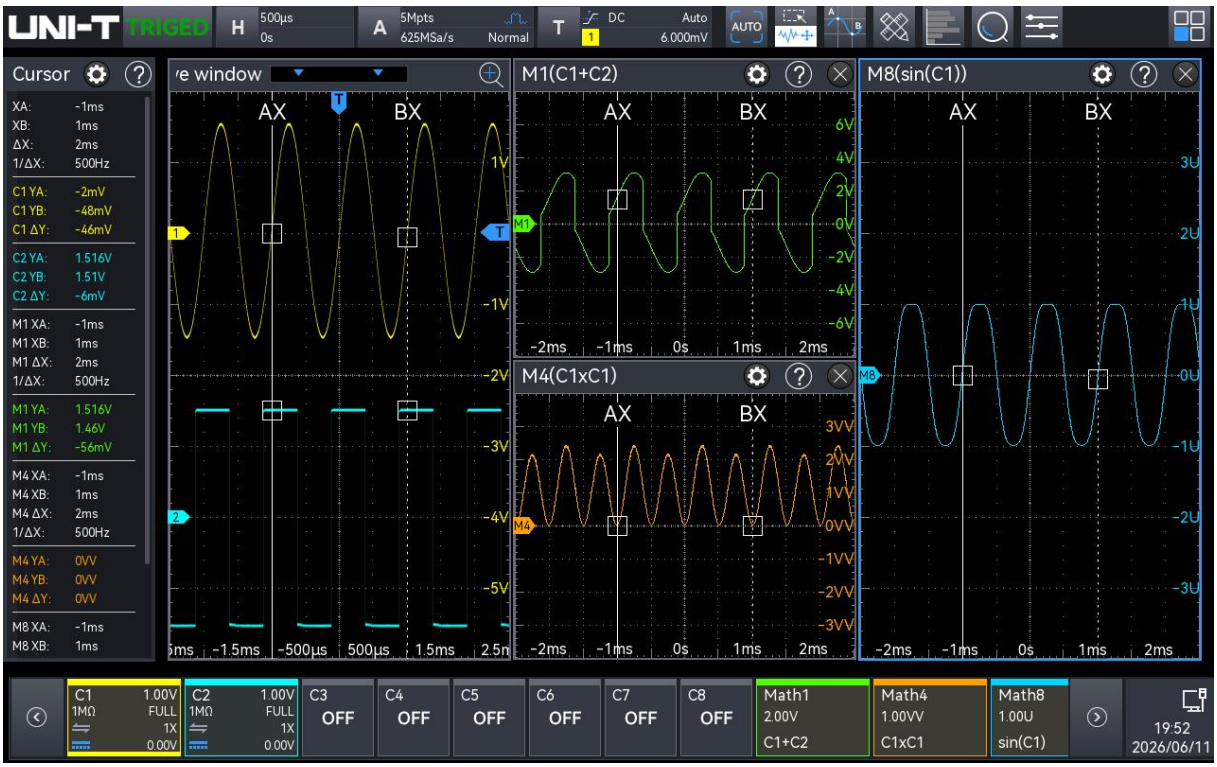
(4) Horizontal cursor: Sets the horizontal cursor position relative to the time base. Two modes can be selected: position fixed and delay fixed.
- Position fixed: The default display is at ±2 divisions. When the time base is changed, the positions of the AX and BX cursors on the screen remain unchanged.
- 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.

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 result information window (top-left corner): "X" indicates the time measurement result of the channel.

"Y" indicates the voltage measurement result at the intersection of the enabled channel and the cursor.

Math waveforms support split-screen display.When using cursors to measure these waveforms, split-screen display is supported.

The cursor position on each split screen can be adjusted independently without affecting other screens.

The positions of the AX and BX cursors can be set using the following methods

- Use the Multipurpose rotary knob to move the cursor.

  Clockwise: Move right

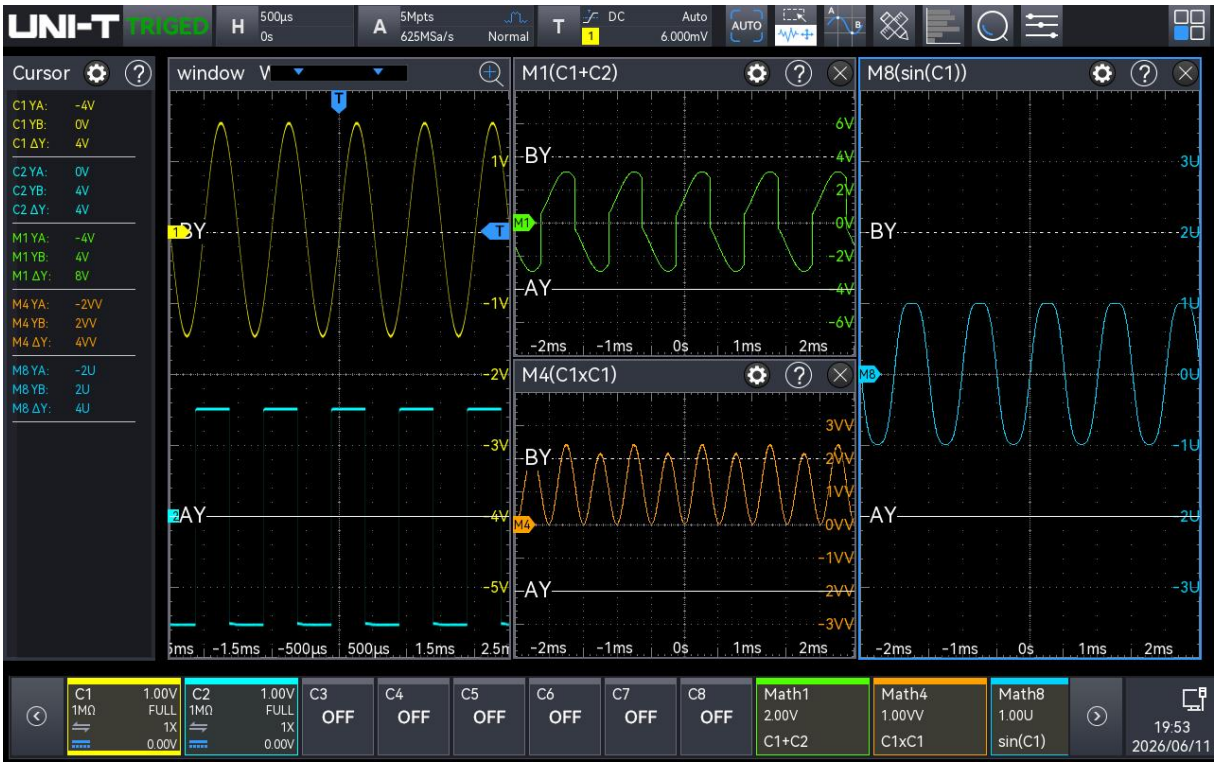
  Counterclockwise: Move left
- Press the Multipurpose rotary knob switch between AX and BX.
- Tap AX or BX, then drag the cursor to the desired position. For details on using the drag gesture,

refer to section Touch Screen - Drag.

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 result information window (top-left corner):

"Y" indicates the voltage measurement result at the intersection of the enabled channel and the cursor.

Math waveforms support split-screen display.When using cursors to measure these waveforms, split-screen display is supported.

The cursor position on each split screen can be adjusted independently without affecting other screens.

The positions of the AY and BY cursors can be set using the following methods

- Use the Multipurpose rotary knob to move the cursor.

  Clockwise: Move up

  Counterclockwise: Move down

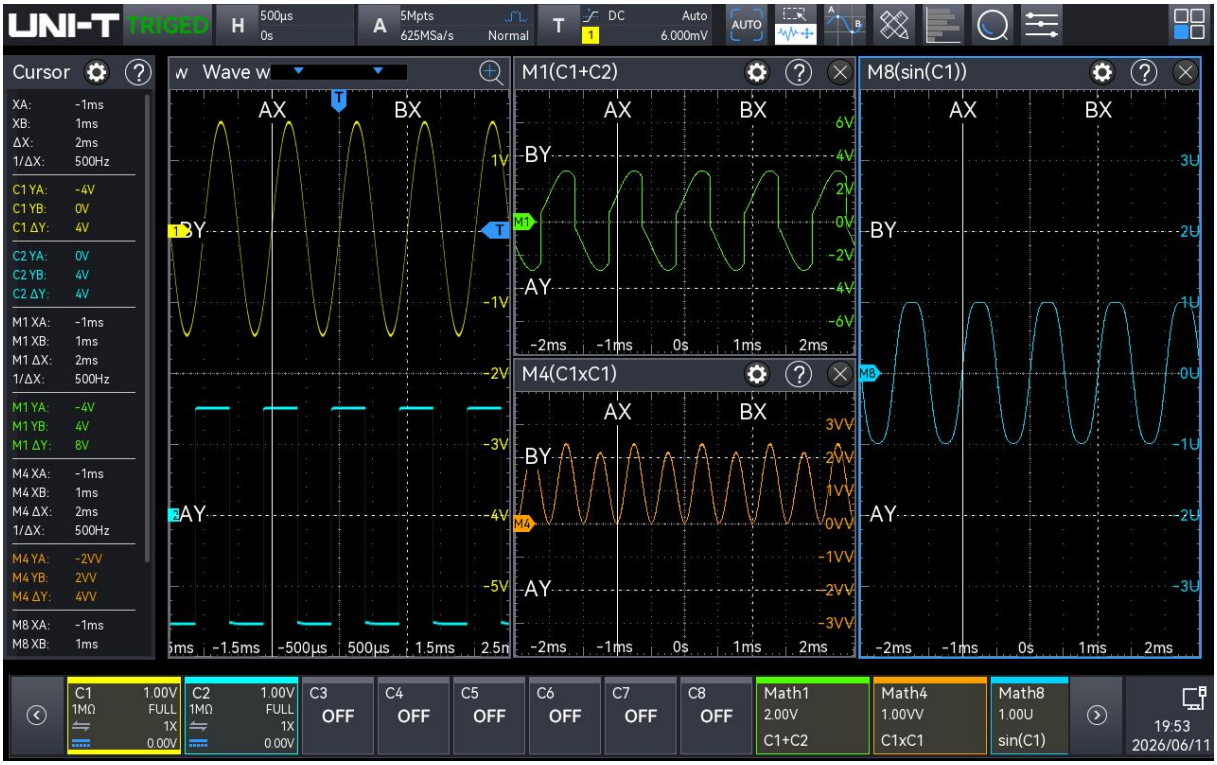
- Press the Multipurpose rotary knob switch between AY and BY.

- Tap AY or BY, then drag the cursor to the desired position. For details on using the drag gesture, refer to section Touch Screen - Drag.

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 result information window (top-left corner): “X” indicates the time measurement result of the channel.

“Y” indicates the voltage measurement result at the intersection of the enabled channel and the cursor.

Math waveforms support split-screen display.When using cursors to measure these waveforms, split-screen display is supported.

The cursor position on each split screen can be adjusted independently without affecting other screens.

The positions of the AX, BX, AY, and BY cursors can be set using the following methods

- Use the Multipurpose rotary knob to move the cursor.

  Clockwise: Move right (up)

Counterclockwise: Move left (down)

- Press the Multipurpose rotary knob switch between AX, BX, AY, and BY
- Tap AX, BX, AY, or BY, then drag the cursor to the desired position. For details on using the drag gesture, refer to section Touch Screen - Drag.

13. Sampling System

- Sampling Rate
- Acquisition Mode
- Memory Depth
- Interpolation Method
- ERES (Enhanced Resolution)

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.

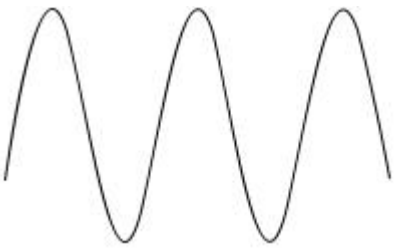
- Click Acquire sampling info label at the top of the screen to enter the sampling setting menu, as shown in the following figure.



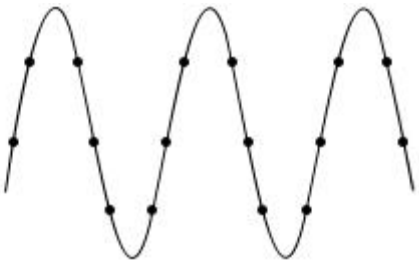
13.1. Sampling Rate

(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



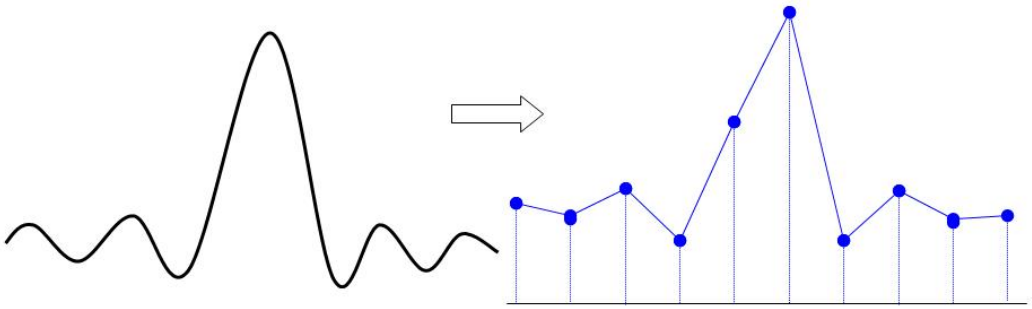
Sampling Point

Sampling rate indicates the time interval between two sampling points. The maximum sampling rate of the UPO6000HD series high-resolution oscilloscopes is 5 GSa/s.

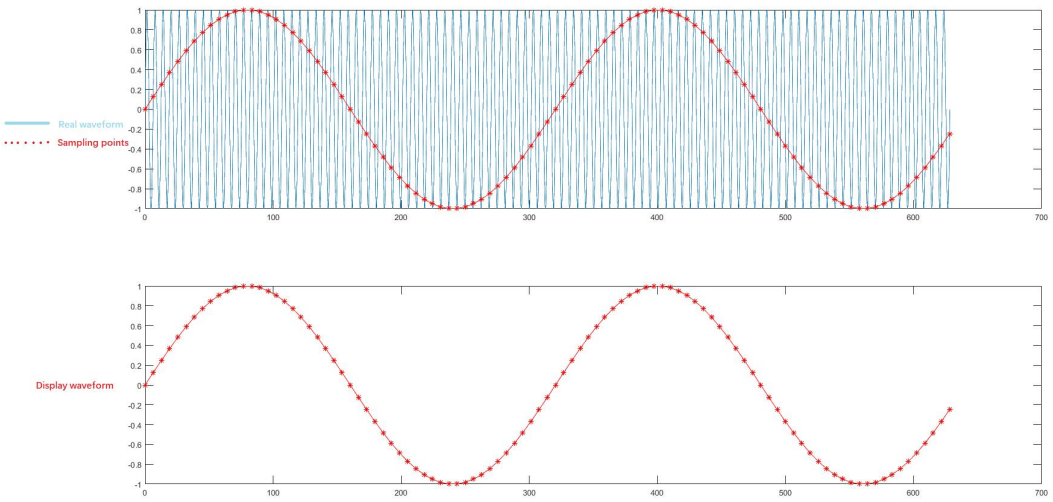
The sampling rate will change with the time base scale and memory depth. The real-time sampling rate is displayed in the Acquire sampling label at the top of the screen, the horizontal time base can be adjusted using the horizontal Scale or by changing the memory depth.

(2) Effect of Low Sampling Rate

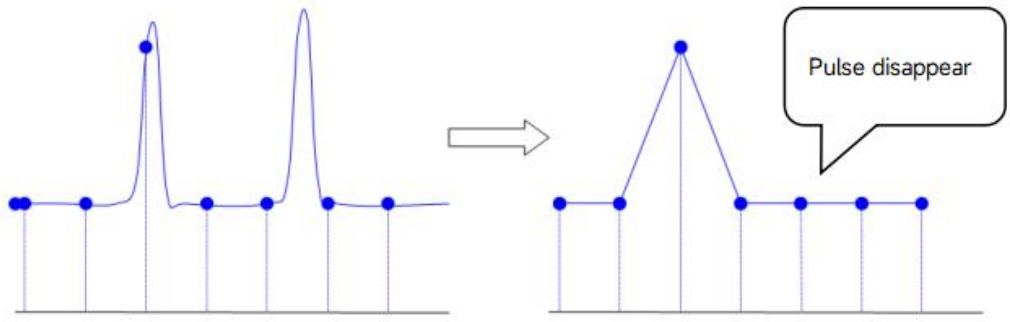
- Waveform distortion: Due to a low sampling rate, waveform details might be lost, and the sampled waveform may differ significantly from than the actual signal, as shown in the following figure.



- 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.

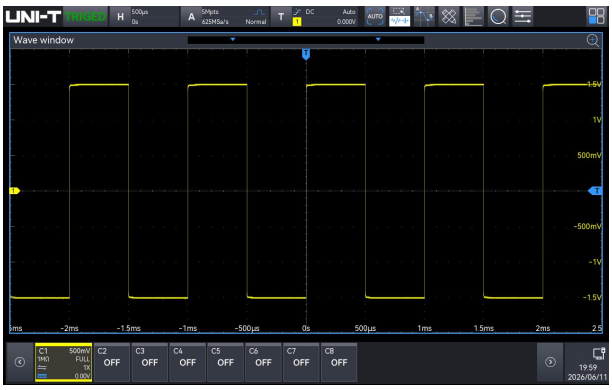
(4) Average

The oscilloscope obtains several waveforms and calculates its averaged value, and then displays the final waveform. This mode can reduce the random noise.

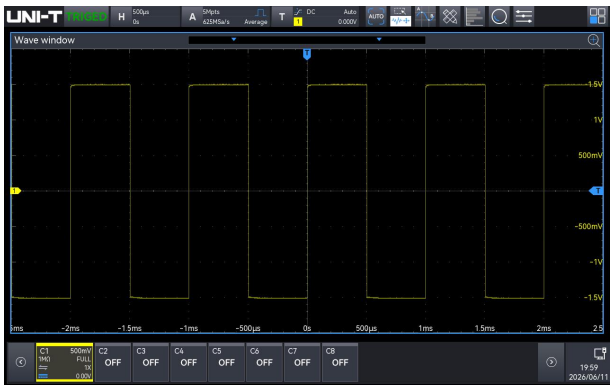
To observe the waveform by changing the acquisition method. If the signal contains large noise, the waveform does not average and the waveform adopts 32 times averaged as shown in the following figure. The higher the average value, the lower the noise and the vertical resolution will be much higher, 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 ERES (Enhanced Resolution).

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 UPO6000HD supports the following memory depth settings: Auto (10 Mpts), 25 kpts, 250 kpts, 500 kpts, 5 Mpts, 50 Mpts, 100 Mpts, or maximum.

Maximum memory depth: With all 8 channels enabled: each channel supports 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

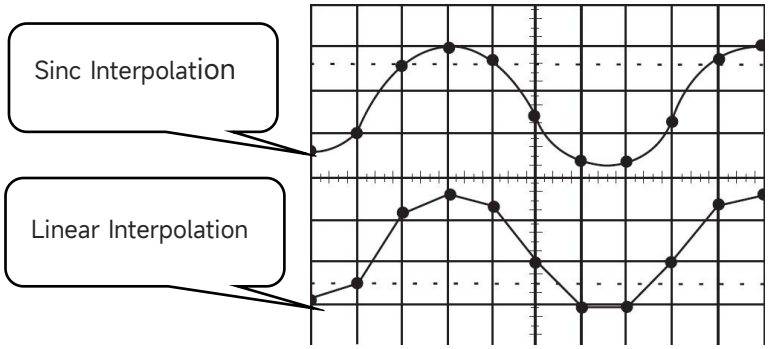
waveform.

The interpolation is divided into Sinc interpolation (sinx/x) and linear interpolation.

Linear interpolation: Straight lines are directly connected at adjacent sampling points.

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.



13.5. ERES (Enhanced Resolution)

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 UPO6000HD 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.

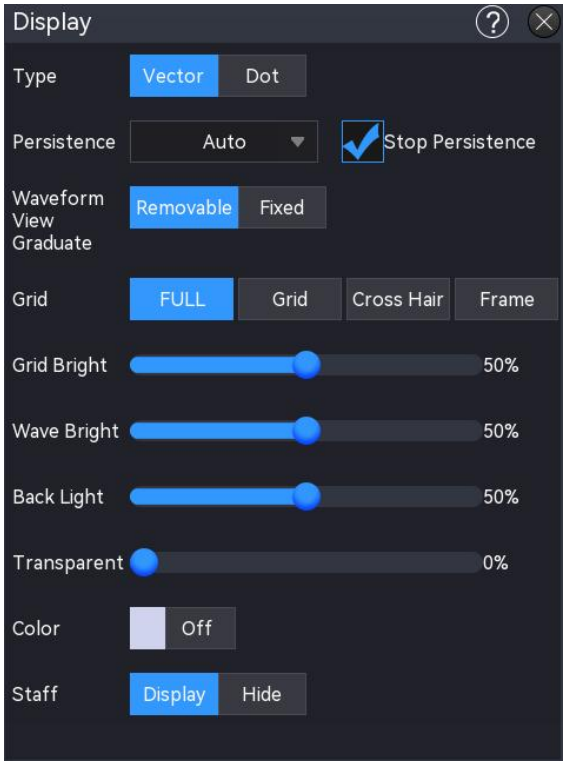
Eres Bit	- 3 dB Bandwidth
1 bit	0.6*Sampling rate
1.5 bits	0.2* Sampling rate
2 bits	0.1* Sampling rate
2.5 bits	0.032* Sampling rate
3 bits	0.012* Sampling rate
4 bits	0.005* Sampling rate

14. Display Control

In the Display menu, set the waveform type, persistence, waveform view scale, grid type, grid brightness, waveform brightness, background brightness, window transparence, color temperature, and staff.

Access the Display menu using the following methods.

- 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 top-right corner of the toolbar 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 setting 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 grid.
- Frame: No crosshair, no grid 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 setting 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 setting is 50%.

14.7. Background Brightness

In the Display setting menu, drag the scroll bar to set the background brightness.

The brightness range: 1%-100%. The default setting is 50%.

14.8. 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%: non-transparent, 100%: fully transparent. The default setting is 50%.

Note: The Utility window is non-transparent by default.

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.

14.10. 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.

15. Storage and Load

- Storage Menu
- Save Waveform
- Save Setting
- Save Image
- Load Setting
- File Browser

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.

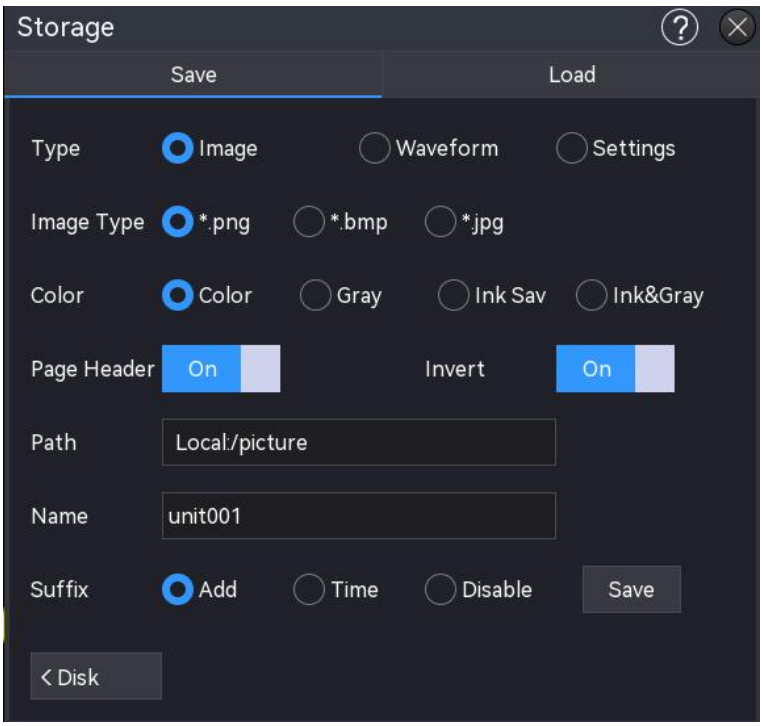
Note: This oscilloscope has three USB HOST ports (one on the front panel and two on the rear panel) for connecting USB flash drives for external storage. It supports only USB flash drives formatted with FAT32, NTFS, or EXFAT.

15.1. Storage Menu

Access the Storage menu using the following methods.

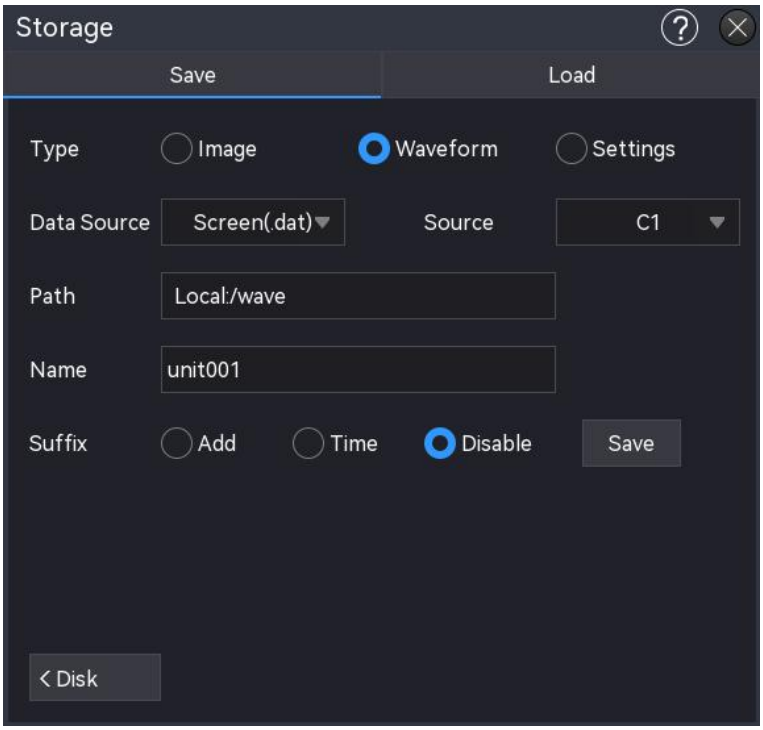
- Click Home icon  in the top right-corner and select the storage icon  to open the storage menu.
- If the display function is added to the toolbar, click storage icon  in the top-right corner of the toolbar to enter the storage menu.

The storage setting menu has two sub-menus: 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 time base) 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 generator 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 - C8 and M1 - M8. When saving .csv, the source can select C1 - C8, M1 - M8, Analog (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 section 5.8 Parameter Setting-Character String.

(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_1.

(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 disabled, the CSV file contains voltage only.

(7) Disk Management

Click Disk to open the file browser. For details on using the file browser, refer to File Browser.

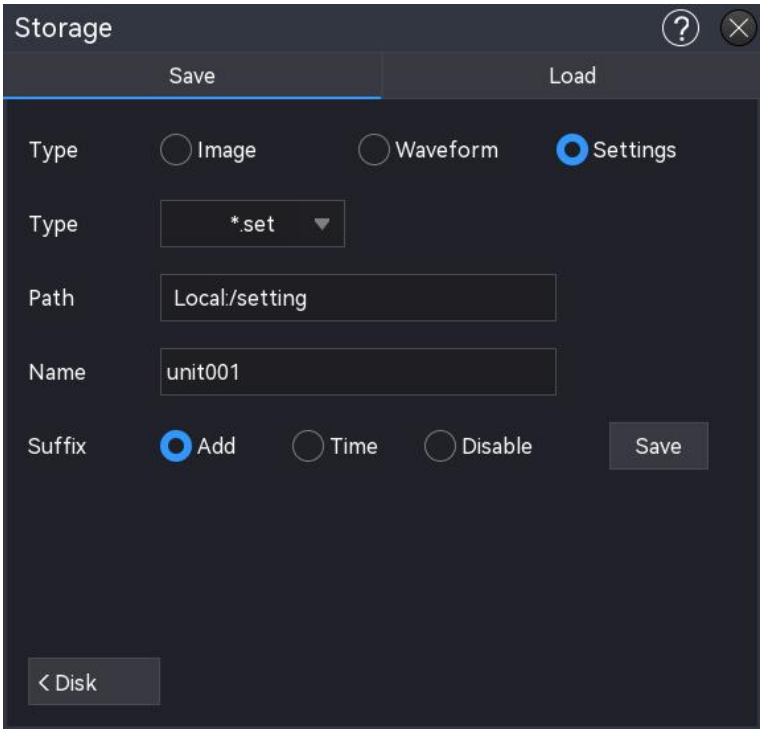
(8) Save

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 5.8 Parameter Setting-Character String.

(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 _1.

(5) Disk Management

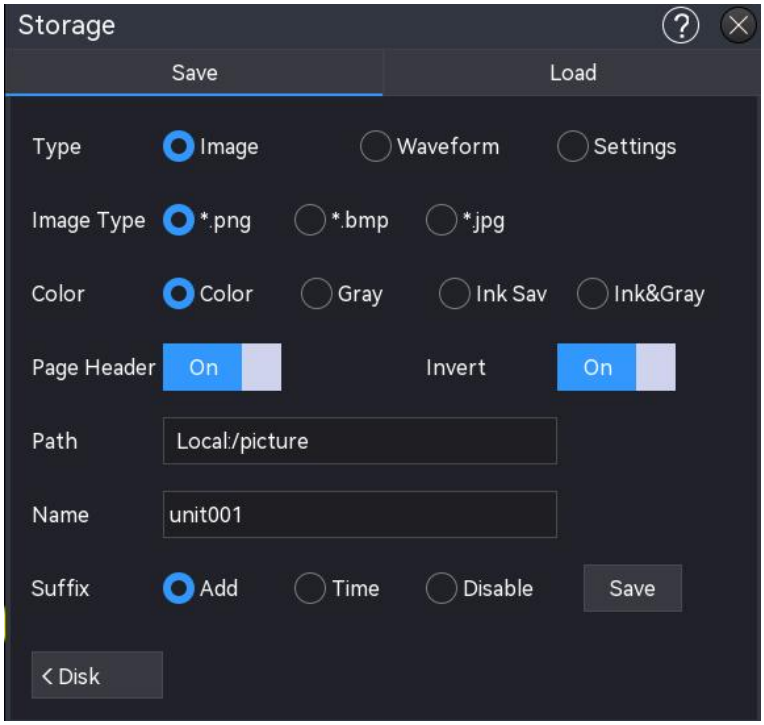
Click Disk to open the file browser. For details on using the file browser, refer to the section of File Browser.

(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 5.8 Parameter Setting-Character String.

(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 _1.

(8) Disk Management

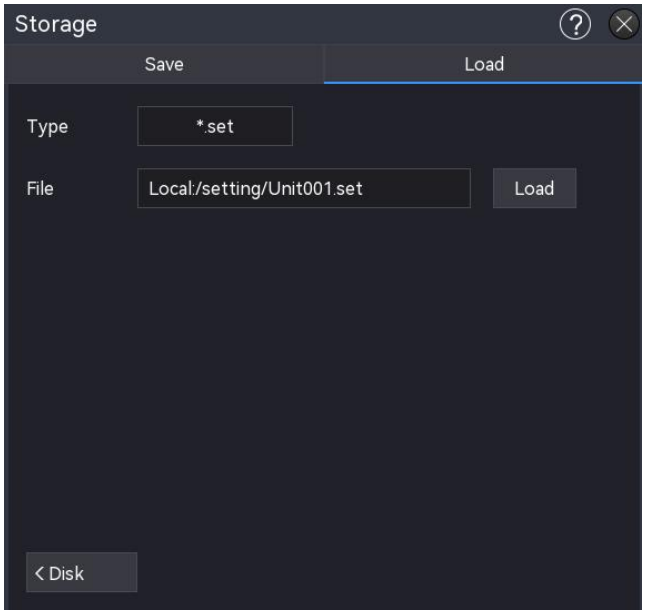
Click Disk to open the file browser. For details on using the file browser, refer to the section of File Browser.

(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

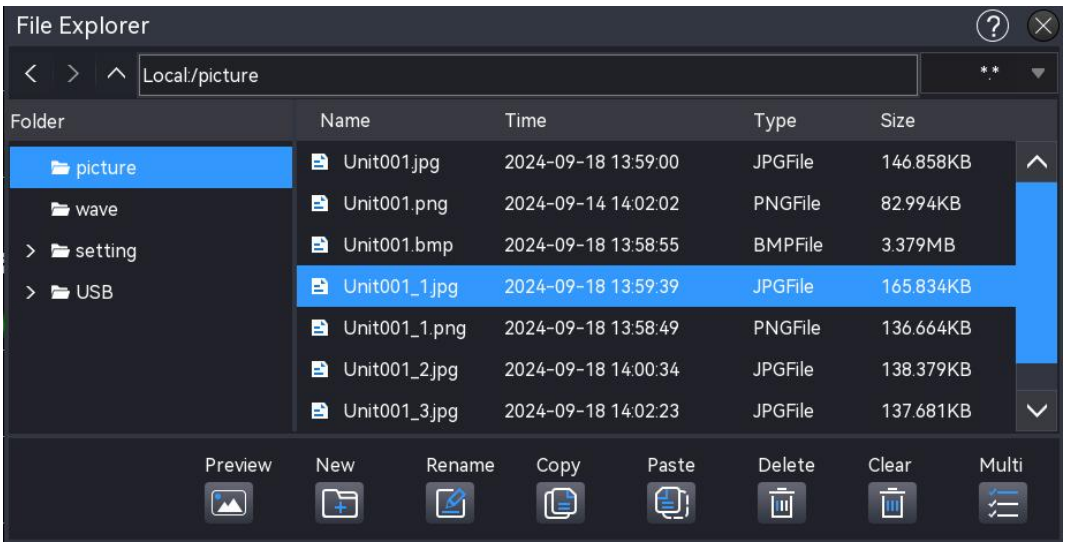
Click Load to load the selected setting file.

(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 5.8 Parameter Setting-Character String.

(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, .set, .pdf, or .html, *.* represents all file types.

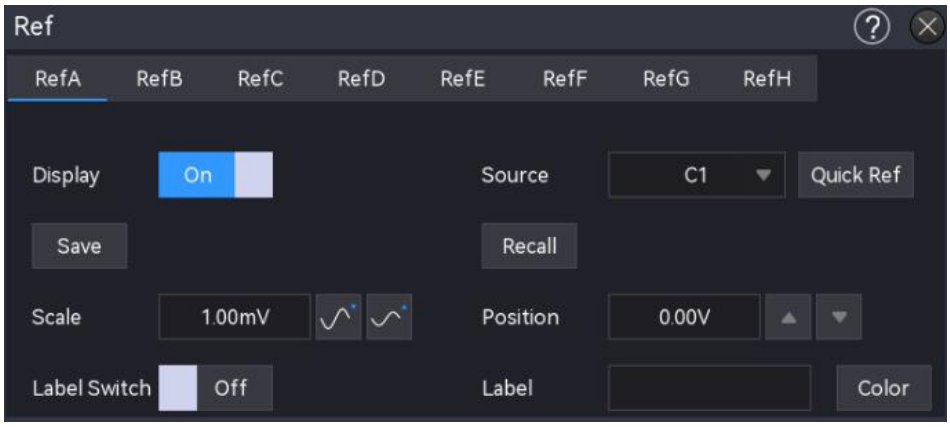
16. Reference Waveform

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 methods.

- Press the Ref button on the front panel to open the reference waveform menu.
- Click Home icon in the top right-corner, then select the reference icon to open the reference waveform menu.
- If the reference waveform is added to the toolbar, click reference icon in the top-right corner of the toolbar to enter the reference waveform menu.

This oscilloscope provides 4 reference waveforms (RefA - RefH), 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 - C8， M1 - M8 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 5.8 Parameter Setting-Character String.

(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.

- In Ref menu, click Vertical Scale input field, then use the Multipurpose rotary knob to set the vertical scale.
- Tap the icon ,  on the right of the vertical scale to increase or decrease the vertical scale.
- Double-click Vertical Scale input field to open the numeric keypad and enter the vertical scale.

16.7. Vertical Position

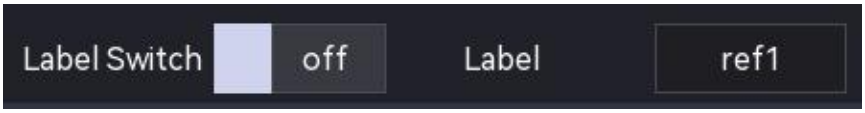
Set the vertical position of Ref wave in the display window, it can be set using the following steps.

- In Ref menu, click Vertical Position input field, then use the Multipurpose rotary knob to set the vertical position.

- Tap the icon , on the right of the vertical position to increase or decrease the vertical position.
- Double-click Vertical Position input field to open the numeric keypad and enter the vertical position.

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.

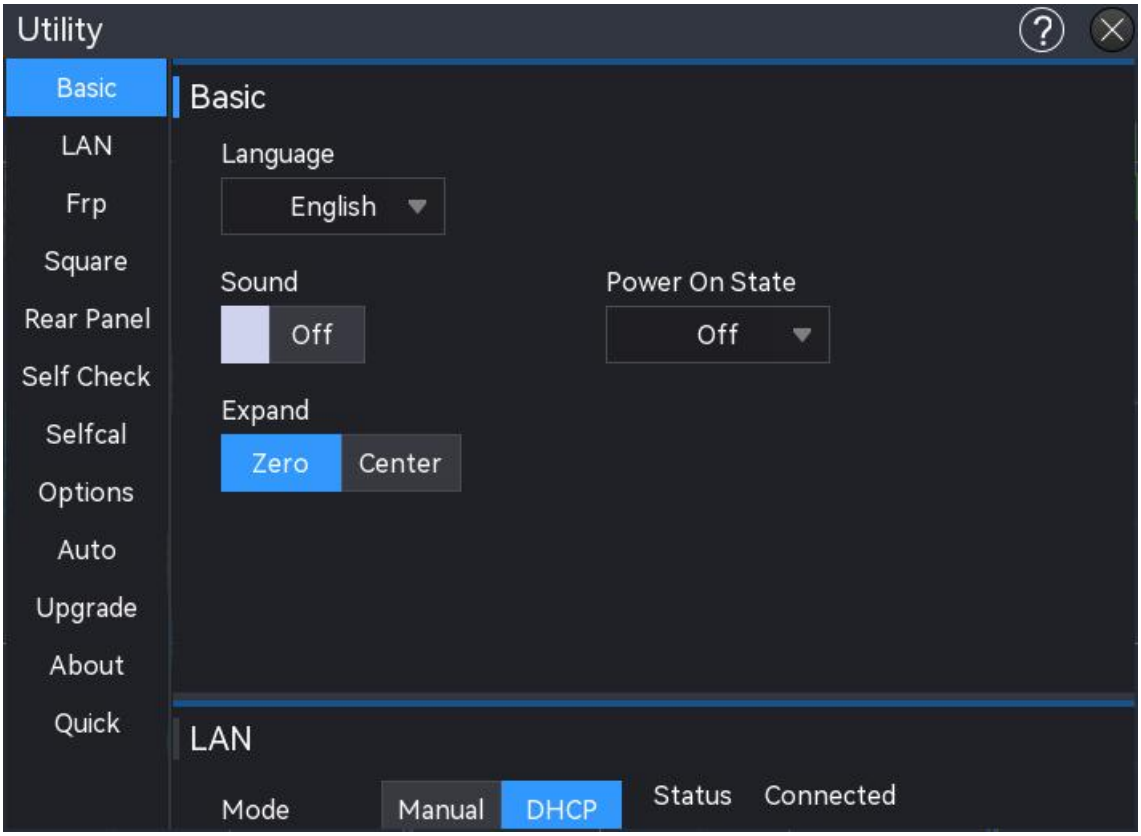
- Source: Click Source to select the source to set the color, the source can select M1 - M8, R1 - R8, FFT1 - FFT4
- Color: Tap the color plate and drag to select the color.

17. Utility Function

Set the system function in the Utility menu.

Access the utility menu using the following methods.

- 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 top-right corner of the toolbar 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. the buzzer will sound during the following operations:

- Pressing a button on the front panel
- Using the touchscreen
- When a message pops up

(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. Otherwise, it can only be powered on by pressing the soft power button on the front panel.
- Always on: When the rear-panel power switch is turned on, the oscilloscope starts automatically.
- 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. The 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	10 Hz, 100 Hz, 1 kHz, 10 kHz, 100 kHz, 3V_REF
Port 2	10 Hz, 100 Hz, 1 kHz, 10 kHz, 100 kHz

17.5. Rear Panel

In the Utility Menu, click Rear Panel or drag the content upward to access the rear panel settings.

(1) 10MHz Sync

- Idle: The [10MHz 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 [10MHz REF In&Out] connector on the rear panel is used as a reference clock input.
- Output: The [10MHz REF In&Out] connector on the rear panel is used as a reference clock output.

(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

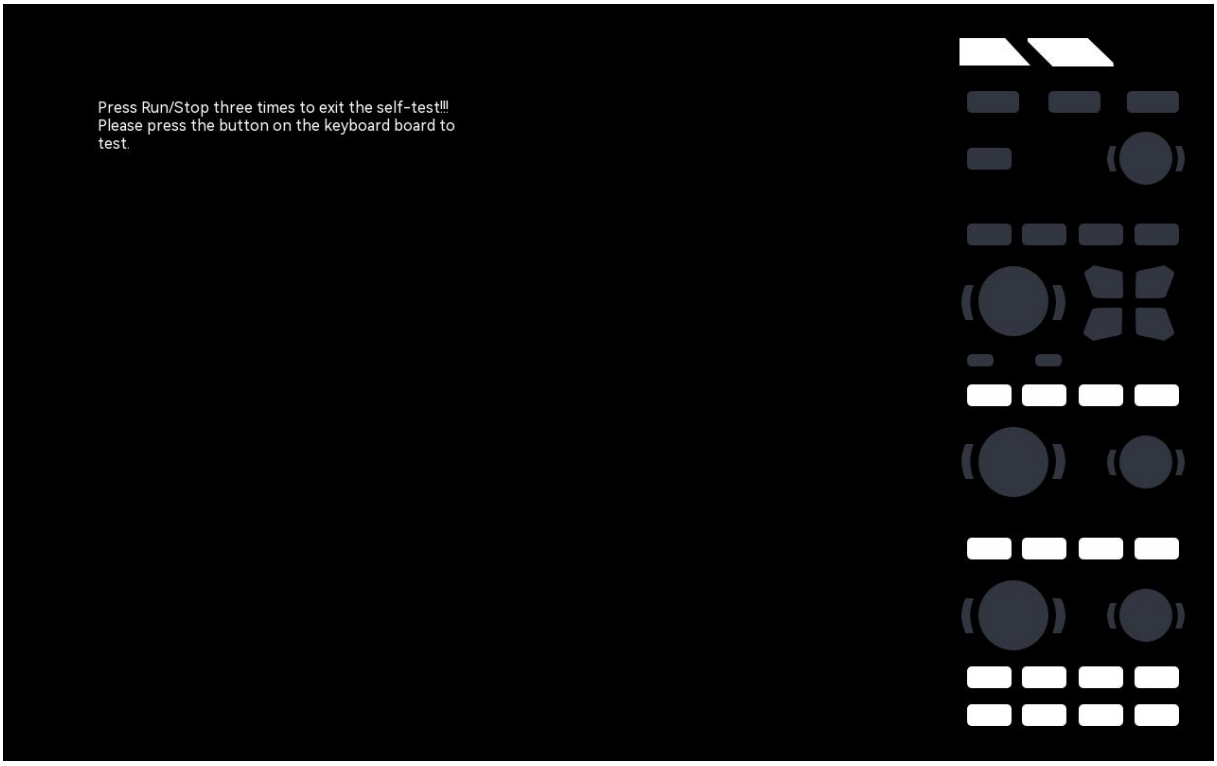
Out] connector when the oscilloscope detects a pass or fail event.

- 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. Self-Inspection

(1) Keyboard Self-Inspection

Keypad detection is mainly used to detect when the front panel keys or knobs of the oscilloscope are not responding or are not responding sensitively. Press the self-check key, the oscilloscope will enter the following figure.



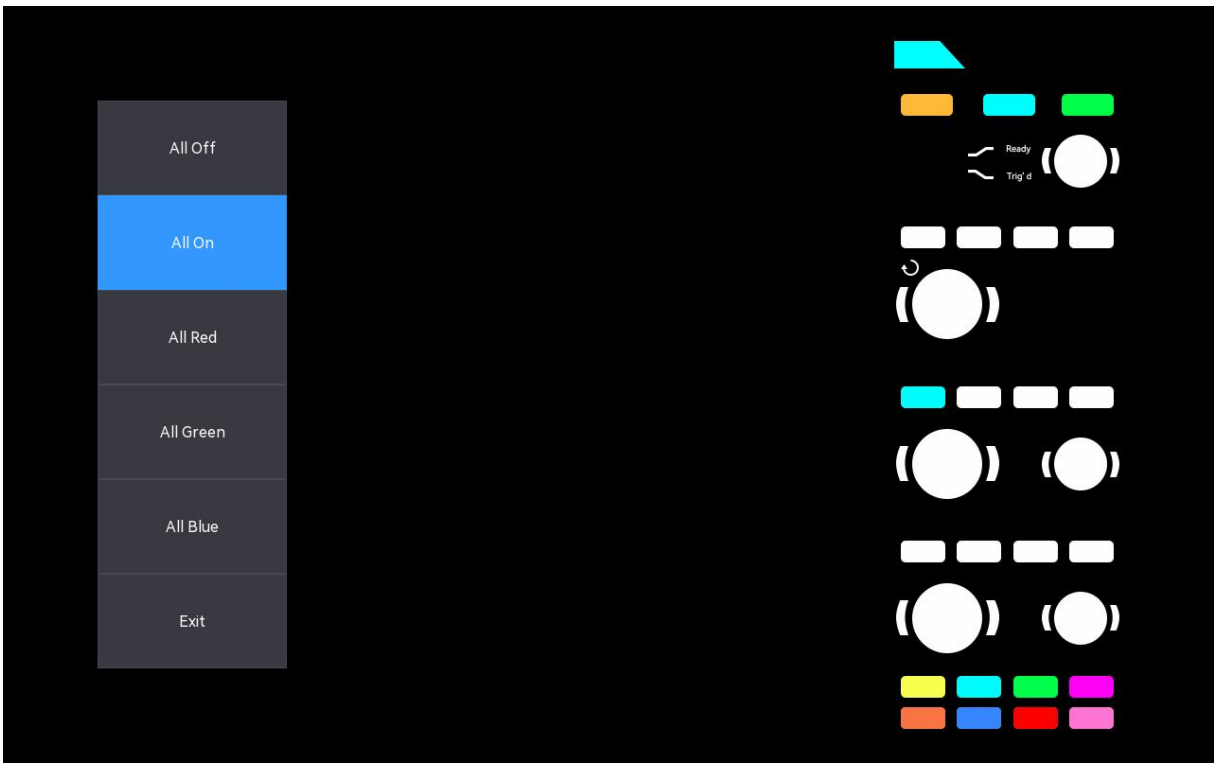
Rotary knob test: Rotate and press each rotary knob from up to down, left to right, observing the rotary whether the rotary knob indicator is lit on the display interface.

Key test: Rotate and press each key from up to down, left to right, observing the rotary whether the key indicator is lit in real time on the display interface.

When all rotary knobs and keys have been tested, press the Run/Stop keys three times to exit the keyboard test in accordance with the on-screen instructions.

(2) LED Detection

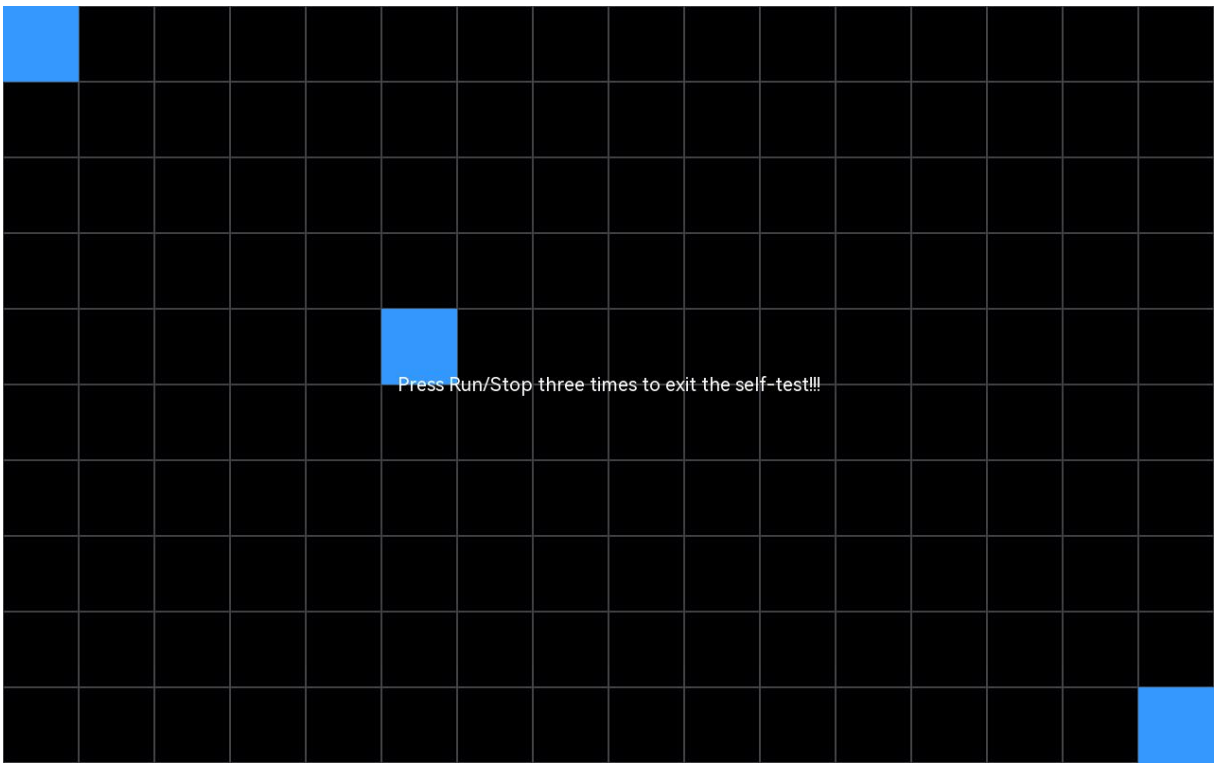
LED detection is mainly used to check whether the key indicator on the front of the oscilloscope can be lit or not, and whether the brightness is poor. When the LED test is pressed, the oscilloscope enters the interface is shown in the following figure.



- All Off: Turn off the LED lights of all keys and knobs.
- All On: Turn on the LED lights of all LED-equipped keys and knobs.
- All Red: Turn on the LED lights of all LED-equipped keys and knobs; set the LED color of knobs and channels to red.
- All Green: Turn on the LED lights of all LED-equipped keys and knobs; set the LED color of knobs and channels to green.
- All Blue: Turn on the LED lights of all LED-equipped keys and knobs; set the LED color of knobs and channels to blue.
- Exit: Exit LED detection.

(3) Touch Screen Detection

Touch screen detection is mainly used to check whether the touch screen is invalid or not responding in time. Press the touch screen detection, the oscilloscope enters the interface is shown in the following figure.



Touch each grid on the touch screen to observe if the grid turns blue.

When all rotary knobs and keys have been tested, press the Run/Stop keys three times to exit the touch screen test in accordance with the on-screen instructions.

(4) Screen Detection

Screen detection is used to check whether the oscilloscope screen has color offset, bad dots, or screen scratches. Press the screen detection, the oscilloscope enters the interface is shown in the following figure. The interface displays pure red.



Press any key on the right panel to switch to the red, green, blue, black, and white according to the on-screen instruction. Observe the screen under the appropriate surface of each color to see if there are any serious problems such as color differences, stains or scratches.

When the color test has been tested, press the Run/Stop keys three times to exit the touch screen test in accordance with the on-screen instructions.

17.7. 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.

17.8. About Oscilloscope

Click About in the Utility menu to check the oscilloscope information.

- Product model
- Serial version number
- IPK version number
- Logic version number
- Hardware version number
- Software version number
- Keypad version number
- Power board version number
- File system version number

Factory Reset: Click Reset to restore the oscilloscope to its factory default settings. This will delete all locally stored files.

17.9. Option

Click Options in the Utility menu to view all available options.

All options support a 540-hour trial period. After the trial expires, users must purchase the corresponding option and obtain a software license to continue using it.

(1) Software License

To obtain a license for a purchased option:

- Purchase the option key from an authorized distributor.
- Visit the UNI-T official website: Test Instrument > Service and Support > License Activation to access the software license activation page.
- On the activation page, enter the purchased key, device serial number, and verification code as prompted. The software license will be generated immediately.

(2) Activate Software License

To install the software license on the device:

- Download the license from the License Information list, save it to the USB root directory, and insert the USB drive into the device.
- Navigate to Utility > Options, then click Activate or Activate All to apply the software license.
- To verify installation, open Utility > Options. In the option trial table:
  Activation Status will display Activated; Trial Duration will display Permanent

Note: If you are registered and logged in to the UNI-T website and provided your email during registration, the generated software license will be sent as an email attachment.

The software license can also be viewed and downloaded again from the license activation page after logging in to the UNI-T website.

UPO6000HD series supports the following optional functions, as listed in the table below.

Model	Option
UPO6000HD Series	CAN, CAN-FD, LIN, FlexRay, SNET, Audio, 1553B, Manchester, ARINC429, PWR, Timing Analysis, and Bandwidth Upgrade

17.10. Auto Setting

Click Auto setting in the Utility menu to enter the auto setting menu.

(1) Channel Setting

- Hold mode: The following settings remain unchanged: bandwidth limit, inverse phase, impedance, unit, probe attenuation ratio, and label state. The following settings are reset to default: coupling (ground), vertical scale, offset, fine-tuning, and channel fold.
- Auto mode: Impedance, unit, probe attenuation ratio, and label state remain unchanged, while all other settings are reset to default.

(2) Acquisition Setting

- Current mode: All settings remain unchanged.
- Auto mode: The acquisition mode is reset to default; all other settings remain unchanged.

(3) Trigger Source

- Current mode: The source and trigger coupling remain unchanged; all other settings are reset to edge trigger, auto mode, and rising edge.
- Auto mode: All parameters are reset to default.

(4) Activate Channel

- Current mode: The switch state on the rear panel remains unchanged when the channel is automatically set.
- Auto mode: The corresponding channel is activated if a signal is present. If no signal is detected, C1 is activated by default.

17.11. Quick Operation

Click Quick setting in the Utility menu to enter the quick operation menu.

(1) Quick Operation

Quick operation can assign specific functions to button on the front panel, including screenshot, save waveform, and save setting.

- Screenshot: Press the button  to save a screenshot to the preset path. The image format, color mode, and file extension follow the settings configured on the Save Image section.
- Save waveform: Press the button  to save channel waveforms to the preset path. The waveform format, signal source, and file extension follow the settings configured on the Save Waveform section.
- Save setting: Press the button  to save the setup file to the preset path. The file extension follows the settings configured on the Save Setting section.

(2) User-Defined

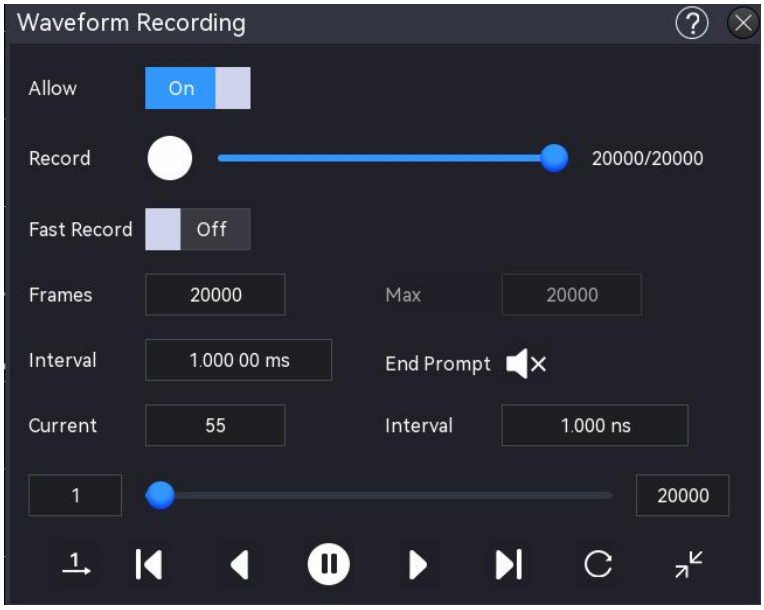
User mode: Select any function from the full function list and assign it to button on the front panel key. All available functions are supported for assignment.

18. Waveform Recording

The Waveform Recording and Playback function enables playback of recorded waveforms, facilitating waveform analysis.

Access the Waveform Recording menu using the following methods.

- Press the Analyze button on the front panel, then select Waveform Recording from the Analyze menu to open the waveform recording menu.
- Click Home icon in the top-right corner of the screen, then click waveform recording icon to enter the waveform recording menu.
- If the waveform recording is added in the toolbar, click waveform recording icon in the top-right corner of the screen 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 to 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 Recording Setting section.

During playback, Current Frame will change in real time. Click pause icon to stop playback manually.

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 5.8 Parameter Setting. Alternatively, click Frames input field, then use the Multipurpose rotary knob to adjust the frame. 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 memory depth ⩾ 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 5.8 Parameter Setting. Alternatively, click Interval input field, then use the Multipurpose rotary knob to adjust the recording interval. The setting range is from 0 s to 10 s

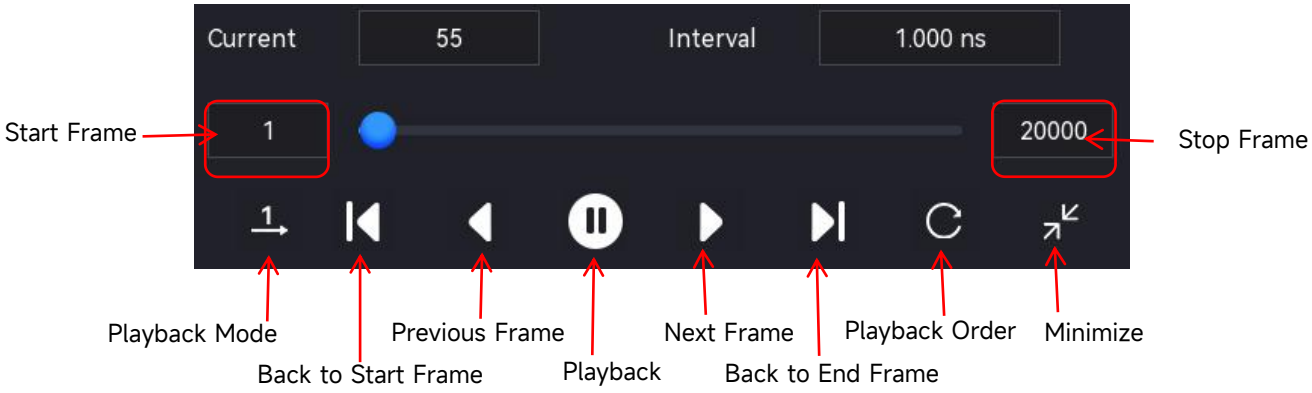
(5) End Sound

: The beeper emits a sound when recording ends.

: The beeper remains silent when recording ends.

18.2. Playback Setting

Before waveform recording, the following playback settings 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 5.8 Parameter Setting. Alternatively, click Current input field, then use the Multipurpose rotary knob to adjust the current frame. 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 5.8 Parameter Setting. Alternatively, click Playback Interval input field, then use the Multipurpose rotary knob to adjust the playback interval. 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 5.8 Parameter Setting. Alternatively, click Start Frame input field, then use the Multipurpose rotary knob to adjust the start frame. The default setting 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 5.8 Parameter Setting. Alternatively, click Stop Frame input field, then use the Multipurpose rotary knob to adjust the stop frame. The default setting is the total number of frames in the recorded waveform.

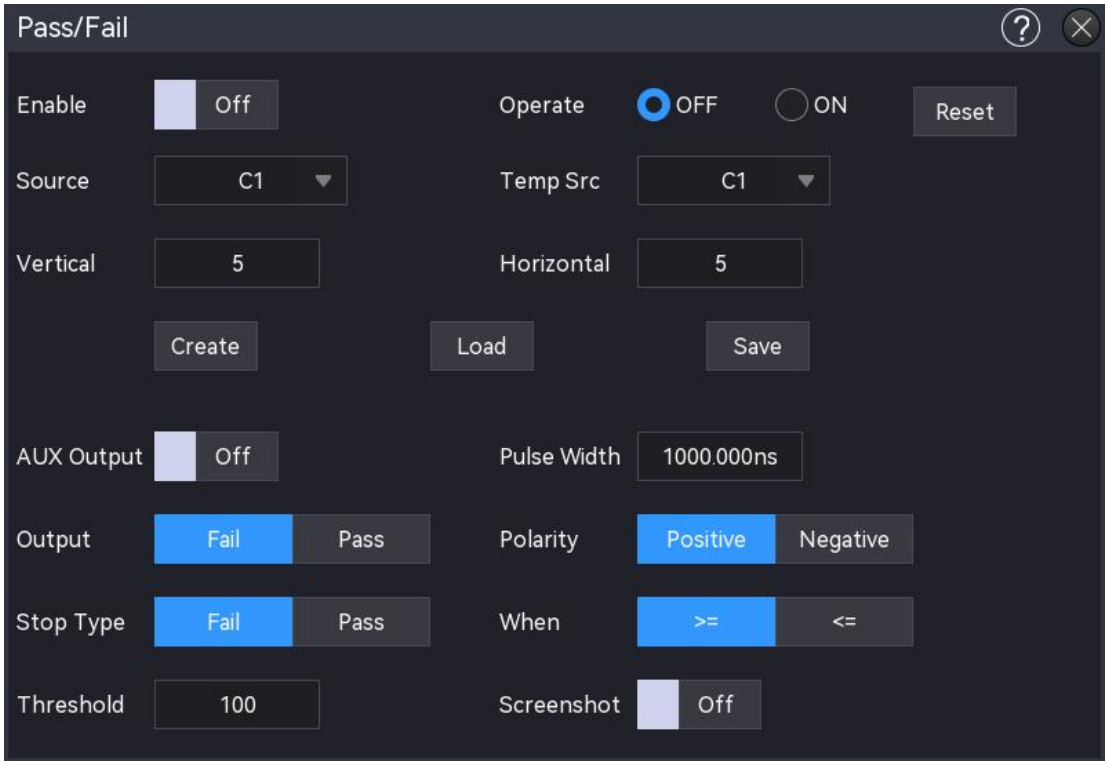
19. Pass/Fail Test

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 a pass or fail is detected, the system can stop monitoring, trigger a buzzer, and save the current screenshot.

Access the Pass/Fail Test menu using the following method.

- Press the Analyze button on the front panel, then select Pass/Fail from the Analyze menu to open the pass/fail menu.
- 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 top-right corner of the screen 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

Click Pass/Fail to turn the pass/fail test ON or OFF.

19.2. Source

Click Source to select C1 - C8 as the trigger source.

19.3. Mask Setting

(1) Mask Source

Click Template Source to select the source for creating a mask. The source can choose from C1 - C8 and Ref. When Ref is selected, a reference waveform is required to generate the mask.

(2) Vertical Setting

Click Vertical input field, then use the Multipurpose rotary knob to set the vertical tolerance. The setting range is from 1 to 100.

(3) Horizontal Setting

Click Horizontal input field, then use the Multipurpose rotary knob 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 (only when a 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 (only when a 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 section of Storage and Load.

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 5.8 Parameter Setting. Alternatively, click Output Pulse Width input field, then use the Multipurpose rotary knob to adjust the output pulse width. 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 5.8 Parameter Setting. Alternatively, click Condition Time input field, then use the Multipurpose rotary knob to adjust the condition time. 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

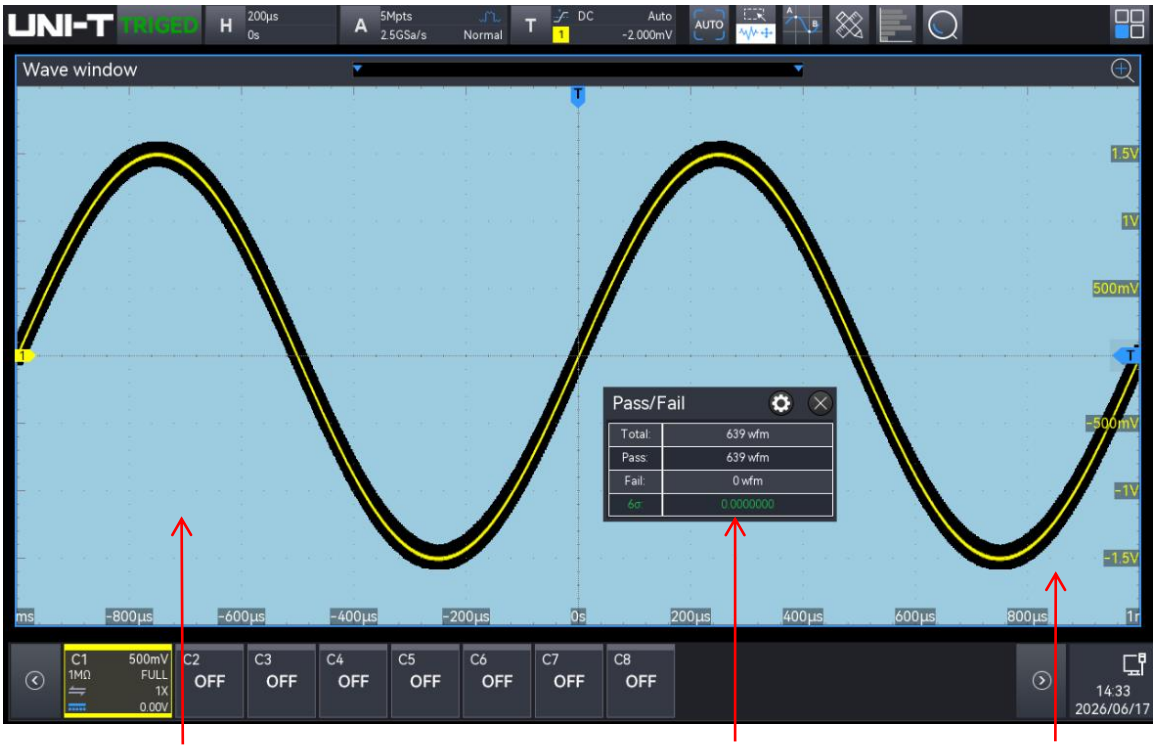
(1) Operation

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.



Blue area is mask area

Test Result

Test waveform:
If it intersects with the masked area, the test will be considered a failure.

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.

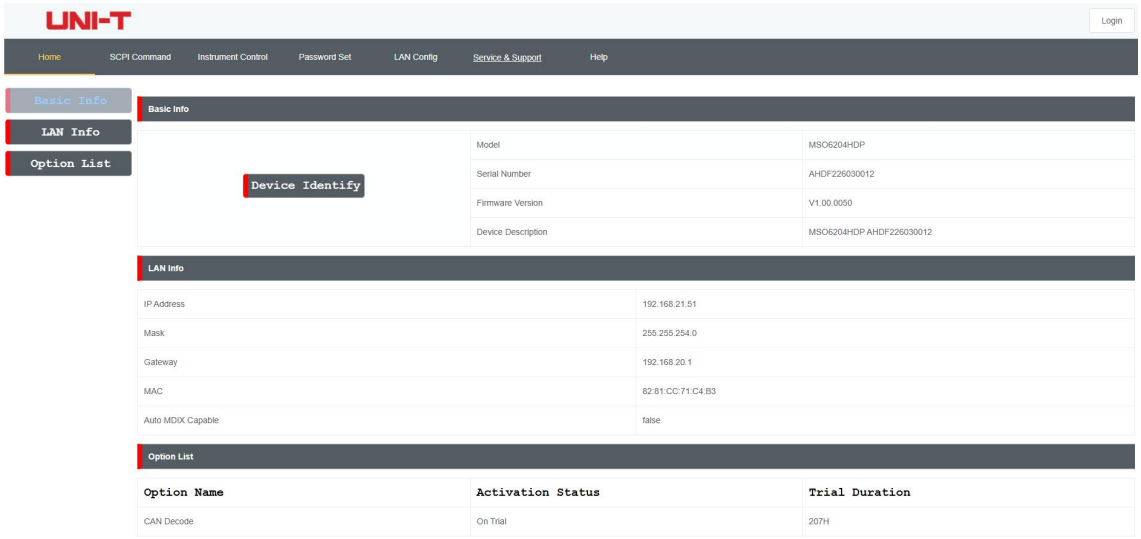
20. Web Access

20.1. 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.

20.2. 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.

20.3. 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.

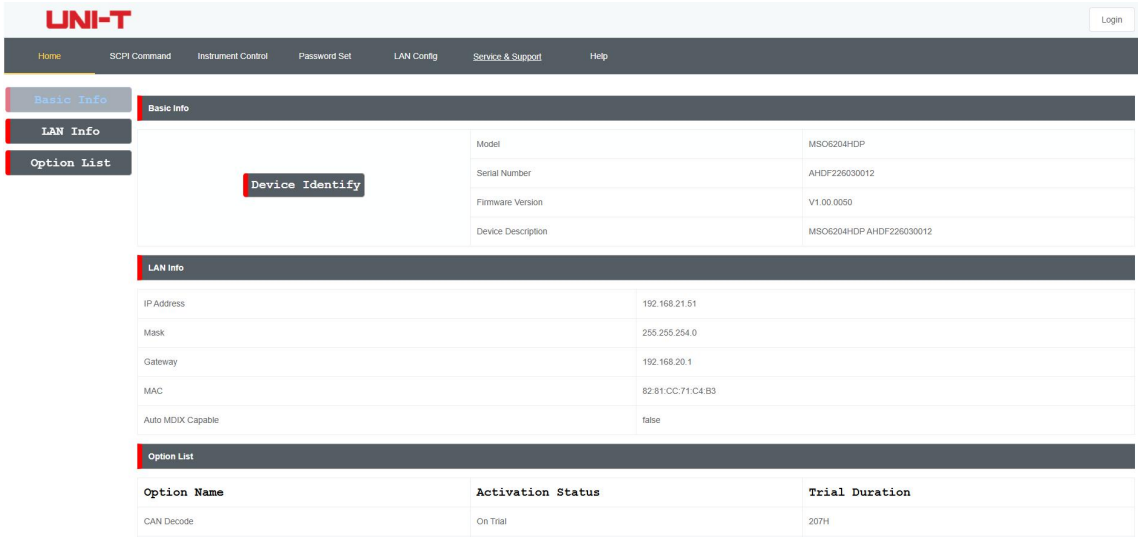
c. Configure the frp proxy IP and port, image port, and control port of the oscilloscope.

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”.

20.4. 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.

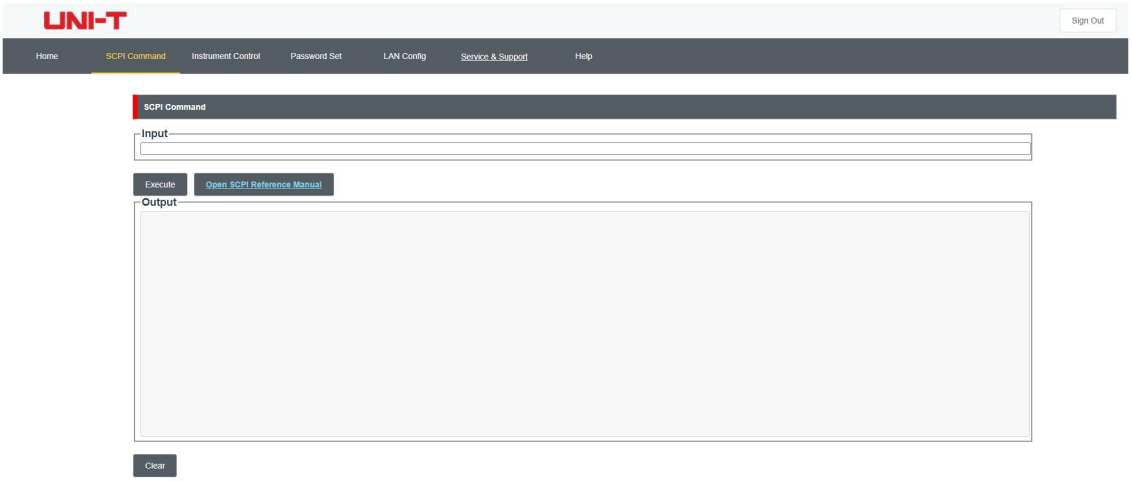


20.5. SCPI

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.

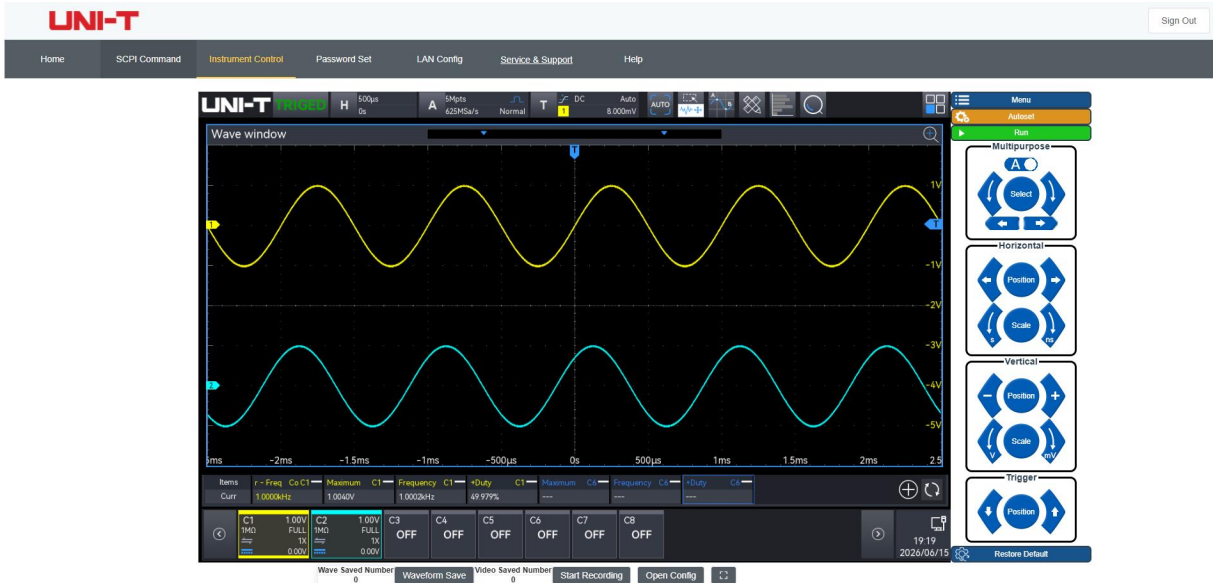
- Open SCPI programming manual: Click SCPI to access the SCPI programming manual for this oscilloscope.
- Clear: Clears the contents of the SCPI output window.



20.6. Instrument Control

The Instrument Control page of the web interface allows remote operation of oscilloscope functions. All touchscreen operations can be performed using a mouse via the web page.

- Click and drag to move waveforms vertically and horizontally, adjust the trigger level position, and reposition pop-up windows.
- Use the mouse wheel or click the virtual knobs to adjust horizontal and vertical waveform positions, modify the time base and Volts/Div settings, switch between coarse and fine adjustment modes, and adjust the trigger level.
- Use mouse clicks to open or close drop-down lists and pop-up dialogs, and to switch between different menu options.
- Double-click screen to open the numeric keypad, alphabetic keypad, and other on-screen keyboards.



(1) Key Area

- Menu: Click Menu key to open the auxiliary menu window box.
- Auto: Click Auto key to perform the Autoset operation.
- Run/Stop: Click Run/Stop key to change the oscilloscope's operating state.
- Reset: Click Restore Default to restore the settings to factory defaults.

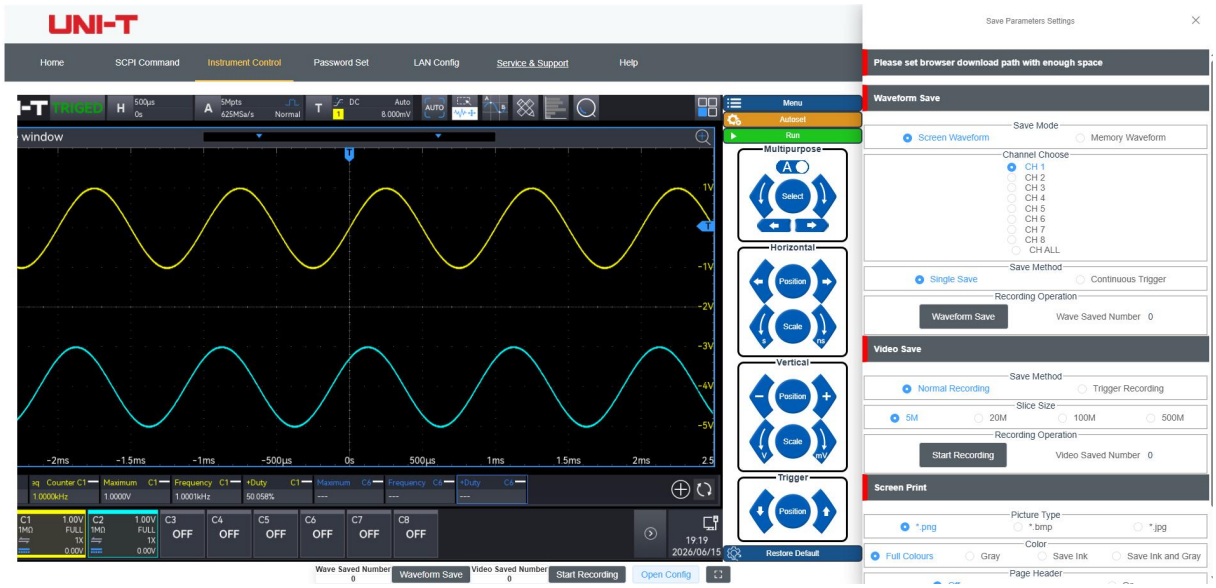
(2) Rotary Knob

- 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 vertical position to center.
- 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 - C8, 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 Storage 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.

20.7. Network Setting

The network information configuration page sets the oscilloscope's network parameters, including LAN settings and external network proxy configuration.

a. Oscilloscope Network Information Settings

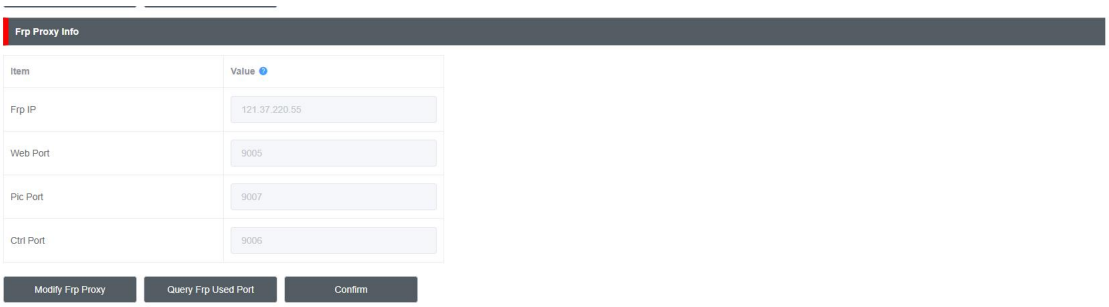


Click Modify Oscilloscope Configuration to configure the oscilloscope's local network settings, including IP acquisition mode (DHCP/STATIC), IP address, subnet mask, and gateway.

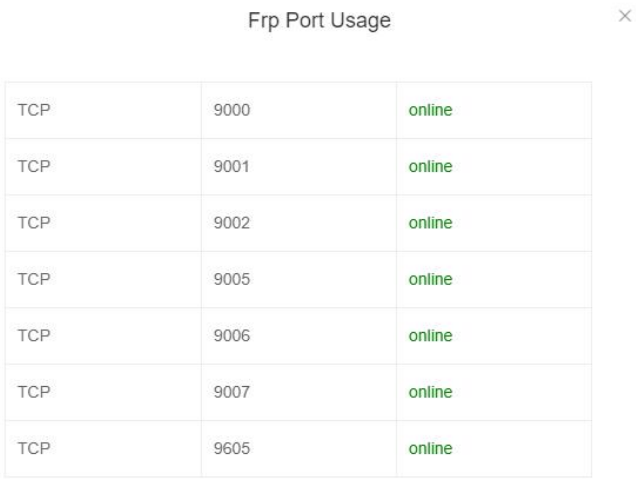
- DHCP: When DHCP is selected as the IP configuration method, there is no need to enter the IP address, subnet mask, or gateway. Click Confirm, and the oscilloscope will automatically obtain an IP address.
- STATIC: When STATIC is selected as the IP configuration method, enter the correct IP address, 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.



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 button on the front panel. After reset, LAN access via port 9000 will be restored.

20.8. 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 button on the front panel to reset the configuration.

Note: When the oscilloscope is restored to default settings, the Web system must be accessed for the first time using the Base64-encoded serial number as the password.



20.9. Help

Click Help page to open the Web usage help page, which provides basic instructions for each functional tab of the oscilloscope Web interface.

20.10. Service and Support

Click Service and Support page to be directed to the UNI-T official website of https://www.uni-trend.com/

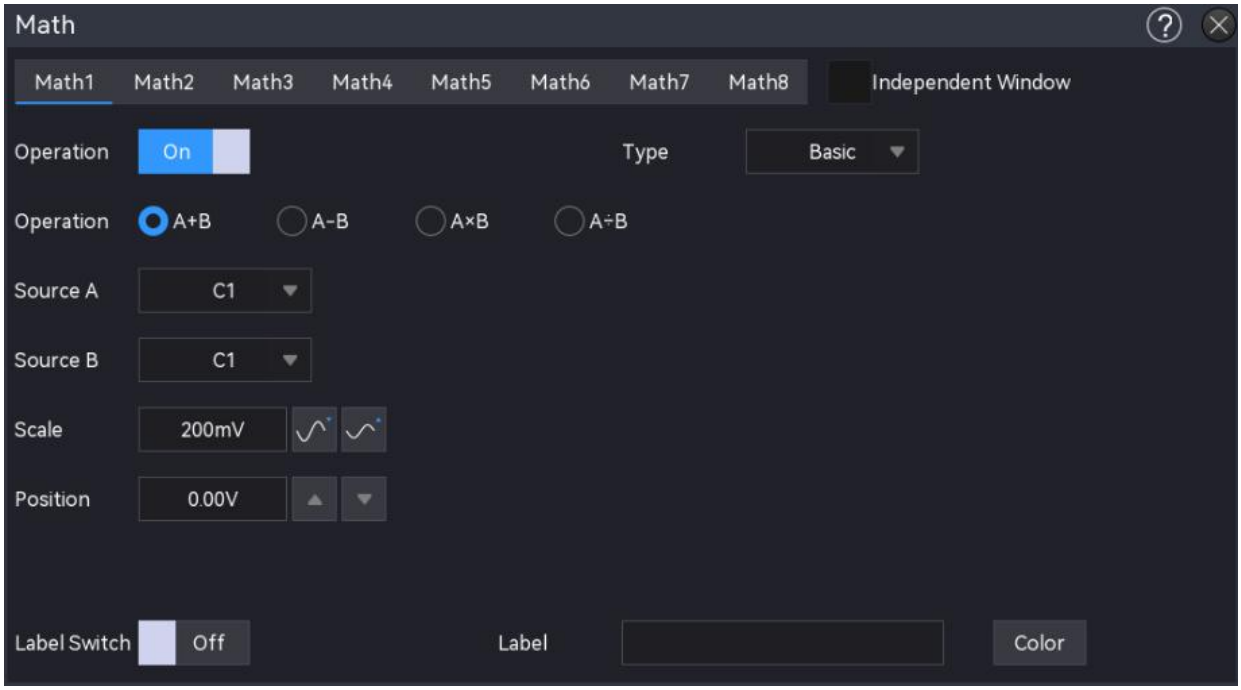
21. Mathematical Operation

- Basic Operation
- Digital Filter
- Advanced Operation

UPO6000HD series high-resolution oscilloscope carries a variety of mathematical operations, including Math, digital filter, and advanced operation.

Access the Math menu using the following methods.

- Press the Math button on the front panel to open the math setting menu.
- Click Home icon in the top right-corner and select the Math icon to open the math setting menu.6
- If the math function is added into the toolbar, click math icon in the top-right corner of the screen to enter the math setting menu.
- When M1 - M8 is opened, click M1 - M8 label at the bottom of screen, or click icon in the top right-corner to enter the math setting menu.



This oscilloscope supports 8 mathematical operations: Math1, Math2, Math3, Math4, Math5, Math6, Math7, and Math8. 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–M8 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–M8 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 ☑: 8 Math waves and the channel wave are displayed in a separate window.

Uncheck ☐: 8 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.

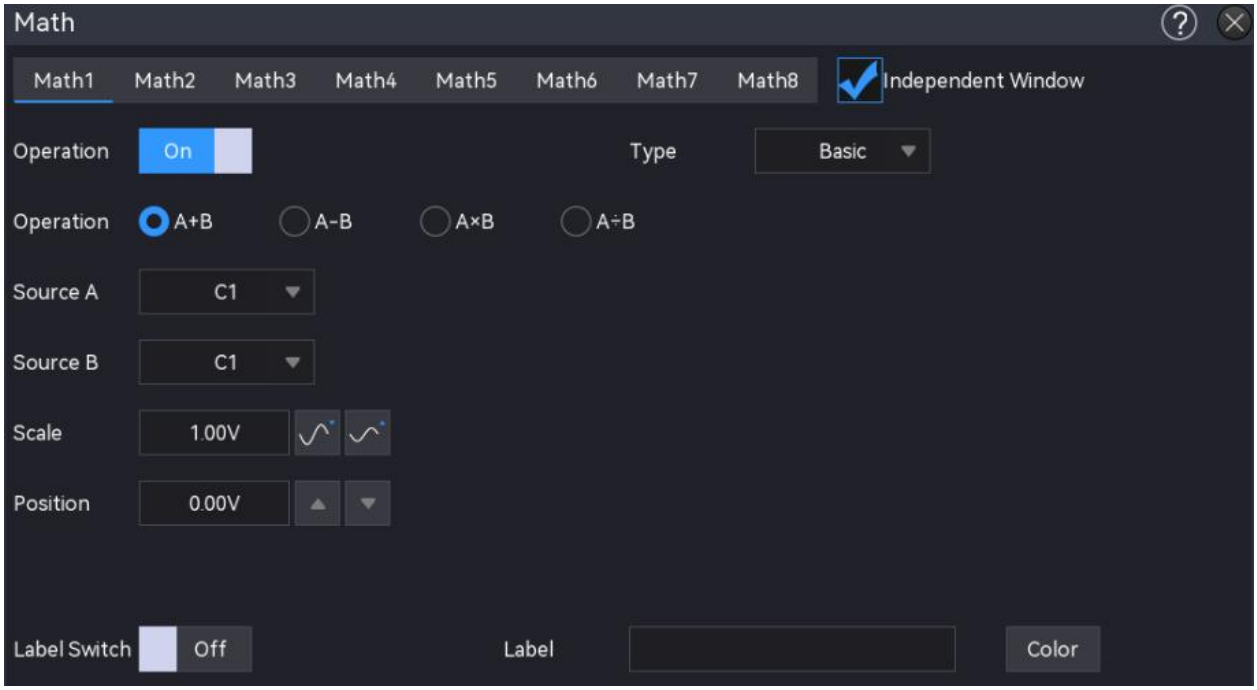
(4) Channel Color

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 - M8, R1 - R8, FFT1 - FFT4.
- Color: Tap the color plate and drag to select the color.

21.1. 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 - C8 separately.

(3) Vertical Scale

Set the vertical scale of Math wave in the display window, it can be set using the following steps.

- In the Math menu, click Scale input field, then use the Multipurpose rotary knob to adjust the vertical scale.
- Tap the icon , on the right of the vertical scale to increase or decrease the vertical

scale.

- Double-click Scale input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.

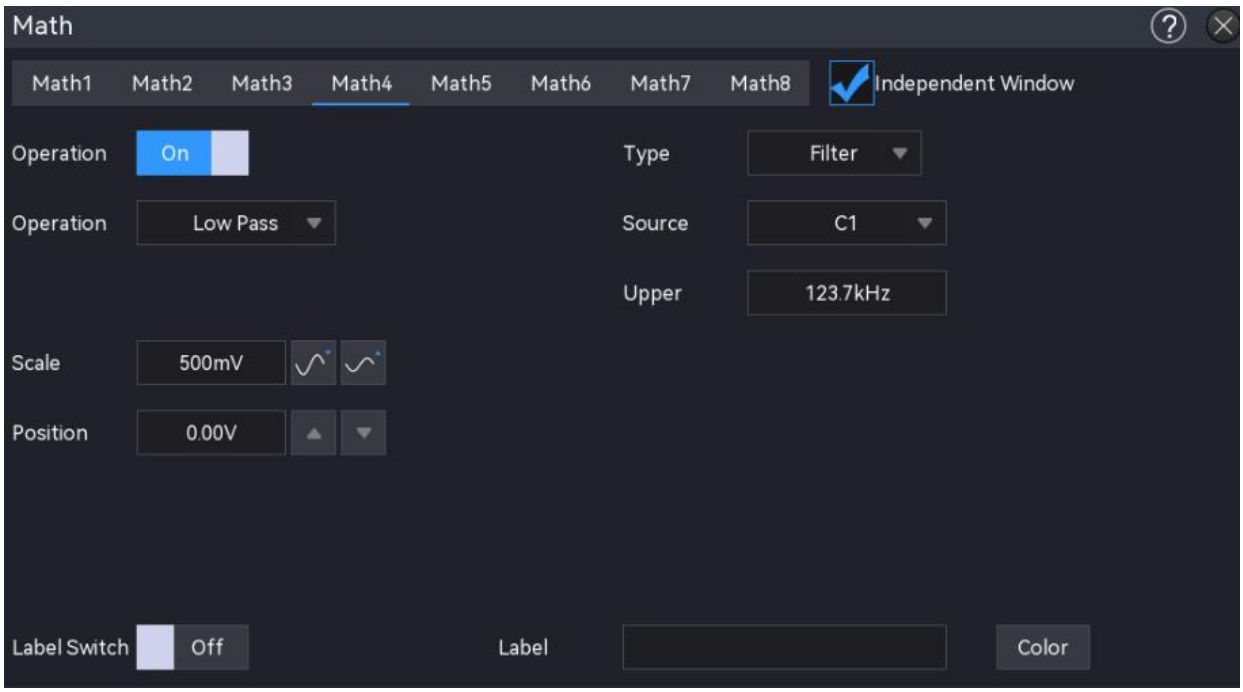
(4) Vertical Position

Set the vertical offset of Math wave in the display window, it can be set using the following methods.

- In the Math menu, click Position input field, then use the Multipurpose rotary knob to adjust the vertical position.
- Tap the icon , on the right of the vertical position to increase or decrease the vertical position.
- Double-click Position input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.

21.2. Digital Filter

In the Math menu, click Filter to open the setting.



(1) Source

Click Source to select C1 - C8 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

Click Lower input field, then use the Multipurpose rotary knob to adjust the lower frequency limit. Alternatively, double-click Lower input field to open the numeric keypad and set the lower frequency limit. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.

In low-pass mode, the lower frequency limit setting is invalid, and the menu is hidden.

(4) Upper Limit of Frequency

Click Upper input field, then use the Multipurpose rotary knob to adjust the upper frequency limit. Alternatively, double-click Lower input field to open the numeric keypad and set the upper frequency limit. For details on using the numeric keypad, refer to section 5.8 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 depends on 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 methods.

- In the Math menu, click Scale input field, then use the Multipurpose rotary knob to adjust the vertical scale.
- Tap the icon ,  on the right of the vertical scale to increase or decrease the vertical scale.
- Double-click Scale input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.

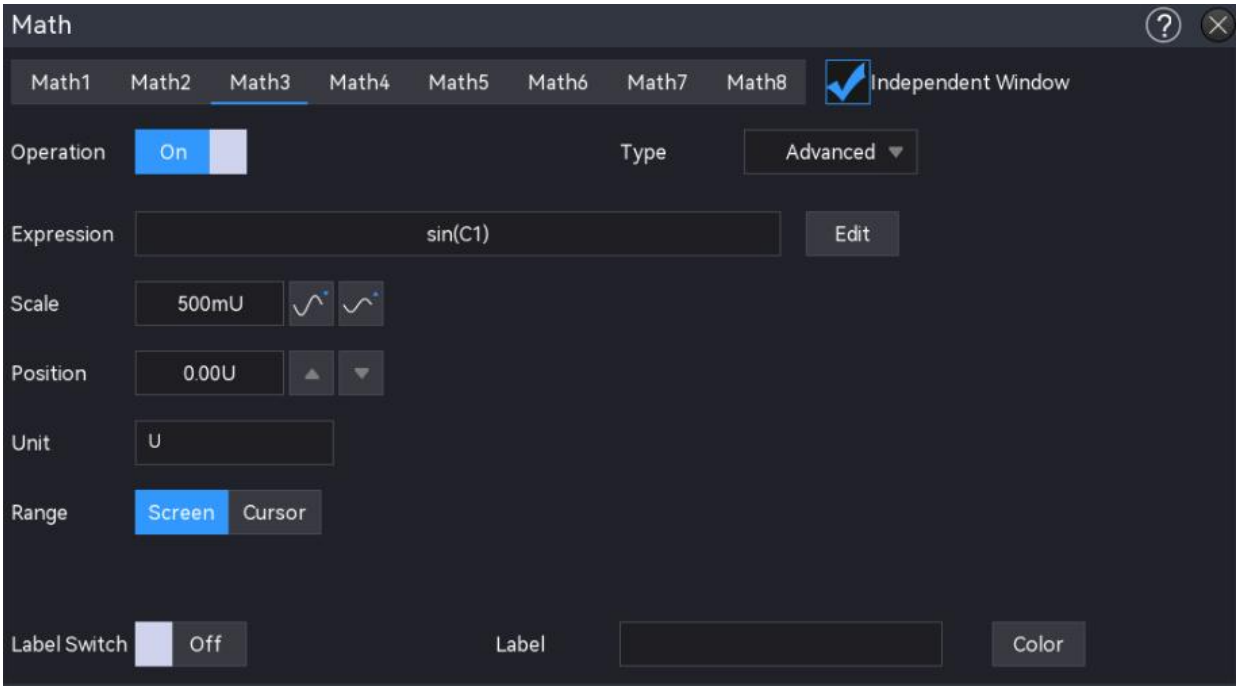
(6) Vertical Position

Set the vertical offset of Math wave in the operation result display window, it can be set using the following steps.

- In the Math menu, click Position input field, then use the Multipurpose rotary knob to adjust the vertical position.
- Tap the icon ,  on the right of the vertical position to increase or decrease the vertical position.
- Double-click Position input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.

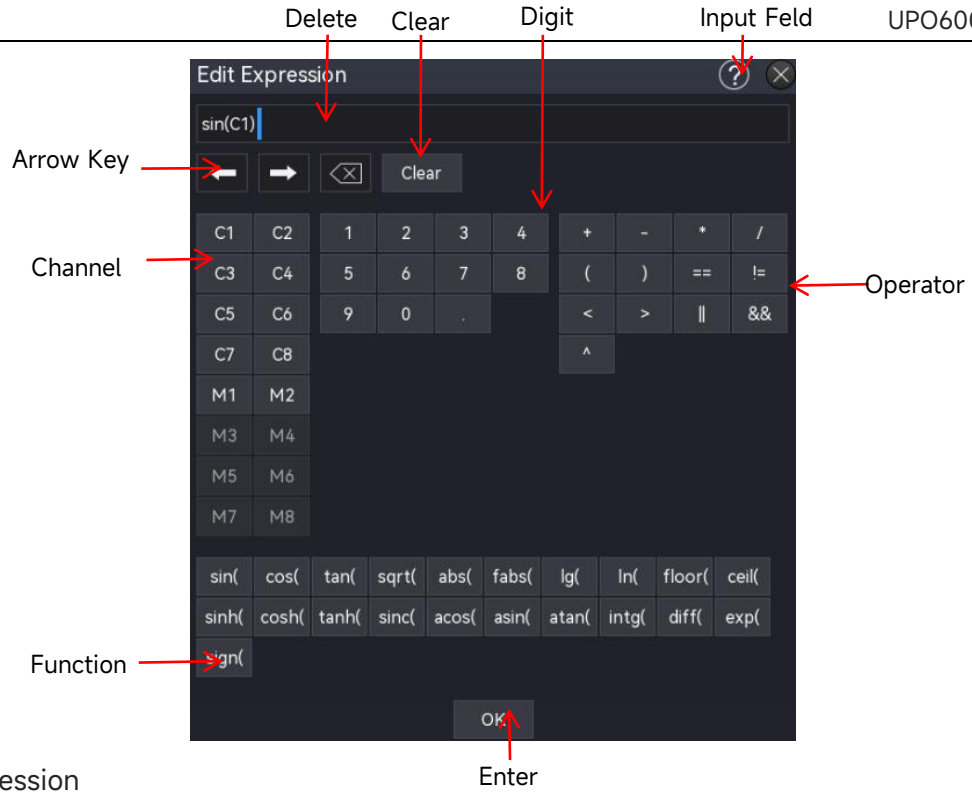
21.3. Advanced Operation

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.



(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 - C8, M1 - M8

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, π, e, .	Perform digit operation

(3) Vertical Scale

Set the vertical scale of Math wave in the operation result display window, it can be set using the following methods.

- In the Math menu, click Scale input field, then use the Multipurpose rotary knob to adjust the vertical scale.
- Tap the icon ,  on the right of the vertical scale to increase or decrease the vertical scale.
- Double-click Scale input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.

(4) Vertical Position

Set the vertical offset of Math wave in the operation result display window, it can be set using the following steps.

- In the Math menu, click Position input field, then use the Multipurpose rotary knob to adjust the vertical position.
- Tap the icon , on the right of the vertical position to increase or decrease the vertical position.
- Double-click Position input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.8 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 *Enter Character String* in the section of 5.8 Parameter Setting-Character String.

(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.

22. 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

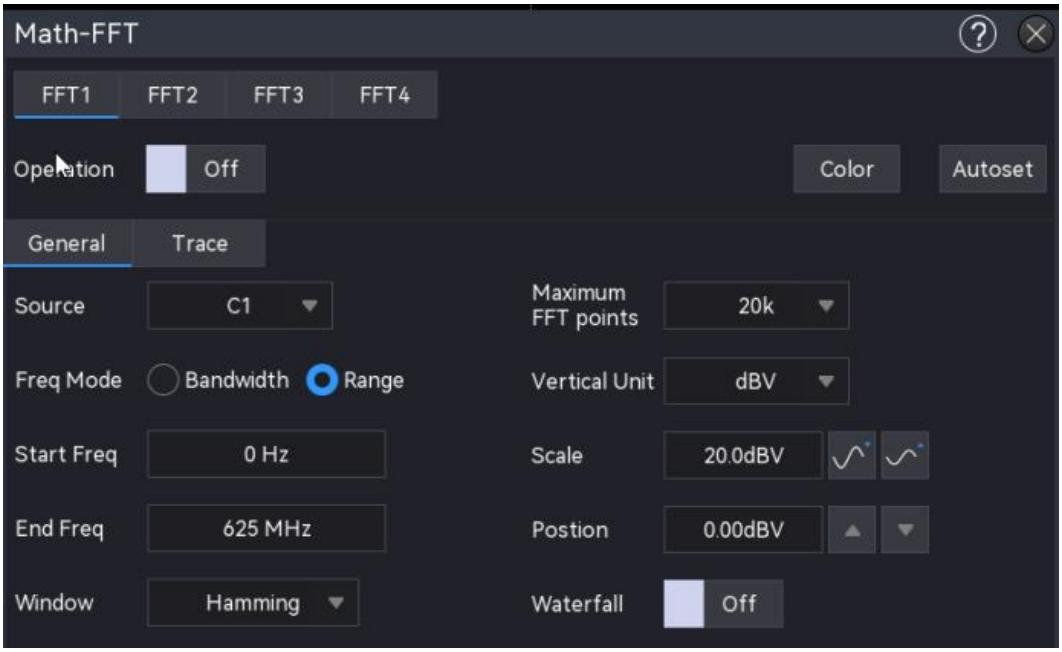
Access the FFT menu using the following methods

- 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 top-right corner of the screen 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 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 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 color for the Math channel, Ref waveform, FFT waveform, and label . Click on the “Channel Color” to enter the setting menu.

- Source: Click on the “Source” to select the source to set the color, the source can select M1 - M8, R1 - R8, FFT1 - FFT4.
- Color: Tap the color plate and drag to select the color.

(4) Source

Click Source to select C1 - C8, M1 - M8, RefA - RefH.

(5) FFT Count

The number of points processed by the FFT spectrum, it can be set to 1k, 2k, 5k, 10k, 20k, 50k, 100k, 200k, 500k, 1M, 2M, 5M, or 10M.

(6) Frequency Mode

a. Range

- Start frequency: Click Start Frequency input field to open the numeric keypad and set the start frequency. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. Alternatively, double-click Start Frequency input field, then

use the Multipurpose rotary knob to adjust the start frequency. Turn clockwise to increase and anticlockwise to decrease.

- Stop frequency: Click Stop Frequency input field to open the numeric keypad and set the stop frequency. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. Alternatively, double-click Stop Frequency input field, then use the Multipurpose rotary knob to adjust the stop frequency. Turn clockwise to increase and anticlockwise to decrease.

b. Bandwidth

- Center frequency: The frequency corresponds to the center of window. Click Center Frequency input field to open the numeric keypad and set the center frequency. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. Alternatively, double-click Center Frequency input field, then use the Multipurpose rotary knob to adjust the center frequency. Turn clockwise to increase and anticlockwise to decrease.
- Bandwidth: The frequency range of frequency domain waveform, set the bandwidth for FFT sweep. Click Bandwidth input field to open the numeric keypad and set the bandwidth. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. Alternatively, double-click Bandwidth input field, then use the Multipurpose rotary knob to adjust the bandwidth. Turn clockwise to increase and anticlockwise to decrease.

(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

The window function is used to reduce the problem of spectral leakage. This oscilloscope provides five FFT window functions, each window function is for different waveforms, as listed in the following table, and the selection is based on the measured waveform and its features. Click Window drop-down list to select the desired function.

Window Function Table

Window function	Feature	Waveform
Rectangle	Provides the best frequency resolution but the poorest amplitude resolution;	Transient or short pulse: The signal level returns to nearly the same value before and after the pulse.

	equivalent to using no window.	Equal-amplitude sine wave: A sine wave with nearly identical amplitude and frequency. Wideband random noise: Random noise with a broad frequency spectrum that changes slowly over time.
Hanning	Compared with the rectangle window, it provides better amplitude resolution but poorer frequency resolution.	Measure sine wave, period and narrow-band random noise waveforms.
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 Math wave in the operation result display window, it can be set using the following methods.

- In the Math menu, click Scale input field, then use the Multipurpose rotary knob to adjust the vertical scale.
- Tap the icon ,  on the right of the vertical scale to increase or decrease the vertical scale.
- Double-click Scale input field to open the numeric keypad to enter the specified numeric value. For details on using the numeric keypad, refer to section 5.8 Parameter Setting.

(10) Vertical Position

Set the vertical offset of Math wave in the operation result display window, it can be set using the following steps.

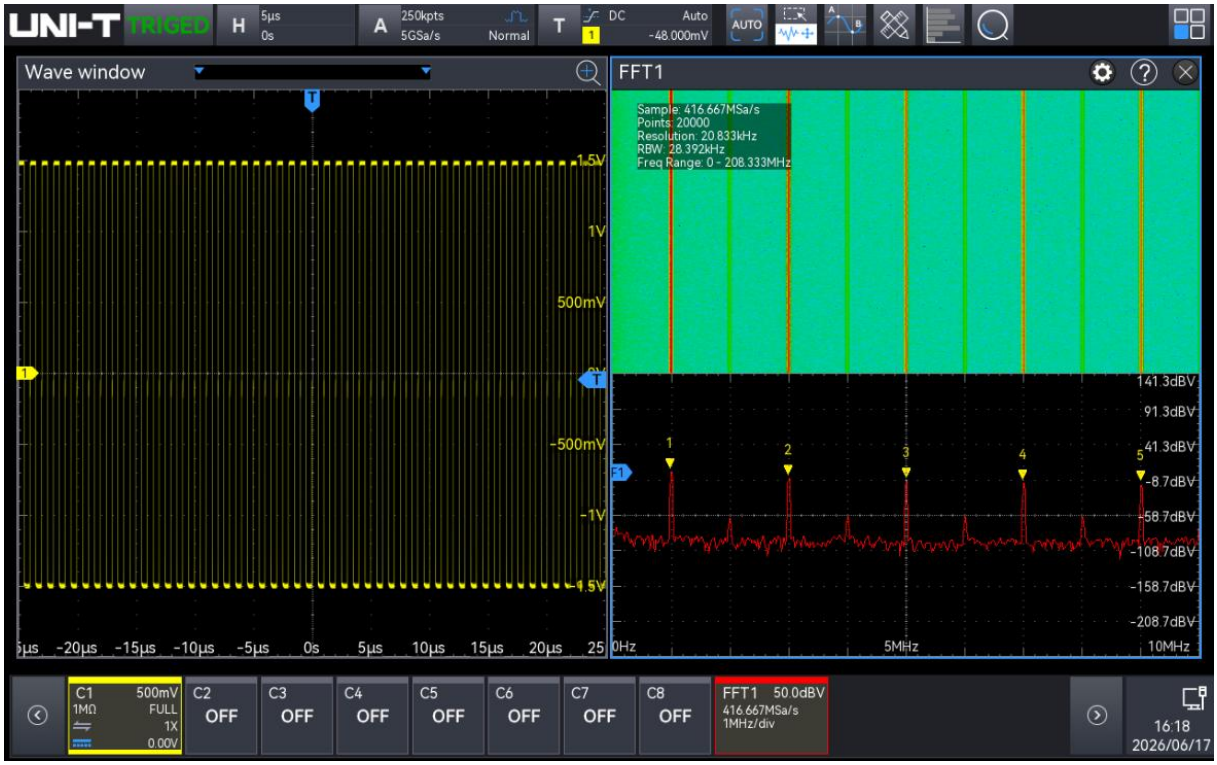
- In the Math menu, click Position input field, then use the Multipurpose rotary knob to adjust the vertical position.
- Tap the icon ,  on the right of the vertical position to increase or decrease the vertical position.
- Double-click Position input field to open the numeric keypad to enter the specified numeric

value. For details on using the numeric keypad, refer to section 5.8 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 300 (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. Click Segment selection input field, then use the Multipurpose rotary knob to adjust the segment selection. Alternatively, double-click the 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 5.8 Parameter Setting. The setting range is from 1 to 300.

(13) Trace

In FFT setting menu, click the Trace or 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: The spectrum waveform graph shows all the sampled values in real time, the

spectrum waveform displays in red.

- Average: The spectrum waveform graph shows the average value of several points taken during the sampling interval; the spectrum waveform displays in blue.
  - Average count: Set the number of average calculation, double-click the Average Count input field to open the numeric keypad to set the average count. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Average Count, then use the Multipurpose rotary knob to adjust the average time. 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 display in grey.

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)..

23. Timing Analysis

- I²C Timing Analysis
- SPI Timing Analysis
- CAN Timing Analysis
- I²S Timing Analysis
- Reset Timing Analysis
- Switch Timing Analysis
- Uart Timing Analysis
- LIN Timing Analysis

Timing analysis is a technique used to verify whether a circuit or system operates as expected under a specific clock signal. It involves analyzing factors such as signal propagation delay, clock frequency, and timing-critical paths to ensure the circuit or system meets timing requirements and functions stably and reliably under various operating conditions. Timing analysis can be performed during both the design phase and the bus validation phase.

During the design phase, timing analysis mainly focuses on identifying critical paths in sequential circuits. Static timing analysis tools, such as Prime Time and Design Compiler, are commonly used to detect these paths and perform optimizations to ensure the timing constraints are met.

During the bus phase, timing analysis involves using tools like logic analyzers, oscilloscopes, and protocol analyzers to capture bus signal waveforms and analyze timing characteristics. The timing analysis function of an oscilloscope typically measures key signal timing parameters based on decoded data, including input voltage, clock frequency, rise and fall times, high and low level durations, data setup time, and hold time.

Access the Timing Analysis menu using the following methods.

- If the timing analysis is added to the toolbar, click the timing analysis icon in the top-right corner of the toolbar to enter the timing analysis menu.
- Press the Analyze button on the front panel, then select the timing analysis to enter the menu.
- Click the Home icon in the top-right of the screen, then select the timing analysis icon to enter the menu.

23.1. General Setting

(1) Analysis

Click Analysis to turn the timing analysis on or off. When timing analysis is enabled and decoding is not already active, the decoding type and corresponding decoding bus for BUS1 will be automatically configured and enabled.

(2) Pause

When analysis is active, statistical analysis starts automatically. Press the Pause button to temporarily stop the analysis. Click Continue to resume.

(3) Reset

Click Reset to clear all current statistical test results. This action does not affect the ongoing analysis process.

(4) Protocol Type

Click Protocol Type drop-down list and select I^2C, SPI, CAN, I^2S, Reset, Switch, Uart, or LIN. Each protocol requires specific configuration of its related parameters.

(5) Stop Condition

For long-term stability testing of some devices, the statistical function can be used to monitor many bus frames (e.g., 10,000 or more) to evaluate reliability over time.

The timing analysis will stop automatically once the selected stop condition is met. Click Stop Condition drop-down list and set the condition for stopping the analysis.

- None: The analysis runs continuously and must be stopped manually.
- Number of failures: The analysis stops once the number of failures reaches the specified limit.
- Total number: The analysis stops when the total number of actual runs reaches the set total number.

(6) Number of Failures

When the stop condition is set to number of failures, the number of failures can be customized. Double-click Failure Number input field to open the numeric keypad to set the number of failures. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the Failure Number, then use the Multipurpose to change the number of failures. The range can be set from 1 to 100,000,00.

(7) Total Number

When the stop condition is set to total number, the total count can be customized. Double-click Total Number input field to open the numeric keypad to set the total number. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. Alternatively, select

the Total Number, then use the Multipurpose to change the total number. The range can be set from 1 to 100,000,00.

(8) Decoding Bus

Click Decode Bus pull-down list and select the decode bus to copy. Available options include Bus1, Bus2, Bus3, and Bus4.

(9) Decoding Menu

Click Decoding to directly jump to the decoding menu.

(10) Copy Decoding Setting

After the decoding type and parameters are configured, select the decoding bus to be copied on the Timing Analysis page, and click Copy Decode to copy the parameter settings of the selected decode bus to the Timing Analysis menu.

(11) Result Table

Click the switch on the right side of the Result Table to turn the analysis result table ON or OFF.

(12) Save

Click Save to enter the export settings menu. Users can export the analysis data list in .csv or .pdf format to internal memory or to an external USB flash drive (only when a USB flash drive is detected).

(13) Export Parameter

Click Export Parameter to enter the export parameter settings menu. Enter the desired file name and save path, then export the current parameter settings in .csv format to internal memory or a USB flash drive (only when a USB flash drive is detected). This enables convenient re-import of the settings later.

(14) Import Parameter

Click Import Parameter to enter the import parameter settings menu. Double-click File Path field to select the appropriate parameter file (.csv) from internal memory or a USB flash drive (only when a USB flash drive is detected) to import and apply the settings.

(15) Default

Click Default to restore all parameter configuration values to their default settings.

23.2. I^2C Timing Analysis

I^2C timing analysis relies on I^2C decoding. Therefore, I^2C decoding must be configured before performing I^2C timing analysis. For detailed instructions, refer to the I^2C Decoding section.

23.2.1. Analysis Setting

(1) SCL Source

Click SCL Source drop-down list to select C1 - C8 as the I^2C clock source.

(2) SDA Source

Click SDA drop-down list Source to select C1 - C8 as the I^2C data source.

(3) Transmission Rate

Click Transmission Rate drop-down list to set the data rate of the I^2C signal to standard, fast, or high speed. Changing the rate will adjust the corresponding parameter standards accordingly.

(4) Filtering Conditions

Click Filtering Conditions drop-down list to configure the filtering condition for I^2C data. This determines whether to analyze transmission frames based on fixed read/write addresses.

- None: No filtering; all data will be tested and analyzed.
- Address: Only data associated with the specified address will be analyzed.
- Address Read: Only read operations for the specified address will be analyzed.
- Address Write: Only write operations for the specified address will be analyzed.

(5) Address

Set the filtering address, double-click Address input field to open the numeric keypad to set the address. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select Address input field, use the arrow keys ←, → to position the cursor, then use the Multipurpose to adjust the address. Adjustable range: 00 - 7F, 000 - 3FF.

23.2.2. Parameter Setting

(1) Bus Level Setting

- Bus level: Set the bus level for timing analysis, which refers to the input voltage (V_{cc}) of the signal. Typically, the input voltage for I^2C is 3.30 V. If the input voltage is 1.8 V or another value, it can be adjusted using the Multipurpose rotary knob. Adjusting the bus level (V_{cc}) will affect the values of V_{IH} and V_{IL}. The relationship between V_{cc}, V_{IH}, and V_{IL} is shown in the table below.
- Input high-level/Low-level voltage (V_{IH}, V_{IL}): These input values are determined by V_{cc} and follow the relationship specified in the table. They can also be manually adjusted by using the Multipurpose rotary knob. However, modifying V_{IH} or V_{IL} will not affect the value of V_{cc}.

Symbol	Parameter	Minimum	Maximum	Unit
V_{IH}	Input high-level voltage	0.7×Vcc	Vcc+0.5	V
V_{IL}	Input low-level	-0.5	0.3×Vcc	V

	voltage			

- Each of the remaining parameters has its own specific standard. Refer to the table below for standard parameter settings. If the user's standard differs from the one shown in the table, double-click the parameter input field to open the numeric keypad to enter a specific standard. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. Alternatively, select the parameter, then use the Multipurpose rotary knob to adjust the standard value. Click Default to restore the default settings.

(2) I^2C Timing Analysis Test Item

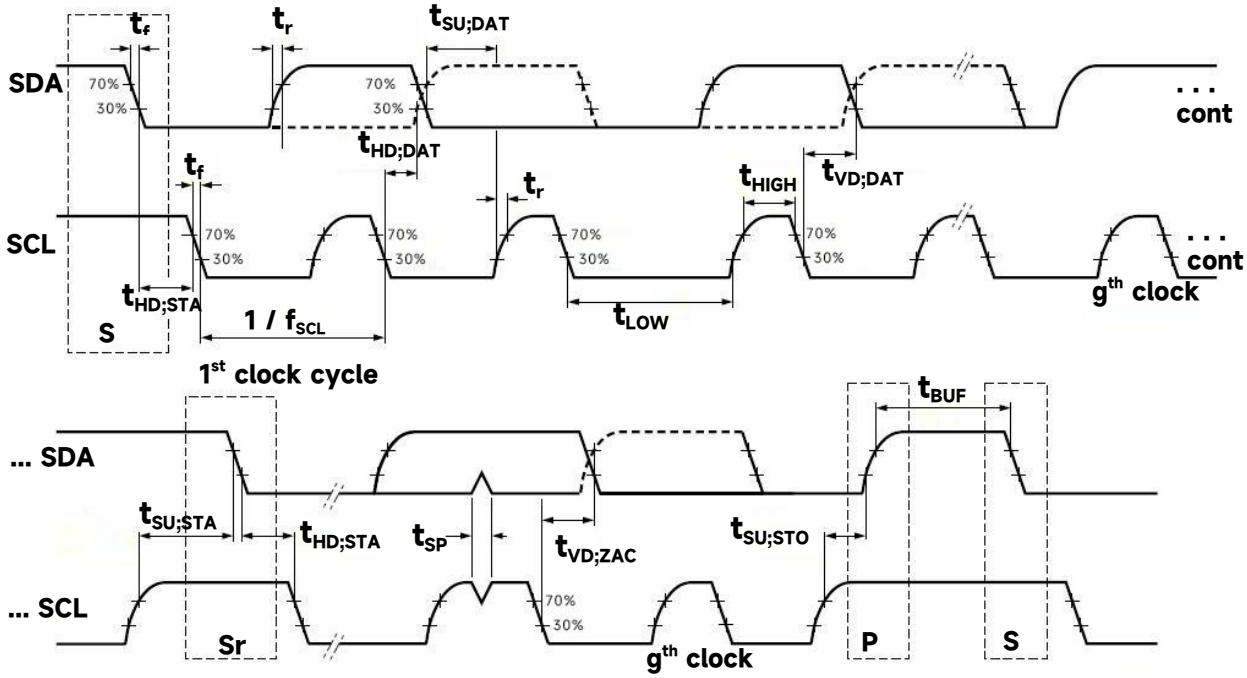
Symbol	Parameter	Standard Mode		Fast Mode		High Speed		Unit
		Minimum	Maximum	Minimum	Maximum	Minimum	Maximum	
Level	Bus level							
VIH	Input high-level voltage	0.7×Vcc	Vcc+0.5	0.7×Vcc	Vcc+0.5	0.7×Vcc	Vcc+0.5	V
VIL	Input low-level voltage	-0.5	0.3×Vcc	-0.5	0.3×Vcc	-0.5	0.3×Vcc	V
F_{SCL}	Clock frequency		100		400		3400	kHz
tHIGH	Clock high-level time	4	-	0.6		0.06	-	μs
tLOW	Clock low-level time	4.7	-	1.3		0.16	-	μs
tR	Clock/data rise time	0	1000	20	300	10	40	ns
tF	Clock/data fall time	0	300	20	300	10	40	ns
tSU.STA	Start signal setup time	4.7	-	0.6		0.16	-	μs
tHD.STA	Start signal hold time	4	-	0.6		0.16	-	μs
tSU.DAT	Data signal setup time	250	-	100		10		ns
tHD.DAT	Data signal hold time	0	3.45	0	0.9	0	0.07	μs
tSU.STO	End signal	4		0.6		0.16		μs

	setup time							
tBUF	Bus idle time	4.7		1.3		0.1		μs
Freq Deviation	Frequency offset	This is an optional item for signal timing analysis and testing. It is not a standard definition, and users can customize the allowable test range based on the actual situation.						kHz
ThresholdH	High threshold	This is an optional item for signal timing analysis and testing. It is not a standard definition, and users can customize the allowable test range based on the actual situation.						V
ThresholdL	Low threshold							
Overshoot	Overshoot							
Monot	Monotonicity							

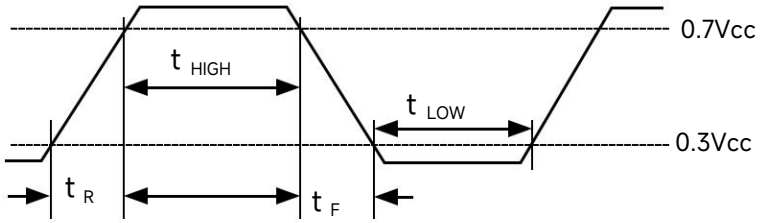
Note: All parameter values are referenced to the VIHmin (0.7Vcc) and the VILmax (0.3Vcc). If these parameters differ from those specified in the user manual, they can be manually adjusted.

(3) Signal Position

The specific signal positions corresponding to each I²C test item are shown below.



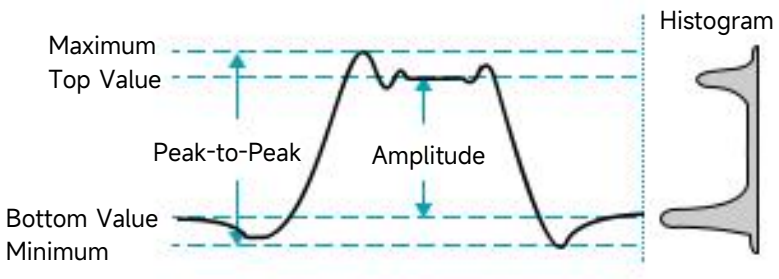
All time-related test items of I²C are based on VIHmin (0.7Vcc) and VILmax (0.3Vcc) levels, illustrated in the figure below.



Test Item: Reference Level

The calculation principles for voltage-related test items in I²C, VIH (high threshold) and VIL (low threshold) are illustrated in the figure below.

- VIH: The voltage measured from the flat top of the waveform to ground, representing the top value.
- VIL: The voltage measured from the flat bottom of the waveform to ground, representing the bottom value.



Calculation of Top and Bottom Value

(4) I²C Test Item Table

Signal Line	Test Item	Description
SCL	VIL.SCL	Low-level voltage of the SCL signal line. See Note 1.
	VIH.SCL	High-level voltage of the SCL signal line. See Note 1.
	Glitch.SCL	Glitch detection item for the SCL signal line. This parameter will be displayed only when glitches are detected; otherwise, it remains hidden. See Note 2.
	Over+.SCL	Positive overshoot test for the SCL signal line. If the measured overshoot voltage exceeds the preset threshold, the result will be marked as Fail. See Note 1.
	Over-.SCL	Negative overshoot test for the SCL signal line. If the measured undershoot voltage exceeds the preset threshold, the result will be marked as Fail. See Note 1.
	Monot.SCL	Monotonicity test for the SCL signal line. If the voltage exceeds the allowable limit, the result will be marked as Fail. See Note 2.

	Freq.SCL(First)	Frequency measured during the first cycle of the SCL clock signal. Although standard specifications define maximum clock frequencies (Standard/Fast/High-speed: 100 kHz / 400 kHz / 3.4 MHz), practical applications may allow extended tolerances. This allowable error can be configured via the "Frequency Deviation" parameter. For example, if a deviation of 1 kHz is set under Standard mode, a frequency less than 101 kHz will result in Pass. See Note 2.
	Freq.SCL(Average)	Weighted average frequency of the SCL clock signal. This value is computed by averaging most samples with consistent periods, excluding outliers. All other conditions are the same as Freq.SCL (First). See Note 2.
SCL	tR.SCL(MAX)	Maximum rise time (tR) of the SCL clock signal. The system automatically evaluates all rise-time samples of the clock signal displayed on the screen (based on the main time base) and selects the maximum value (i.e., the worst-case timing) as the result. See Note 2.
	tF.SCL(MAX)	Maximum fall time (tF) of the SCL clock signal. The measurement method is the same as for tR.SCL (MAX). See Note 2.
	tLOW(MIN)	Minimum low/high-level period of the SCL clock signal. The system automatically analyzes all low-level or high-level period samples of the SCL signal within the current screen (main time base) and selects the minimum value (i.e., the worst-case timing) as the result. See Note 2.
	tHIGH(MIN)	
SDA	Glitch.SDA	Glitch test for the SDA signal line. See Note 2.
	Over+.SDA	Positive/negative overshoot test for the SDA signal

	Over-.SDA	line. If the measured overshoot voltage exceeds the preset threshold, the software flags the result as Fail. See Note 1.
	tR.SDA(MAX)	Maximum rise time and fall time of the SDA data signal. During test, the system automatically analyzes all rise/fall time samples of the SDA signal within the current screen (main time base) and selects the maximum value (i.e., the worst-case timing) as the result. See Note 3.
	tF.SDA(MAX)	
	tSU.STA(MIN)	Setup time (tSU) of the start signal (tested only under restart conditions). The system automatically counts all setup-time samples for the start condition visible on the screen (main time base) and selects the minimum value (the waveform with the worst timing) as the measurement result. See Note 2.
	tHD.STA(MIN)	Hold time (tHD) of the start signal. The system automatically counts all hold-time samples for the start condition visible on the screen (main time base) and selects the minimum value (the waveform with the worst timing) as the measurement result. See Note 2.
	tSU.DAT(MIN)	Setup time of the data signal. The system automatically counts all data setup-time samples on the screen (main time base) and selects the minimum value (the waveform with the worst timing) as the measurement result. See Note 3.
	tHD.DAT(MIN)(MIN, MAX)	Hold time of the data signal. The system automatically counts all data hold-time samples on the screen (main time base) and selects both the minimum and maximum values as the measurement results. (There are upper-limit restrictions under certain special conditions.) If the filtering function is enabled, only the samples

		that meet the filtering conditions will be counted. See Note 2.
	tSU.STO(MIN)	Setup time of the stop signal. The system automatically counts all setup-time samples for the stop condition on the screen (main time base) and selects the minimum value (the waveform with the worst timing) as the measurement result. The condition for tSU.STO is to continuously record the latest rise time of the SCL signal until the stop condition occurs. See Note 3.
	tBUF(MIN)	Bus idle time between the previous stop and the current start condition. The system automatically counts all bus idle-time samples on the screen (main time base) and selects the minimum value (the waveform with the worst timing) as the measurement result. If there is only one frame of data on the screen, the test condition for tBUF cannot be met. In this case, the idle time before the Start signal will be output as the test result. If this value meets the requirement, it will be displayed as Pass. It is recommended to have more than two frames of data on the screen for more accurate results. See Note 1.

Note 1: Even if the filtering function is enabled, the measurement samples will still be processed based on the data from the entire screen (main time base) and are not affected by the filtering range.

Note2: If the filtering function is enabled, the samples will be statistically analyzed with ADDR as the filtering condition, without distinguishing between R/W (whether reading or writing, only the samples corresponding to the address are considered).

Note 3: If the filtering function is enabled, the samples will be statistically analyzed with ADDR (including both reading and writing samples), ADDR+R (only considering the reading samples of the address), or ADDR+W (only considering the writing samples of the address) as the filtering conditions, with R/W distinguished.

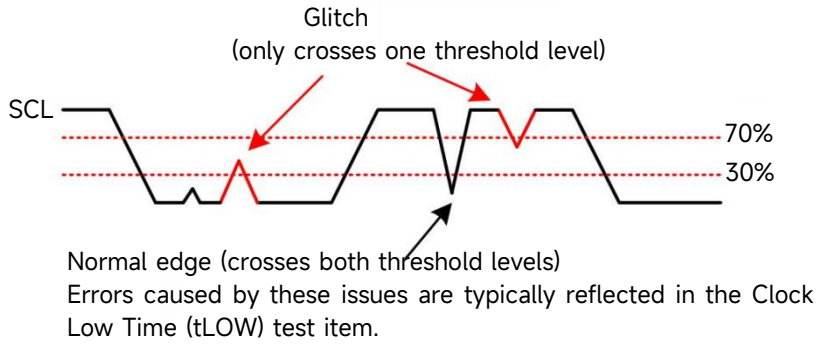
(5) Signal Quality Items

Signal quality testing includes glitch, overshoot, and monotonicity detection.

- SCL Signal Glitch Detection

  When the SCL clock signal changes state, if the signal only crosses one threshold level (either high or low), it is identified as a glitch, as illustrated in the figure below.

  Note: If a signal crosses both threshold levels in a single transition, it will be treated as a normal edge by the master controller. However, this may cause errors in other test items.

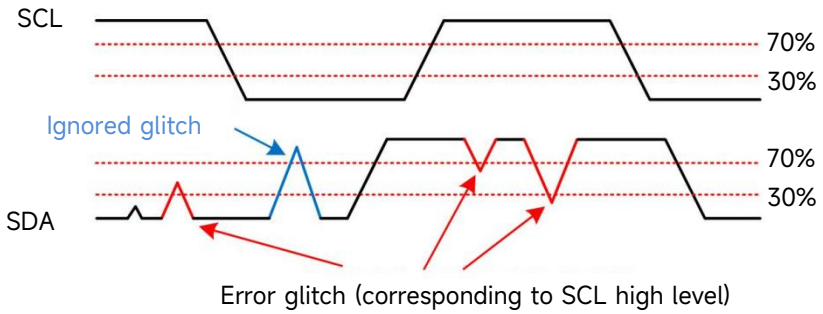


SCL waveform only crosses one threshold level

- SDA Signal Glitch Detection

  For the SDA data signal line, glitch detection is only valid when SCL is at a high level. Any glitch that appears while the SCL is high, regardless of whether it crosses one or two threshold levels is determined to be a glitch, as shown in the figure below.

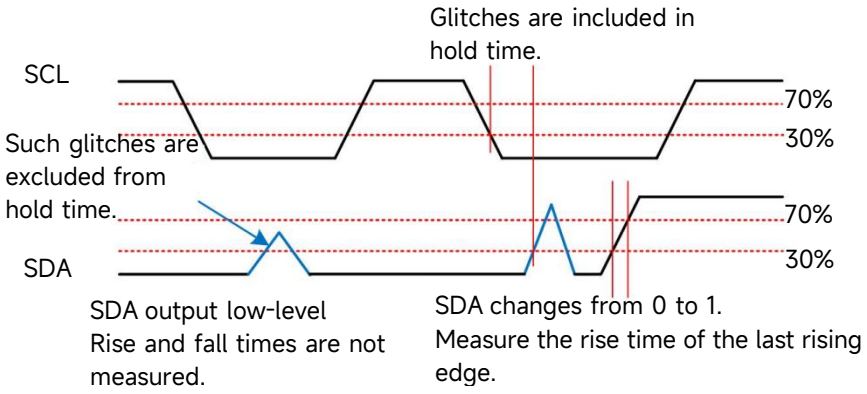
  Note: During the period when SCL is low, glitches on the SDA line are ignored, since they have no effect on bus operation.



Referencing the SCL level

- SDA Signal Glitch

  Glitches excluded from rise and fall time statistics: When measuring the rise or fall time of the SDA signal, the software automatically filters out and ignores any glitches to prevent measurement errors. However, glitches can still affect the accuracy of setup and hold time measurements. As illustrated in the figure below, the system uses the last transition (from 0 to 1 or from 1 to 0) of the SDA line before the SCL signal goes high as the valid rise or fall time.



Calculation of rising/falling time

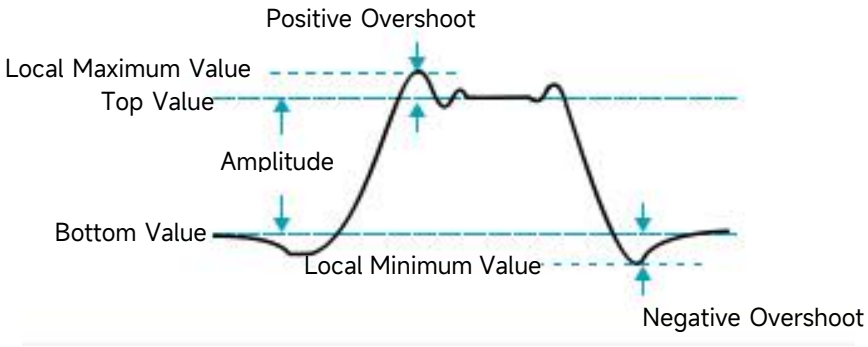
- Overshoot Determination Principle

  Classification of Overshoots: Overshoots are categorized into positive and negative types. Their respective positions on the waveform are shown in the figure below. The maximum overshoot value is recorded and compared with a preset threshold (default is 0.7 V in the oscilloscope, but it can be manually adjusted). If the measured overshoot exceeds the threshold, the result is marked as Fail; otherwise, it is Pass.

  Overshoot Calculation Formulas:

  Positive Overshoot = Local Maximum Value − Top Value

  Negative Overshoot = Bottom Value − Local Minimum Value



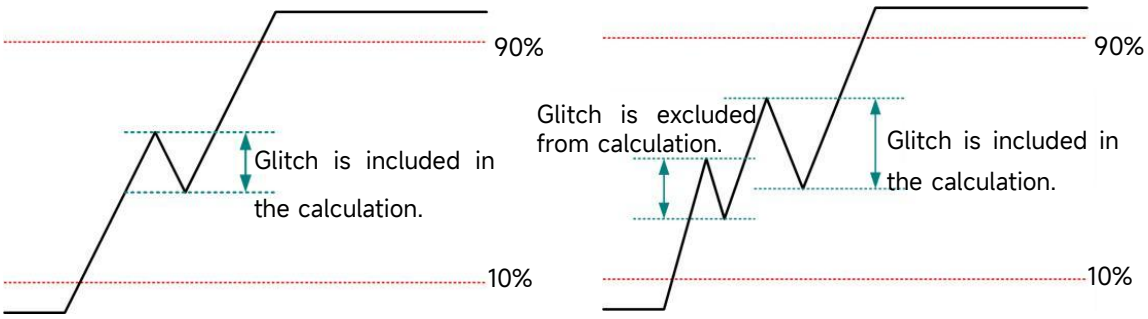
Overshoot

- Monotonicity Determination Principle

  Monotonicity, also referred to as glitch, describes the phenomenon in which a signal exhibits non- monotonic behavior during its rising or falling edge transitions, typically due to interference

  The evaluation of glitch voltage amplitude is as follows:

  If the measured glitch voltage amplitude exceeds the voltage reference threshold specified in the “Monotonicity” parameter settings, the test result is marked as Fail.



Glitch on rising edge of waveform

Note: The 90% and 10% values in the figure above correspond to the user-defined high and low thresholds, respectively.

(6) I^2C Timing Analysis Configuration – Test Item

Symbol	Parameter	Description
Level	Bus level	The unit is volts (V). If the bus voltage is Vcc, then: Vcc < Vih(max) <= 1010 0.5×Vcc <= Vih(min) < Vcc 0.01 <= Vil(max) < 0.5×Vcc - 10 <= Vil(min) <= - 0.01
VIH(max)	Maximum input high - level voltage	
VIH(min)	Minimum input high - level voltage	
VIL(max)	Maximum input low - level voltage	
VIL(min)	Minimum input low - level voltage	
Freq(max)	Maximum frequency limit	The unit is kHz. The value equals the default rate plus the frequency deviation.
tHIGH(min)	Minimum time of clock high - level	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tLOW(min)	Minimum time of clock low - level	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tR(max)	Maximum rise time of clock/data	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tR(min)	Minimum rise time of clock/data	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000. The setting must satisfy the condition: tR(min) < tR(max).
tF(max)	Maximum fall time of clock/data	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tF(min)	Minimum fall time of clock/data	The unit is nanoseconds (ns), with a range from 0 to 999,000,000. The setting must satisfy the condition: tF(min) < tF(max).
tSU.STA(min)	Minimum setup time of start signal	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.

tHD.STA(min)	Minimum hold time of start signal	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tSU.DAT(min)	Minimum setup time of data signal	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tHD.DAT(min)	Minimum hold time of data signal	The unit is nanoseconds (ns), with a range from 1 to 999,000,000. The setting must satisfy the condition: tHD.DAT(min) < tHD.DAT(max).
tHD.DAT(max)	Maximum hold time of data signal	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tSU.STO(min)	Minimum setup time of end signal	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tBUF(min)	Minimum time of bus idle	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
Frequency Offset	Clock frequency offset	The unit is Hertz (Hz), with a range from -10,000 to 10,000.
ThresholdH	High threshold	The unit is percent (%), with a range from 20 to 90.
ThresholdL	Low threshold	The unit is percent (%), with a range from 10 to 80. The setting must satisfy the condition: Low Thresh<=High Thresh-10
Overshoot	Default overshoot value	The unit is volts (V), with a range from 0.01 to 1,000.
Monot	Default monotonicity value	

23.3. SPI Timing Analysis

SPI timing analysis relies on SPI decoding. Therefore, SPI decoding must be configured before using SPI timing analysis. For detailed instructions, refer to the SPI Decoding section.

23.3.1. Analysis Setting

(1) Clock Source

Click Clock Source drop-down list to select C1 - C8 as the SPI clock source.

(2) Chip Select Source

Click CS Source drop-down list to select None or C1 - C8 as the SPI CS source. If set to None, parameters related to the chip select signal will not be tested.

(3) Input Source

Click Input Source drop-down list to select None or C1 - C8 as the SPI input source. If set to None, SDI-related test items will be skipped during this test.

(4) Output Signal Source

Click Output Source drop-down list to select None or C1 - C8 as the SPI output source. This output can be routed to other devices. If set to None, SDO-related test items will be skipped during this test.

(5) Sampling Edge

Set the sampling edge of the SPI clock.

Rising Edge: Corresponds to positive clock polarity in decoding.

Falling Edge: Corresponds to negative clock polarity in decoding.

23.3.2. Parameter Setting

(1) Bus Level Setting

- Bus level: Set the bus level for timing analysis, which refers to the input voltage (V_{cc}) of the signal. Typically, the input voltage for SPI is 3.30 V. If the input voltage is 1.8 V or another value, it can be adjusted using the Multipurpose rotary knob. Adjusting the bus level (V_{cc}) will affect the values of V_{IH} and V_{IL}. The relationship between V_{cc}, V_{IH}, and V_{IL} is shown in the table below.
- Input high-level/Low-level voltage (V_{IH}, V_{IL}): These input values are determined by V_{cc} and follow the relationship specified in the table. They can also be manually adjusted by using the Multipurpose rotary knob. However, modifying V_{IH} or V_{IL} will not affect the value of V_{cc}.

Symbol	Parameter	Minimum	Maximum	Unit
V_{IH}	Input high-level voltage	0.7×Vcc	Vcc+0.5	V
V_{IL}	Input low-level voltage	-0.5	0.3×Vcc	V

- Each of the remaining parameters has its own specific standard. Refer to the table below for standard parameter settings. If the users' standard differs from the one shown in the table, double-click the parameter input field to open the numeric keypad to enter a specific standard. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the parameter, then use the Multipurpose rotary knob to adjust the standard value. Click Default to restore the default settings.

(2) SPI Timing Analysis Test Item

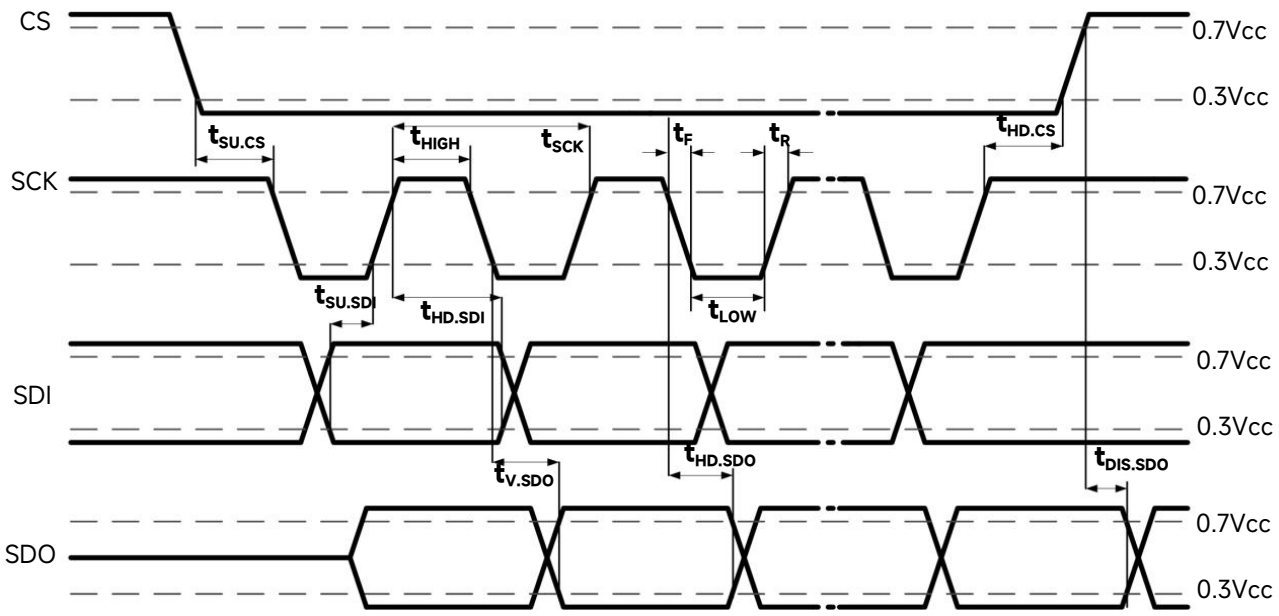
Symbol	Parameter	Minimum	Maximum	Unit
Level	Bus level			
VIH	Input high-level voltage	0.7×Vcc	Vcc+0.5	V

VIL	Input low-level voltage	-0.5	0.3×Vcc	V
Freq (min)	Minimum frequency filter	10		kHz
Freq (max)	Maximum frequency filter	1		MHz
tR	Clock rise time		2.0	μs
tF	Clock fall time		2.0	μs
t_{HIGH}	Clock high-level time	5.0	-	ns
t_{LOW}	Clock low-level time	5.0	-	ns
$t_{SU.CS}$	CS signal setup time	5.0		ns
$t_{SU.SDI}$	Data input signal setup time	2.0	-	ns
$t_{HD.CS}$	CS signal hold time	2.0		ns
$t_{HD.SDI}$	Data input signal hold time	0		ns
$t_{HD.SDO}$	Data output signal hold time	0		ns
$t_{V.SDO}$	Maximum data output signal valid time	1.0		μs
$t_{DIS.SDO}$	Minimum time from CS inactive to data off	This item is optional and not defined in standard specifications. Users can customize the acceptable test range based on actual application requirements.		
$Over_+$	Positive overshoot	This item is part of optional signal quality testing and is not defined in standard specifications. Users may define the acceptable test range according to their specific needs.		
$Over_-$	Negative overshoot			
Glitch	Glitch			

Note: All parameter values are referenced to the VIHmin (0.7Vcc) and the VILmax (0.3Vcc). If these parameters differ from those specified in the user manual, they can be manually adjusted.

(3) Signal Position

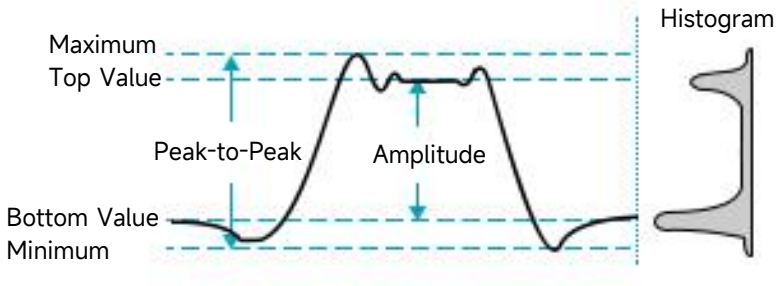
The specific signal positions for SPI timing-related test items are shown below.



All time-related test items of SPI are based on VIHmin (0.7Vcc) and VILmax (0.3Vcc) levels. The 0.7Vcc and 0.3Vcc levels serve as the high and low threshold values in the parameter settings, and they are configurable.

The calculation principles for voltage-related test items in SPI, VIH (high threshold) and VIL (low threshold) are illustrated in the figure below.

- VIH: The voltage measured from the flat top of the waveform to ground, representing the top value.
- VIL: The voltage measured from the flat bottom of the waveform to ground, representing the bottom value.



Calculation of Top and Bottom Value

(4) SPI Timing Analysis Test Item

Signal	Test Item	Description
SCK	VIL.SCK	Low-level voltage of the SCK signal line, as shown in the bottom value of the figure.
	VIH.SCK	High-level voltage of the SCK signal line, as shown in the top value of the figure.
	Glitch.SCK	Glitch test item for the SCK signal line.
	Over+.SCK	Positive overshoot test item for the SCK signal line. If the overshoot voltage exceeds the preset value, the software will judge this item as

		Fail.
	Over-.SCK	Negative overshoot test item for the SCK signal line. If the overshoot voltage exceeds the preset value, the software will judge this item as Fail.
	Freq.SCK	Frequency measurement based on a threshold of 0.7Vcc (configurable). The displayed measurement value is for the first clock cycle, where Freq.SCK = 1/tSCK.
	tR.SCL(MAX)	The rise time (tR) and fall time (tF) of the clock. During measurement, the system will automatically count all the rise-time and fall-time samples on the screen (main time base) and select the maximum value (the worst timing waveform) as the measurement result.
	tF.SCL(MAX)	
	tLOW(MIN)	The low-level period (tLOW) and high-level period (tHIGH) of the SCK clock signal. During measurement, the system will automatically count all the low-level or high-level period samples on the screen (main time base) and select the minimum value (the worst timing waveform) as the measurement result.
	tHIGH(MIN)	
CS	VIL.CS	Low-level voltage of the chip select (CS) signal line, as shown in the bottom value of the figure.
	VIH.CS	High-level voltage of the chip select (CS) signal line, as shown in the top value of the figure.
	Glitch.CS	Glitch test item for the CS signal line.
	tSU.CS(MIN)	Setup time for the chip select signal. The system will automatically count all the setup-time samples for the CS signal on the screen (main time base) and select the minimum value as the measurement result.
	tHD.CS(MIN)	Hold time for the chip select signal. The system will automatically count all the hold-time samples for the CS signal on the screen (main time base) and select the minimum value as the measurement result.
SDI	VIL.SDI	Low-level voltage of the SDI signal line, as shown in the bottom value of the figure.
	VIH.SDI	High-level voltage of the SDI signal line, as shown in the top value of the figure.
	Glitch.SDI	Glitch test item for the SDI signal line.
	tSU.SDI(MIN)	Setup time for the data input signal. The system will automatically count all the setup-time samples for the data signal on the screen (main time base) and select the minimum value as the measurement result.
	tHD.SDI(MIN)	Hold time for the data input signal. The system will automatically count all the hold-time samples for the data signal on the screen

		(main time base) and select the minimum value as the measurement result.
SDO	VIL.SDO	Low-level voltage of the SDO signal line, as shown in the bottom value of the figure.
	VIH.SDO	High-level voltage of the SDO signal line, as shown in the top value of the figure.
	Glitch.SDO	Glitch test item for the SDO signal line.
	tV.SDO(MIN)	Data output valid time. The system will automatically count all the output valid time samples for the data signal on the screen (main time base) and select the minimum value as the measurement result.
	tHD.SDO(MIN)	Hold time for the data output signal. The system will automatically count all the hold-time samples for the data signal on the screen (main time base) and select the minimum value as the measurement result.
	tDIS.SDO(MIN)	Minimum time from CS invalid to data being turned off. The system will automatically count all the CS invalid to data turn-off time samples on the screen (main time base) and select the minimum value as the measurement result.

(5) Signal Quality Items

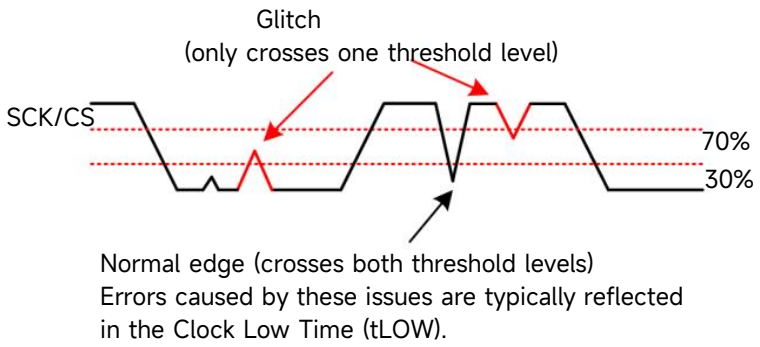
Signal quality testing includes glitch and overshoot detection.

- Glitch Determination Principle

a. SCK/CS Signal Glitch Detection

When the SCK/CS clock signal changes state, if the signal only crosses one threshold level, it is identified as a glitch, as illustrated in the figure below.

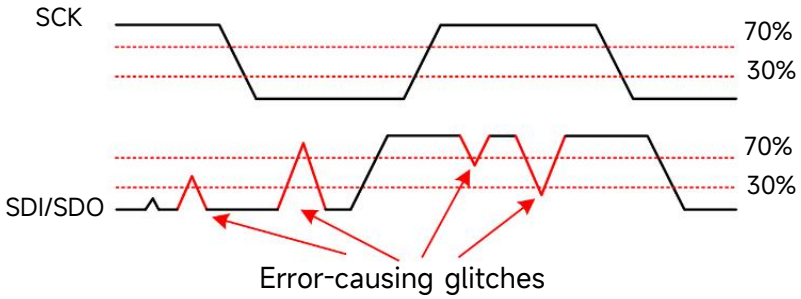
Note: If a signal crosses both threshold levels in a single transition, it will be treated as a normal edge by the master controller. However, this may cause errors in other test items.



SCK/CS waveform only crosses one threshold level

b. SDI/SDO Signal Glitch Detection

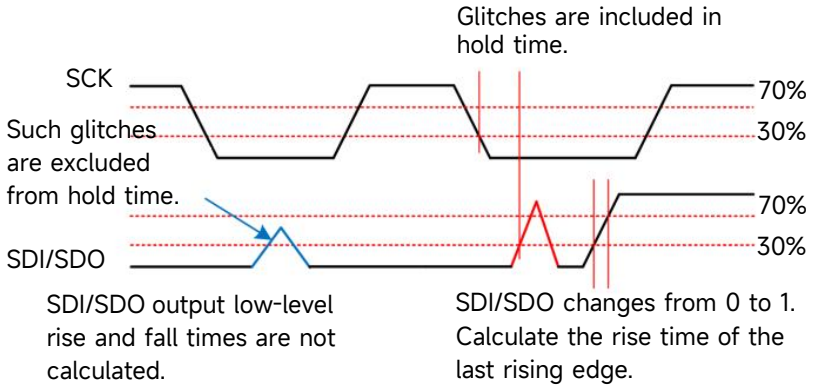
Glitches that appear on the SDI or SDO data signal lines, whether during the high or low level of the SCK clock always affect the bus and are therefore treated as error-causing glitches. As shown in the figure below, these glitches will be reported as errors.



SDI/SDO signal glitch detection

c. SDI/SDO Signal Glitch

Glitches excluded from rise and fall time statistics: When measuring rise or fall times, the software will automatically filter out and ignore glitches to prevent measurement errors. However, glitches can affect the measurement of setup and hold times. As shown in the figure below, the system will use the most recent transition (from 0 to 1 or from 1 to 0) on the SDI/SDO line before the SCK clock goes high as the rise or fall time.



Calculation of rising/falling time

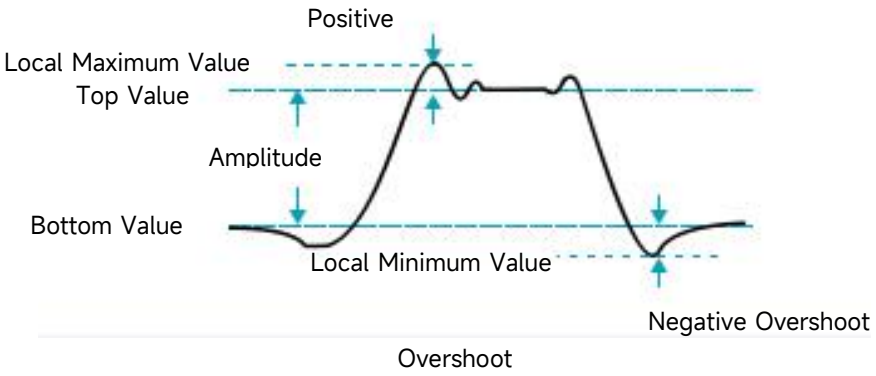
■ Overshoot Determination Principle

Classification of Overshoots: Overshoots are categorized into positive and negative types. Their respective positions on the waveform are shown in the figure below. The maximum overshoot value is recorded and compared with a preset threshold (default is 0.7 V in the oscilloscope, but it can be manually adjusted). If the measured overshoot exceeds the threshold, the result is marked as Fail; otherwise, it is Pass.

Overshoot Calculation Formulas:

Positive Overshoot = Local Maximum Value − Top Value

Negative Overshoot = Bottom Value − Local Minimum Value



(6) SPI Timing Analysis Configuration – Test Item

Symbol	Parameter	Description
Level	Bus level	The unit is volts (V). If the bus voltage is Vcc, then: Vcc<VIH(max)<=1010 0.5×Vcc<= VIH(min)<Vcc 0.01<=VIL(max)<0.5×Vcc -10<=VIL(min)<= -0.01
VIH(max)	Maximum input high - level voltage	
VIH(min)	Minimum input high - level voltage	
VIL(max)	Maximum input low - level voltage	
VIL(min)	Minimum input low - level voltage	
tHIGH(min)	Minimum time of clock high - level	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tLOW(min)	Minimum time of clock low - level	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tR(max)	Maximum rise time of clock	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tF(max)	Maximum fall time of clock	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tSU.CS(min)	Minimum setup time of CS signal	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tHD.CS(min)	Minimum hold time of CS signal	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tSU.SDI(min)	Minimum setup time of data input signal	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tHD.SDI(min)	Minimum hold time of data input signal	The unit is nanoseconds (ns), with a range from 0 to 999,000,000.
tV.SDO(min)	Maximum data output valid time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tHD.SDO(min)	Minimum data output valid	The unit is nanoseconds (ns), with a range

	time	from 1 to 1,000,000,000.
tDIS.SDO	tDIS.SDO switch	ON, OFF
tDIS.SDO(min)	Minimum time from CS inactive to data disable	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
Speed Filter	Speed filter switch	ON, OFF
Freq(min)	Minimum frequency filter	The unit is Hertz (Hz), with a range from 1 to 1,990,000,000. The setting must satisfy the condition: Freq(min) < Freq(max).
Freq(max)	Maximum frequency filter	The unit is Hertz (Hz), with a range from 2 to 2,000,000,000.
Hith Thresh	High threshold	The unit is percent (%), with a range from 20 to 90.
Low Thresh	Low threshold	The unit is percent (%), with a range from 10 to 80. The setting must satisfy the condition: Low Thresh<=High Thresh-10
Over	Default overshoot value	The unit is volts (V), with a range from 0.01 to 1000.

23.4. CAN Timing Analysis

CAN timing analysis software is a plugin designed to automatically test the electrical characteristics of the CAN bus. It is suitable for hardware testing of all CAN bus products.

CAN timing analysis relies on CAN decoding; therefore, CAN decoding must be configured before using the timing analysis function. For detailed instructions, refer to the CAN Decoding section.

23.4.1. Analysis Setting

(1) Source

Click Source drop-down list to select C1 - C8 as CAN signal source.

(2) Signal Type

Click Signal Type drop-down list to select from CAN_L, CAN_H, or CAN_DIFF. The default voltage measurement range varies depending on the selected signal type.

(3) 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 a custom bitrate.

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

(4) Filter Conditions

Click Filter Conditions drop-down list to set the CAN signal frame ID filter. Different filter conditions correspond to different configurable ranges for the frame ID.

- None: All CAN frames will be tested.
- Standard frame: Only the bit time of the specified standard frame ID will be tested.
- Extended frame: Only the bit time of the specified extended frame ID will be tested.

(5) Frame ID

- When the filter condition is set to standard frame or extended frame, the frame ID can be configured. Double-click Frame ID input field to open the numeric keypad to set the frame ID. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select Frame ID input field, use the arrow keys ←, → to position the cursor, then use the Multipurpose to adjust the frame ID. Adjustable range: 000 - 7FF, 00000000 - 1FFFFFFF.

(6) Bit Time Error

Bit time error refers to the deviation between the calculated bit time based on the signal edge and the configured bitrate.

Click Bit Time Error to select the error threshold. Available options are: 0.35%, 0.45%, and 0.5%.

23.4.2. Parameter Setting

(1) Test Item

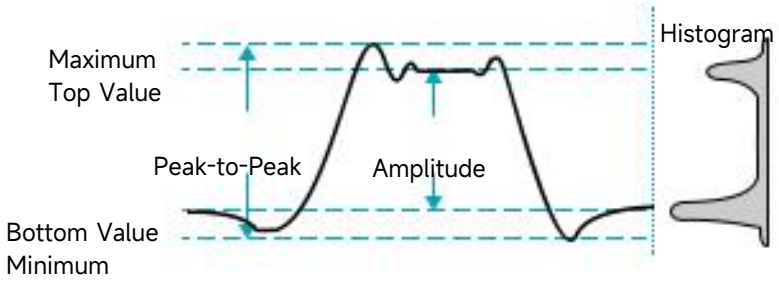
Dominant and recessive bit timing analysis test items in the CAN timing analysis software.

Measurement Item	Description
Dominant.x	Dominant bit voltage test (see Note 1)
Recessive.x	Recessive bit voltage test (see Note 1)
Bit Time (MIN,MAX)	Bit time test, outputs the minimum and maximum values observed during statistics

Note 1: “x” can be CAN_DIFF, CAN_L, or CAN_H, representing differential or single-ended line voltages, respectively.

(2) Dominant and Recessive Voltage Configuration

The dominant and recessive voltage levels on the CAN bus are determined by analyzing the voltage histogram in the specified direction. Depending on the selected test mode (CAN Differential, CAN_H single-ended, or CAN_L single-ended), the dominant and recessive test results correspond to either the top or bottom values in the waveform, as illustrated in the figure below.



Calculation of Top and Bottom Value

According to Section 4.1 of the GMW14241 standard, the acceptable voltage ranges for dominant bits and recessive bits are specified in Table 1 and Table 2, respectively.

If the measured voltage exceeds the specified range, the software will analyze the result and report it as Fail.

Table 1 Dominant Bit Voltage Range

Notation	Value		
	Minimum	Nominal	Maximum
CAN_L	0.5 V (preferred: 0.8 V)	1.5 V	2.25 V (preferred: 2.0 V)
CAN_H	2.75 V (preferred: 3.0 V)	3.5 V	4.5 V (preferred: 4.25 V)
CAN_DIFF	1.5 V	2.0 V	3.0 V

Table 2 Recessive Bit Voltage Range

Notation	Value		
	Minimum	Nominal	Maximum
CAN_L	2.0V	2.5V	3.0V
CAN_H	2.0V	2.5V	3.0V
CAN_DIFF	-120 mV	0	+12 mV

(3) Parameter Table

Parameter	Description
Dominant(max)	The upper limit of the allowable voltage value for the dominant bit (logic 0). When the measured value exceeds this value, the result will be judged as "Fail". Under normal circumstances, the default value can be used.
Dominant(min)	The lower limit of the allowable voltage value for the dominant bit (logic 0). When the measured value is less than this value, the result will be judged as "Fail". Under normal circumstances, the default value can be used.
Recessive(max)	The upper limit of the allowable voltage value for the recessive bit

	(logic 1). When the measured value exceeds this value, the result will be judged as "Fail". Under normal circumstances, the default value can be used.
Recessive(min)	The lower limit of the allowable voltage value for the recessive bit (logic 1). When the measured value is less than this value, the result will be judged as "Fail". Under normal circumstances, the default value can be used.
Threshold	This parameter is valid when the bus type is set to CAN_DIFF. When the bus type is CAN_L or CAN_H, this parameter cannot be set, and the threshold is automatically the midpoint between the dominant and recessive voltage levels (i.e., 50%).
Bit_time	This refers to the allowable error in the bit time reference value. There are three options: 0.35%, 0.45%, and 0.50%. When the CAN bus is single-wire, select 0.35%. When the CAN bus is a high-speed differential, select 0.45%. When the CAN bus is a medium-speed differential, select 0.50%. These choices are based on the specifications in sections 4.3.10, 4.1.10, and 4.2.10 of the GMW14241 standard.

23.5. I²S Timing Analysis

It is a plugin capable of automatically testing the electrical characteristics of the I²S bus. It can complete the DC and AC characteristic analysis of the bus signals in a very short time, comparing them with the nominal parameters in the device manual, and directly output the test results (Pass/Fail). It also supports report export. This plugin is suitable for hardware testing of all I²S buses, especially for hardware testing of mass-produced products. It can complete the workload that traditionally takes 30 to 60 minutes within just a few minutes.

I²S Timing Analysis depends on Audio decoding. Therefore, when using I²S timing analysis, it is necessary to first configure the audio decoding. For the detailed configuration process of audio decoding, please refer to the Audio Decoding section.

23.5.1. Analysis Setting

(1) Clock Source

Click Clock Source drop-down list to select C1 - C8 as the I²S clock source.

(2) Chip Select Signal Source

Click CS Source drop-down list to select C1 - C8 as the I²S CS source.

(3) Data Source

Click Data Source drop-down list to select C1 - C8 as the I^2S data signal source.

(4) Sender

Click Sender drop-down list select Master or Slave, indicating whether the data sender or data receiver is the command sender.

23.5.2. Parameter Setting

(1) Bus Level Setting

- Bus level: Set the bus level for timing analysis, which refers to the input voltage (V_{cc}) of the signal. Typically, the input voltage for I^2S is 3.30 V. If the input voltage is 1.8 V or another value, it can be adjusted using the Multipurpose rotary knob. Adjusting the bus level (V_{cc}) will affect the values of V_{IH} and V_{IL}. The relationship between V_{cc}, V_{IH}, and V_{IL} is shown in the table below.
- Input high-level/Low-level voltage (V_{IH}, V_{IL}): These input values are determined by V_{cc} and follow the relationship specified in the table. They can also be manually adjusted by using the Multipurpose rotary knob. However, modifying V_{IH} or V_{IL} will not affect the value of V_{cc}.

Symbol	Parameter	Minimum	Maximum	Unit
V_{IH}	Input high-level voltage	0.7×Vcc	Vcc+0.5	V
V_{IL}	Input low-level voltage	-0.5	0.3×Vcc	V

- Each of the remaining parameters has its own specific standard. Refer to the table below for standard parameter settings. If the users' standard differs from the one shown in the table, double-click the parameter input field to open the numeric keypad to enter a specific standard. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the parameter, then use the Multipurpose rotary knob to adjust the standard value. Click Default to restore the default settings.

(2) I^2S Timing Analysis Test Item

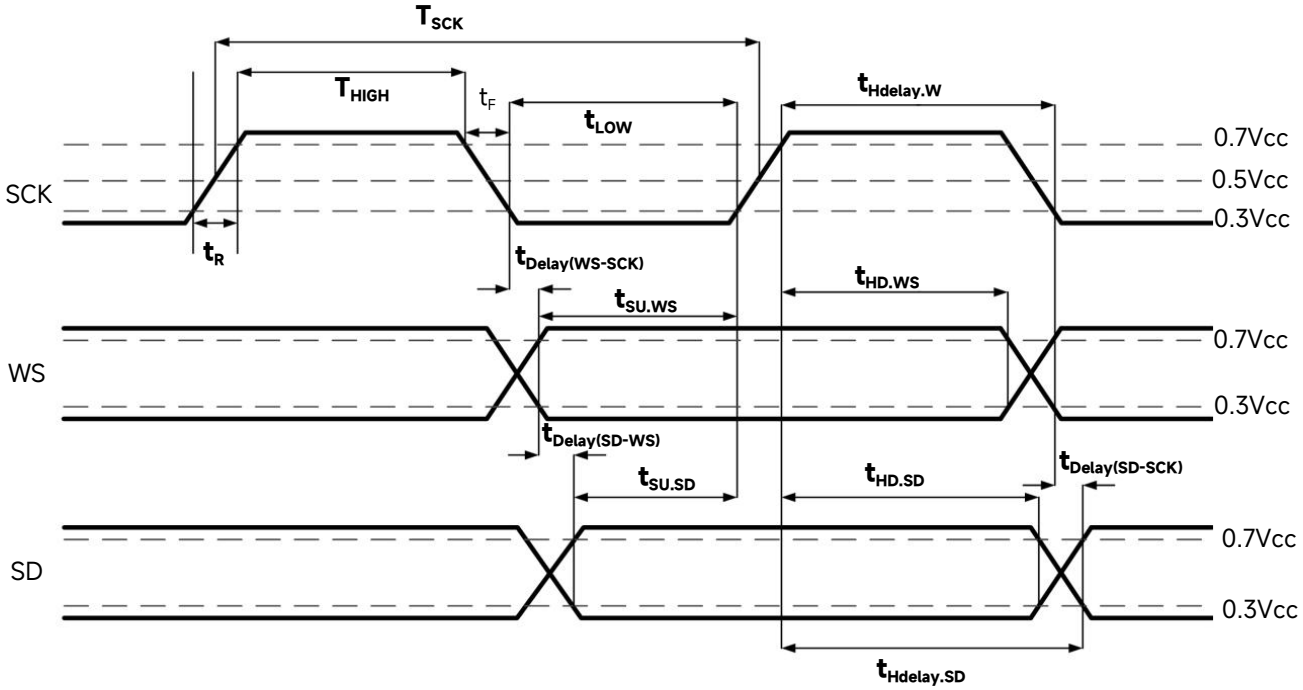
Symbol	Parameter	Minimum	Maximum	Unit
Level	Bus level			
VIH	Input high-level voltage	0.7×Vcc	Vcc+0.5	V
VIL	Input low-level voltage	-0.5	0.3×Vcc	V
tJitter	Clock jitter	-	100	μs
tR	Clock rise time	-	100	μs
tF	Clock fall time	-	100	μs
t_{HIGH}	Clock high-level time	160	-	ns

t_{LOW}	Clock low-level time	160	-	ns
t_{SU}	Signal setup time	60	-	ns
t_{HD}	Signal hold time	0	-	ns
t_{HDelay}	Hold delay time	-	500	μs
t_{Delay}	Transmission delay time	-80	80	μs
$Over_+$	Positive overshoot	This item is part of optional signal quality testing and is not defined in standard specifications. Users may define the acceptable test range according to their specific needs.		
$Over_-$	Negative overshoot			
Monot	Monotonicity			
Glitch	Glitch			

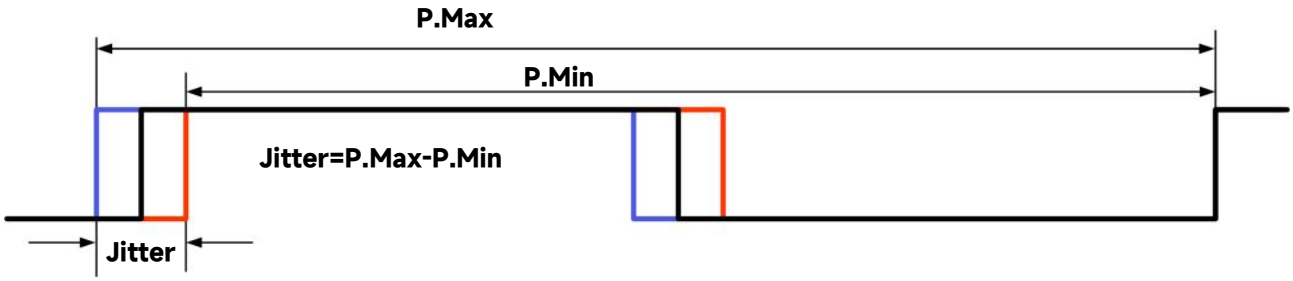
Note: All parameter values are referenced to the VIHmin (0.7Vcc) and the VILmax (0.3Vcc). If these parameters differ from those specified in the user manual, they can be manually adjusted.

(3) Signal Position

The specific signal positions for I²S timing-related test items are shown below.



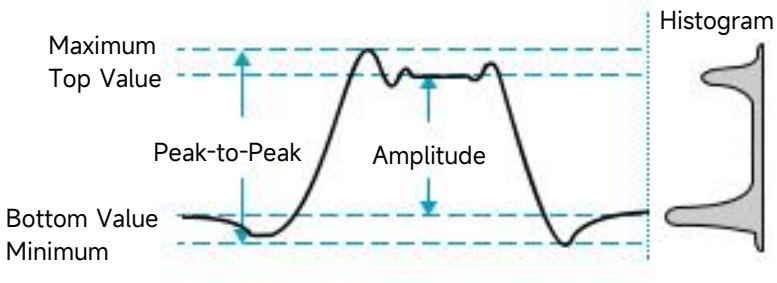
The analysis of the clock jitter test item in I²S is shown in the figure below.



Clock Jitter

The calculation principles for voltage-related test items in I²S, VIH (high threshold) and VIL (low threshold) are illustrated in the figure below.

- VIH: The voltage measured from the flat top of the waveform to ground, representing the top value.
- VIL: The voltage measured from the flat bottom of the waveform to ground, representing the bottom value.



Calculation of Top and Bottom Value

(4) I²S Timing Analysis Test Item

Signal	Test Item	Description
SCK	VIL.SCK	Low-level voltage of the SCK signal line, as shown in the bottom value of the figure.
	VIH.SCK	High-level voltage of the SCK signal line, as shown in the top value of the figure.
	Glitch.SCK	Glitch test item for the SCK signal line. This item is only displayed when a glitch is detected during analysis.
	Over+.SCK	Positive overshoot test item for the SCK signal line. If the overshoot voltage exceeds the preset value, the software will judge this item as Fail.
	Over-.SCK	Negative overshoot test item for the SCK signal line. If the overshoot voltage exceeds the preset value, the software will judge this item as Fail.
	Monot.SCK	Monotonicity (i.e., dip) test item for the SCK signal line. If the dip voltage exceeds the preset threshold, the software will mark this item as "Fail".
	Freq.SCK(Avg)	The frequency of the SCK clock signal is measured at a threshold of 0.5Vcc. The average frequency is calculated using a weighted algorithm based on many samples with similar periods (excluding outlier cycles that are too long or too short).
	Jitter.SCK	The jitter value is calculated as the difference between the maximum and minimum of the measured clock periods. It is compared against a preset threshold. If the value exceeds the threshold, the result is Fail, otherwise "Pass".

		Since this item is not strictly required, the default threshold is set relatively large, and it usually passes under normal conditions. Set it manually only if specific requirements exist.
	tR.SCL(MAX)	tR and tF represent the rise and fall times of the clock signal. The system automatically analyzes all clock rise time and fall time samples on the screen (main time base), and outputs the maximum value among the samples (the worst timing waveform) as the result.
	tF.SCL(MAX)	
	tLOW.SCK(MIN)	tLOW and tHIGH represent the low-level and high-level durations of the SCK clock signal. The system automatically analyzes all low-level or high-level duration samples on the screen (main time base), and outputs the minimum value (the worst timing waveform) as the result.
	tHIGH.SCK(MIN)	
WS	VIL.WS	The low-level (VIL) voltage of the WS signal line, as shown in the bottom value of the figure.
	VIH.WS	The high-level (VIL) voltage of the WS signal line, as shown in the top value of the figure.
	Glitch.WS	Glitch test item for the WS signal line.
	Over+.WS	Positive and negative overshoot test items for the WS signal line. If the overshoot voltage exceeds the preset value, the result is Fail.
	Over-.WS	
	Monot.WS	Monotonicity (glitch) test item for the WS signal line. If the glitch voltage exceeds the preset threshold, the result is Fail.
	Freq.WS(Avg)	The frequency of the WS clock signal is measured at a threshold of 0.5Vcc. The average frequency is calculated using a weighted algorithm based on many samples with similar periods.
	Jitter.WS	Jitter value = Maximum - Minimum. Compared with the preset threshold: if greater, the result is Fail; otherwise, it is Pass. Default values are loose since this item is not mandatory. Normally it passes under standard conditions. Adjust manually for stricter needs.
	tSU.WS(MIN) LOW	Setup and hold times for the low level of the word select signal. The system automatically analyzes all relevant samples and outputs the minimum value (the worst timing waveform) as the result.
	tHD.WS(MIN) LOW	Hold time for the low-level of the word select signal. The system automatically analyzes all relevant low-level hold time samples on the screen (main time base), and outputs the

		minimum value (the worst timing waveform) as the result.
	tSU.WS(MIN) HIGH	Setup time for the high-level of the word select signal. The system automatically analyzes all relevant high-level setup time samples on the screen (main time base), and outputs the minimum value (the worst timing waveform) as the result.
	tHD.WS(MIN) HIGH	Hold time for the high-level of the word select signal. The system automatically analyzes all relevant high-level hold time samples on the screen (main time base), and outputs the minimum value (the worst timing waveform) as the result.
	tHDelay.WS(MAX)	Hold delay time of the word select signal from the beginning to the end of a single sampling. The system automatically analyzes all relevant word select hold delay samples on the screen (main time base) and outputs the maximum value as the result.
SD	VIL.SD	Low-level voltage of the SDI signal line, as shown in the bottom value of the figure.
	VIH.SD	High-level voltage of the SDI signal line, as shown in the top value of the figure.
	tSU.SD(MIN) LOW	Setup time for the low-level of the data signal. The system automatically analyzes all relevant low-level setup time samples on the screen (main time base) and outputs the minimum value (the worst timing waveform) as the result.
	tHD.SD(MIN) LOW	Hold time for the low-level of the data signal. The system automatically analyzes all relevant low-level hold time samples on the screen (main time base) and outputs the minimum value (the worst timing waveform) as the result.
	tSU.SD(MIN) HIGH	Setup time for the high-level of the data signal. The system automatically analyzes all relevant high-level setup time samples on the screen (main time base) and outputs the minimum value (the worst timing waveform) as the result.
	tHD.SD(MIN) HIGH	Hold time for the high-level of the data signal. The system automatically analyzes all relevant high-level hold time samples on the screen (main time base) and outputs the minimum value (the worst timing waveform) as the result.
	tHDelay.SD(MAX)	Hold delay time of the data signal from the beginning to the end of a single sampling. The system automatically analyzes all relevant data hold delay samples on the screen (main time base) and outputs the maximum value as the result.
WS-SCK	tDELAY(WS-SCK)(	Transmission delay between the word select and clock signals.

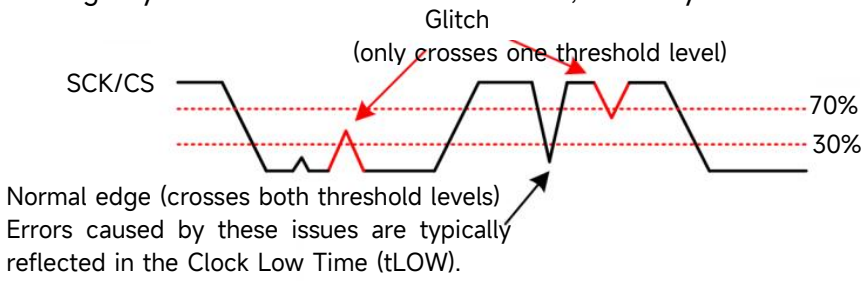
	MIN,MAX)	The system automatically analyzes all relevant delay samples between WS and SCK on the screen (main time base), and outputs the minimum and maximum values as the result.
SD-SCK	tDELAY(SD-SCK) (MIN,MAX)	Transmission delay between the data and clock signals. The system automatically analyzes all relevant delay samples between SD and SCK on the screen (main time base), and outputs the minimum and maximum values as the result.
SD-WS	tDELAY(SD-WS) (MIN,MAX)	Transmission delay between the data and word select signals. The system automatically analyzes all relevant delay samples between SD and WS on the screen (main time base), and outputs the minimum and maximum values as the result.

(5) Glitch Determination Principle

- SCK Signal Glitch Detection

  When the SCK clock signal changes state, if the signal only crosses one threshold level, it is identified as a glitch, as illustrated in the figure below.

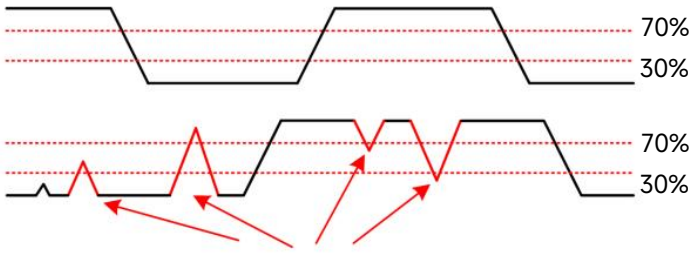
  Note: If a signal crosses both threshold levels in a single transition, it will be treated as a normal edge by the master controller. However, this may cause errors in other test items.



SCK waveform only crosses one threshold level

- WS/SD Signal Glitch Detection

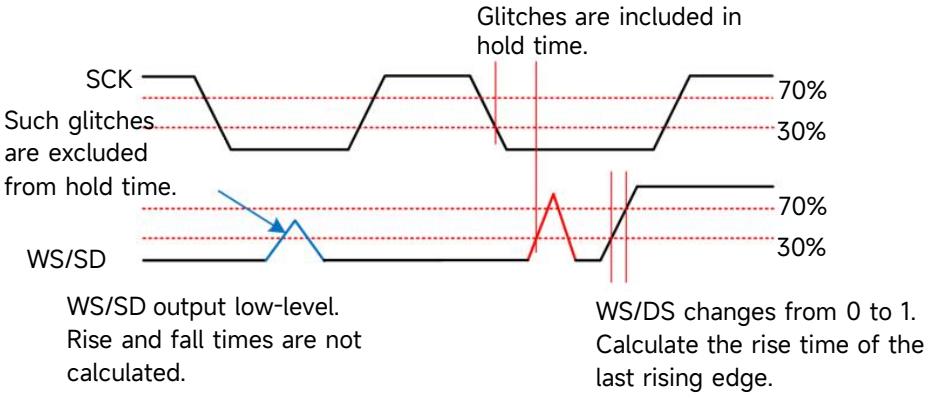
  Glitches occurring on the WS or SD signal lines, whether during the high or low level period of the SCK signal, will always affect the bus operation. Therefore, any detected glitch on WS or SD is considered as error-causing glitch, as illustrated in the figure below.



WS/SD signal Glitch Detection

- WS/SD Signal Glitch Detection

Glitches excluded from rise and fall time statistics: When measuring rise or fall times, the software will automatically filter out and ignore glitches to prevent measurement errors. However, glitches can affect the measurement of setup and hold times. As shown in the figure below, the system will use the most recent transition (from 0 to 1 or from 1 to 0) on the WS/SD line before the SCK clock goes high as the rise or fall time.



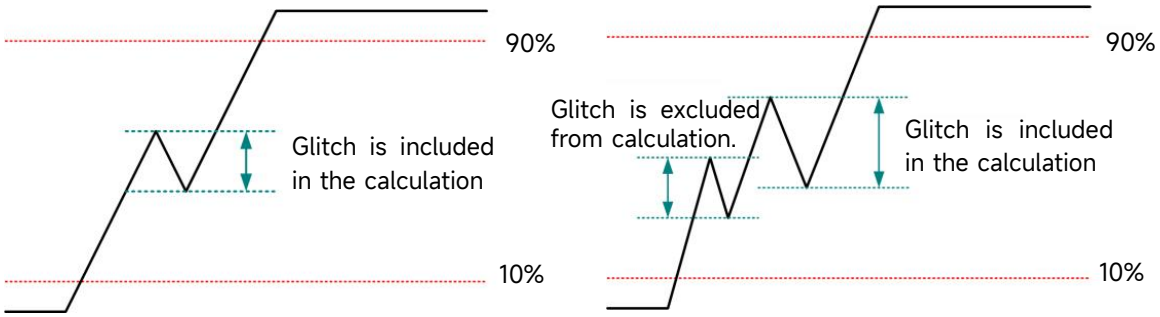
Calculation of rising/falling time

■ Monotonicity Determination Principle

Monotonicity, also referred to as glitch, describes the phenomenon in which a signal exhibits non-monotonic behavior during its rising or falling edge transitions, typically due to interference

The evaluation of glitch voltage amplitude is as follows:

If the measured glitch voltage amplitude exceeds the voltage reference threshold specified in the “Monotonicity” parameter settings, the test result is marked as Fail.



Glitch on rising edge of waveform

Note: The 90% and 10% values in the figure above correspond to the user-defined high and low thresholds, respectively.

(6) I^2S Timing Analysis Configuration – Test Item

Symbol	Parameter	Description
Level	Bus level	The unit is volts (V).
VIH(max)	Maximum input high - level voltage	If the bus voltage is Vcc, then: Vcc<VIH(max)<=1010

VIH(min)	Minimum input high - level voltage	0.5×Vcc<= VIH(min)<Vcc 0.01<=VIL(max)<0.5×Vcc -10<=VIL(min)<= -0.01
VIL(max)	Maximum input low - level voltage	
VIL(min)	Minimum input low - level voltage	
tJitter(max)	Maximum jitter time of clock	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tHIGH(min)	Minimum time of clock high - level	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tLOW(min)	Minimum time of clock low - level	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tR(max)	Maximum rise time of clock	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tF(max)	Maximum fall time of clock	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tSU(min)	Minimum setup time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tHD(min)	Minimum hold time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tHDelay(max)	Maximum hold delay time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tDelay(min)	Minimum transmission delay time	The unit is nanoseconds (ns), with a range from -1000,000,000 to 999,000,000
tDelay(max)	Maximum transmission delay time	The unit is nanoseconds (ns), with a range from -999,000,000 to 1,000,000,000
Over(V)	Default overshoot value	The unit is volts (V), with a range from 0.01 to 1000.
Monot(V)	Default monotonicity value	
High Thresh	High threshold	The unit is percent (%), with a range from 30 to 90.
Mid Thresh	Middle threshold	The unit is percent (%), with a range from 20 to 80.
Low Thresh	Low threshold	The unit is percent (%), with a range from 10 to 70. The setting must satisfy the condition: Low Thresh<=High Thresh − 20.

23.6. Reset Timing Analysis

Reset Timing Analysis is used to test and analyze the timing characteristics of system reset signals.

Reset devices are typically equipped with a Watchdog function, and some also support Manual Reset.

Reset Timing Analysis can measure the electrical characteristics of reset signals, watchdog signals, and manual reset signals, and it supports exporting the test results to a report.

23.6.1. Analysis Setting

(1) Reset Source

Click Reset Source drop-down list to select C1 - C8 as the reset source.

(2) WDI Source

Click Word Select Source drop-down list and select C1 - C8 to set the signal source for the watchdog (WDI) line.

(3) MR Source

Click Data Source drop-down list and select C1 - C8 or None to set the signal source for the manual reset (MR) line.

(4) Reset Level

Click Reset Level drop-down list and select Low Level or High Level to define the active level of the reset signal.

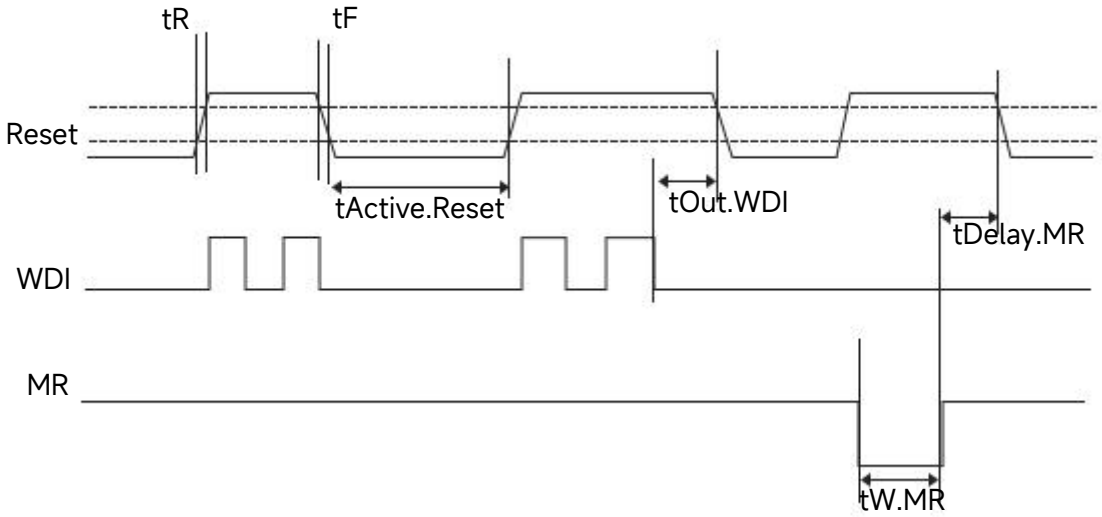
23.6.2. Parameter Setting

(1) Reset Timing Analysis Test Item

Symbol	Parameter	Minimum	Maximum	Unit
Level	Bus level			
VOH	Output signal high level	-	0.8*VDD	V
VOL	Output signal low level	0.4	-	V
tR	Clock rise time	-	100	μs
tF	Clock fall time	-	100	μs
tActive	Active time	-	6	μs
tOut	Timeout	300	112	ms
tW	Pulse width	-	1	μs
tDelay	Delay time	100	-	ns
$Over_+$	Positive overshoot	This item is part of optional signal quality testing and is not defined in standard specifications. Users may define the acceptable test range according to their specific needs.		
$Over_-$	Negative overshoot			
Monot	Monotonicity			
Glitch	Glitch			

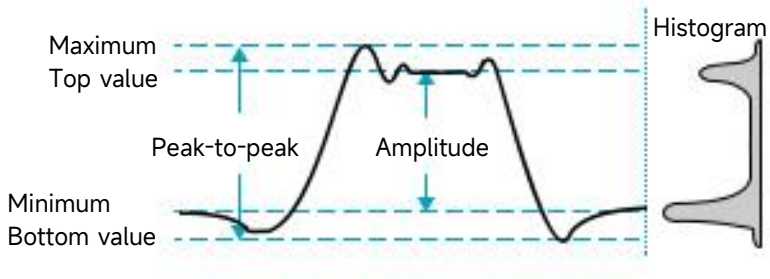
(2) Signal Position

The specific signal positions corresponding to each Reset test item are shown below.



The calculation principles for voltage-related test items in Reset, VOH (high threshold) and VOL (low threshold) are illustrated in the figure below.

- VOH: The voltage measured from the flat top of the waveform to ground, representing the top value.
- VOL: The voltage measured from the flat bottom of the waveform to ground, representing the bottom value.



Calculation of Top and Bottom Value

(3) Reset Timing Analysis Test Item

Signal	Test Item	Description
Reset	VOH.Reset	Output high level of the Reset signal line.
	VOL.Reset	Output low level of the Reset signal line.
	tR.Reset(max)	Rise time (tR). During measurement, the system records all rise time samples on the main time base and outputs the maximum value (worst-case timing) as the measurement result. Rise time (tR). During measurement, the system records all rise time samples on the main time base and outputs the maximum value (worst-case timing) as the measurement result.

	tF.Reset(max)	Fall time (tF). During measurement, the system records all fall time samples on the main time base and outputs the maximum value (worst-case timing) as the measurement result.
	tActive.Reset(min)	Assertion time (tActive), i.e., the duration when the Reset signal is active. The system records all assertion time samples on the main time base and outputs the minimum value (worst-case timing) as the measurement result.
	Over+.Reset	Positive overshoot test for the Reset signal line. The test is marked as Fail if the positive overshoot exceeds the preset limit.
	Over-.Reset	Negative overshoot test for the Reset signal line. The test is marked as Fail if the negative overshoot exceeds the preset limit. Negative overshoot test for the Reset signal line. The test is marked as Fail if the negative overshoot exceeds the preset limit.
	Monot.Reset	Monotonicity test for the Reset signal line. It refers to points where the signal exhibits non-monotonic behavior on the edge. The test is marked as Fail if the measured value exceeds the preset limit.
	Glitch.Reset	Glitch test for the Reset signal line. It refers to cases where the signal crosses only one threshold without forming a valid edge. The system records all glitches on the main time base and outputs the one with the largest voltage drop.
WDI	tOut.WDI	Timeout test for the WDI signal line. Measures the duration from when the watchdog fails to toggle normally to the automatic timeout reset. The system records all samples on the main time base and outputs the maximum and minimum values.
MR	tW.MR(min)	Pulse width test for the MR signal line. Measures the duration when the MR signal is active. The system records all samples on the main time base and outputs the minimum value.
	tDelay.MR(max)	Reset delay test for the MR signal line. Measures the time from assertion of the manual reset signal to assertion of the Reset signal. The system records all samples on the main time base and

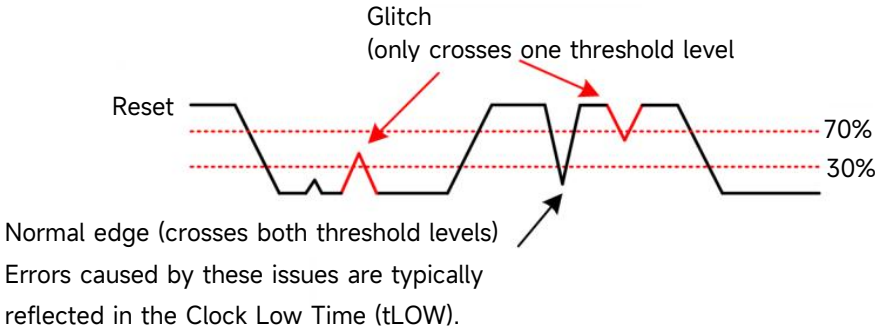
		outputs the maximum value. Reset delay test for the MR signal line. Measures the time from assertion of the manual reset signal to assertion of the Reset signal. The system records all samples on the main time base and outputs the maximum value.

(4) Glitch Determination Principle

■ Reset Signal Glitch Detection

When the Reset signal changes state, if the signal only crosses one threshold level, it is identified as a glitch, as illustrated in the figure below.

Note: If a signal crosses both threshold levels in a single transition, it will be treated as a normal edge by the master controller. However, this may cause errors in other test items.



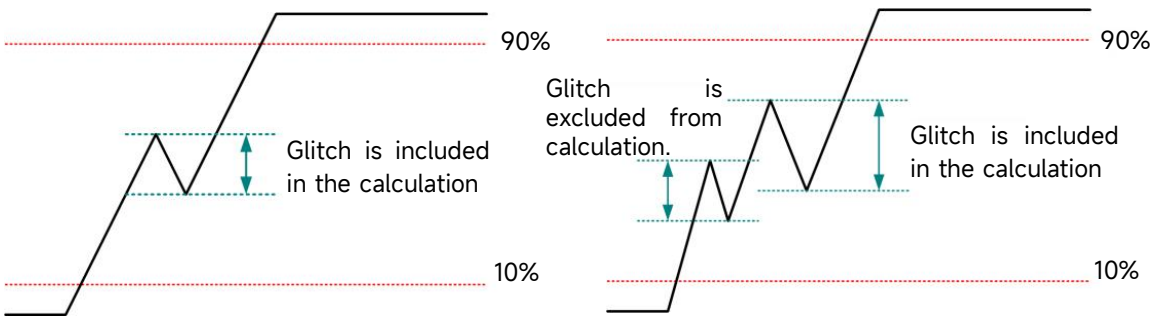
Reset waveform only crosses one threshold level

■ Monotonicity Determination Principle

Monotonicity, also referred to as glitch, describes the phenomenon in which a signal exhibits non-monotonic behavior during its rising or falling edge transitions, typically due to interference

The evaluation of glitch voltage amplitude is as follows:

If the measured glitch voltage amplitude exceeds the voltage reference threshold specified in the “Monotonicity” parameter settings, the test result is marked as Fail.



Glitch on rising edge of waveform

Note: The 90% and 10% values in the figure above correspond to the user-defined high and low thresholds, respectively.

(5) Reset Timing Analysis Configuration – Test Item

Symbol	Parameter	Description
Level	Bus level	The unit is volts (V), with a range from 1 to 1,000.
VOH(min)	Minimum output high voltage	The unit is volts (V), with a range from 1 to 1,000.
VOL(max)	Maximum output high voltage	The unit is volts (V), with a range from 1 to 1,000.
tR(max)	Maximum rise time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tF(max)	Maximum fall time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tActive.Reset(min)	Minimum reset assertion time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tOut.WDI(max)	Maximum WDI timeout	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000. The setting must satisfy the condition: tOut.WDI(max) > tOut.WDI(min)
tOut.WDI(min)	Minimum WDI timeout	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000. The setting must satisfy the condition: tOut.WDI(max) > tOut.WDI(min)
tW.MR(min)	Minimum manual reset pulse width	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tDelay.MR(max)	Maximum manual reset delay time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
Overshoot	Default overshoot value	The unit is volts (V), with a range from 0.01 to 1,000.
Monot	Default monotonicity value	The unit is volts (V), with a range from 0.01 to 1,000.
High Thresh	High threshold	The unit is percent (%), with a range from 20 to 90. The setting must satisfy the condition: Low Thresh < High Thresh – 10.
Low Thresh	Low threshold	The unit is percent (%), with a range from 10 to 80. The setting must satisfy the condition: Low Thresh < High Thresh – 10.

23.7. Switch Timing Analysis

A Switch signal refers to a signal that alternates between logic high and low states. Switch timing analysis measures the high and low levels, rise and fall times, and switching cycle of the signal during state transitions.

23.7.1. Analysis Setting

(1) Source

Click Source drop-down list to select C1 - C8 as the switch source.

(2) Polarity

Click Polarity drop-down list to select positive or negative.

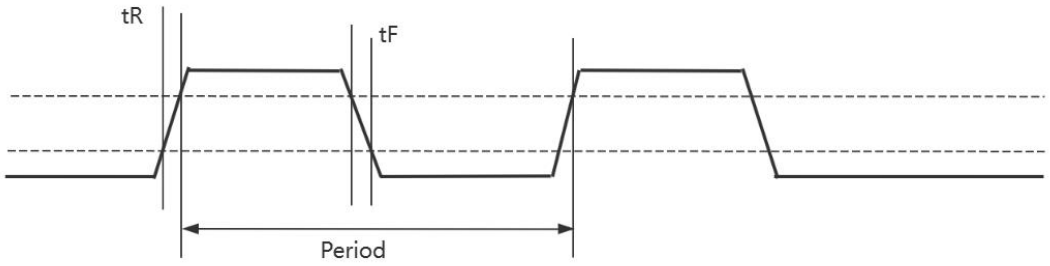
23.7.2. Parameter Setting

(1) Switch Timing Analysis Test Item

Symbol	Parameter	Minimum	Maximum	Unit
Level	Bus level			
VIH	Input high-level voltage	3.8	2.31	V
VIL	Input low-level voltage	0.99	-0.5	V
tR	Rise time	100	---	ns
tF	Fall time	100	---	ns
Period	Period	---	---	ns

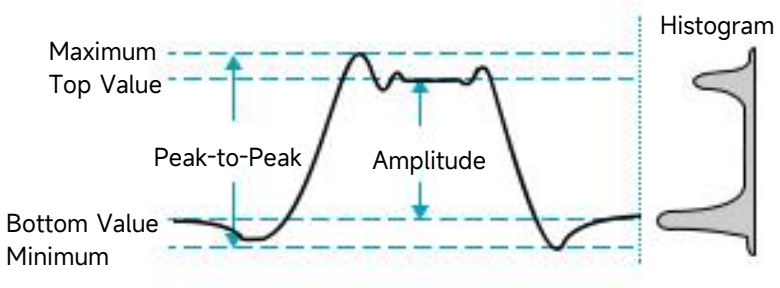
(2) Signal Position

The specific signal positions for Switch timing–related test items are shown below.



The calculation principles for voltage-related test items in Switch, VIH (high threshold) and VIL (low threshold) are illustrated in the figure below.

- VIH: The voltage measured from the flat top of the waveform to ground, representing the top value.
- VIL: The voltage measured from the flat bottom of the waveform to ground, representing the bottom value.



Calculation of Top and Bottom Value

(3) Switch Timing Analysis Test Item

Test Item	Description
VIH	Input high-level voltage
VIL	Input low-level voltage
Period	Cycle of the test signal during switching. The system records all samples on the main time base, calculates the maximum and minimum values, and outputs the maximum value as the result.
tR(max)	Rise time of the test signal. The system records all samples on the main time base and outputs the maximum value as the result.
tF(max)	Fall time of the test signal. The system records all samples on the main time base and outputs the maximum value as the result.

(4) Switch Timing Analysis Configuration – Test Item

Symbol	Parameter	Description
Level	Bus level	The unit is volts (V). If the bus voltage is Vcc, then: Vcc<Vih(max)<=1010 0.5×Vcc<= Vih(min)<Vcc 0.01<=Vil(max)<0.5×Vcc -10<=Vil(min)<=-0.01
VIH(max)	Maximum input high - level voltage	
VIH(min)	Minimum input high - level voltage	
VIL(max)	Maximum input low - level voltage	
VIL(min)	Minimum input low - level voltage	
tR(max)	Maximum rise time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tF(max)	Maximum fall time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
High Thresh	High threshold	The unit is percent (%), with a range from 20 to 90.

		The setting must satisfy the condition: Low Thresh < High Thresh - 10
Low Thresh	Low threshold	The unit is percent (%), with a range from 10 to 80. The setting must satisfy the condition: Low Thresh < High Thresh - 10

23.8. Uart Timing Analysis

UART is a serial, asynchronous, full-duplex communication protocol. It features a simple communication circuit, allowing bidirectional data transfer using only a pair of transmission lines. The UART timing analysis function can automatically measure the timing parameters of UART signals and export the results to a report.

23.8.1. Analysis Setting

(1) Source

Click Source drop-down list to select C1 - C8 as the Uart source.

(2) Bitrate

Click Bitrate drop-down list to select 2400 bps, 4800 bps, 9600 bps, 19200 bps, 38400 bps, 57600 bps, 115200 bps, or a custom bitrate.

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

(3) Bit Time Error

Bit time error refers to the deviation between the calculated bit time based on the signal edge and the configured bitrate.

23.8.2. Parameter Setting

(1) Bus Level Setting

- Bus level: Set the bus level for timing analysis, which refers to the input voltage (V_{cc}) of the signal. Typically, the input voltage for Uart is 3.30 V. If the input voltage is 1.8 V or another value, it can be adjusted using the Multipurpose rotary knob. Adjusting the bus level (V_{cc}) will affect the values of V_{IH} and V_{IL}. The relationship between V_{cc}, V_{IH}, and V_{IL} is shown in the table below.
- Input high-level/Low-level voltage (V_{IH}, V_{IL}): These input values are determined by V_{cc} and follow the relationship specified in the table. They can also be manually adjusted by using the Multipurpose rotary knob. However, modifying V_{IH} or V_{IL} will not affect the value of V_{cc}.

Symbol	Parameter	Minimum	Maximum	Unit
V_{IH}	Input high-level voltage	0.7×Vcc	Vcc+0.5	V
V_{IL}	Input low-level voltage	-0.5	0.3×Vcc	V

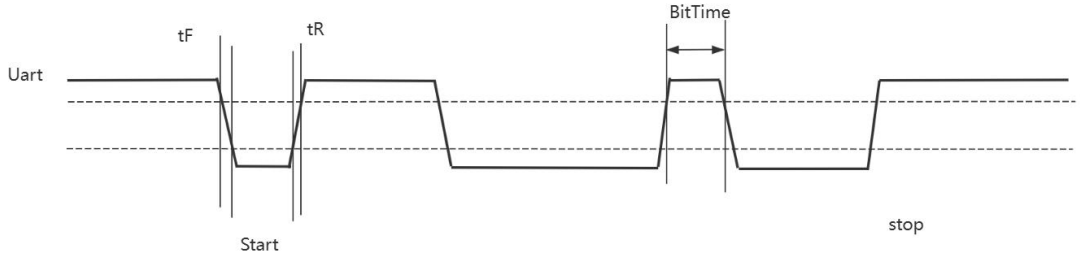
- Each of the remaining parameters has its own specific standard. Refer to the table below for standard parameter settings. If the users' standard differs from the one shown in the table, double-click the parameter input field to open the numeric keypad to enter a specific standard. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the parameter, then use the Multipurpose rotary knob to adjust the standard value. Click Default to restore the default settings.

(2) Uart Timing Analysis Test Item

Symbol	Parameter	Minimum	Maximum	Unit
VIH	Input high-level voltage	0.7Vcc	Vcc + 0.5	V
VIL	Input low-level voltage	-0.5	0.3Vcc	V
tR	Rise time	---	100	ns
tF	Fall time	---	100	ns
Bit Time	Bit time	---	---	ns
Over+	Positive overshoot	This item is part of optional signal quality testing and is not defined in standard specifications. Users may define the acceptable test range according to their specific needs.		V
Over-	Negative overshoot			
Monot	Monotonicity (glitch)			
Glitch	Glitch			

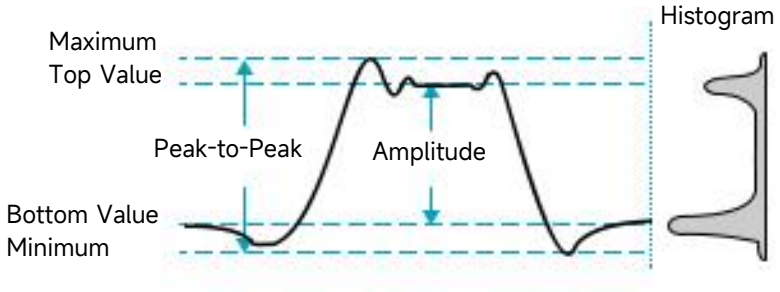
(3) Signal Position

The specific signal positions for Uart timing–related test items are shown below.



The calculation principles for voltage-related test items in Uart, VIH (high threshold) and VIL (low threshold) are illustrated in the figure below.

- VIH: The voltage measured from the flat top of the waveform to ground, representing the top value.
- VIL: The voltage measured from the flat bottom of the waveform to ground, representing the bottom value.



Calculation of Top and Bottom Value

(4) Uart Timing Analysis Test Item

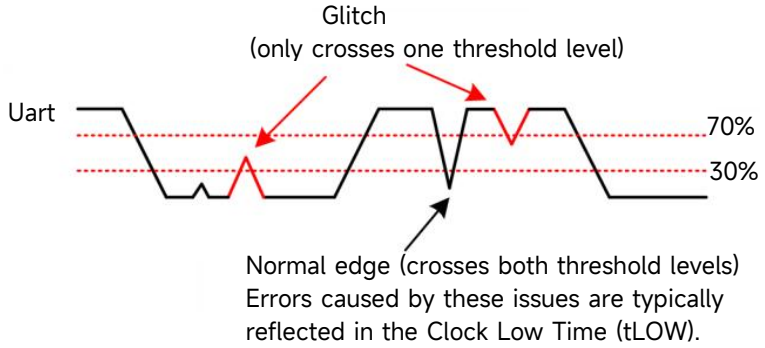
Test Item	Description
VIH	Input high-level voltage
VIL	Input low-level voltage
tR(max)	Rise time of the signal: During measurement, the system records all rise time samples on the screen (main time base) and outputs the maximum value as the result (worst-case timing).
tF(max)	Fall time of the signal: During measurement, the system records all fall time samples on the screen (main time base) and outputs the maximum value as the result (worst-case timing).
BitTime	Bit duration. During testing, the system records all bit time samples on the screen (main time base) and outputs the maximum and minimum values of the samples.
Over+	Positive overshoot test item. If the positive overshoot exceeds the preset threshold, the software marks the test result as Fail.
Over-	Negative overshoot test item. If the negative overshoot exceeds the preset threshold, the software marks the test result as Fail.
Monot	Signal monotonicity (glitch) test item. Detects non-monotonic positions on the signal edge. If the measured value exceeds the preset threshold, the software marks the test result as Fail.
Glitch	Signal glitch test item. Refers to the condition where the signal crosses only one threshold level without forming a valid edge. The system records all glitches across the entire screen (main time base) and outputs the glitch with the maximum voltage difference as the result.

(5) Glitch Determination Principle

■ Uart Signal Glitch Detection

When the Uart signal changes state, if the signal only crosses one threshold level, it is identified as a glitch, as illustrated in the figure below.

Note: If a signal crosses both threshold levels in a single transition, it will be treated as a normal edge by the master controller. However, this may cause errors in other test items.



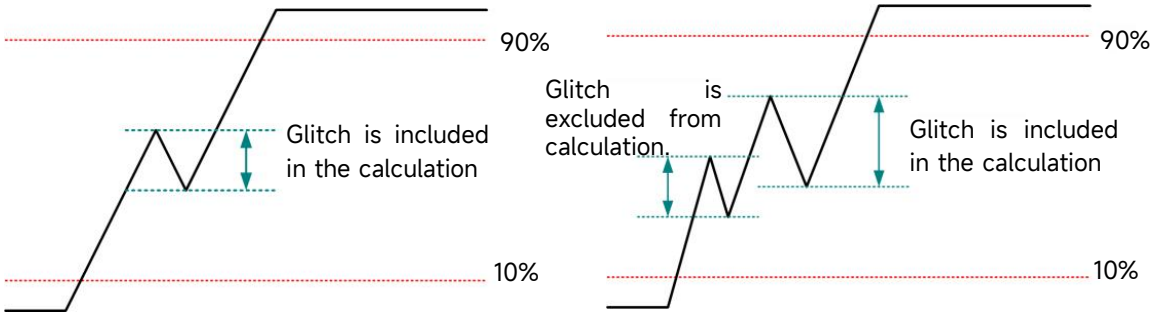
Uart waveform only crosses one threshold level

■ Monotonicity Determination Principle

Monotonicity, also referred to as glitch, describes the phenomenon in which a signal exhibits non-monotonic behavior during its rising or falling edge transitions, typically due to interference

The evaluation of glitch voltage amplitude is as follows:

If the measured glitch voltage amplitude exceeds the voltage reference threshold specified in the “Monotonicity” parameter settings, the test result is marked as Fail.



Glitch on rising edge of waveform

Note: The 90% and 10% values in the figure above correspond to the user-defined high and low thresholds, respectively.

(6) Uart Timing Analysis Configuration – Test Item

Symbol	Parameter	Description
Level	Bus Level	The unit is volts (V).
Vih(max)	Maximum input high - level voltage	If the bus voltage is Vcc, then: Vcc<Vih(max)<=1010

Vih(min)	Minimum input high - level voltage	0.5×Vcc<= Vih(min)<Vcc 0.01<=Vil(max)<0.5×Vcc -10<=Vil(min)<=-0.01
Vil(max)	Maximum input low - level voltage	
Vil(min)	Minimum input low - level voltage	
tR(max)	Maximum rise time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tF(max)	Minimum rise time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
Overshoot	Default overshoot value	The unit is nanoseconds (ns), with a range from 0.01 to 1,000.
Monot	Default monotonicity value	The unit is nanoseconds (ns), with a range from 0.01 to 1,000.
ThresholdH	High threshold	The unit is percent (%), with a range from 20 to 90. The setting must satisfy the condition: Low Thresh < High Thresh − 10.
ThresholdL	Low threshold	The unit is percent (%), with a range from 10 to 80. The setting must satisfy the condition: Low Thresh < High Thresh − 10.

23.9. LIN Timing Analysis

A LIN signal is a signal with switching logic level states. LIN timing analysis measures the signal's high and low levels, rise and fall times, and switching cycle during level transitions.

23.9.1. Analysis Setting

(1) Source

Click Source drop-down list to select C1 - C8 as the LIN source.

(2) Bitrate

Click Bitrate drop-down list to select 1.2 kbps, 2.4 kbps, 4.8 kbps, 9.6 kbps, 10.417 kbps, 19.2 kbps, 20 kbps, or a custom bitrate.

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

(3) Bit Time Error

Bit time error refers to the deviation between the calculated bit time based on the signal edge and the configured bitrate.

(4) Filtering Conditions

Click Filtering Conditions drop-down list to configure the filtering condition for LIN data. This determines whether to analyze PID transmission frames.

- None: No filtering; all data will be tested and analyzed.
- PID: Tests and analyzes only the transmission frames corresponding to the selected protected ID.

(5) PID

Set the filtering PID, double-click PID input field to open the numeric keypad to set the address. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select PID input field, use the arrow keys ←, → to position the cursor, then use the Multipurpose to adjust the address. Adjustable range: 00 - 7F.

23.9.2. Parameter Setting

(1) Bus Level Setting

- Bus level: Set the bus level for timing analysis, which refers to the input voltage (V_{cc}) of the signal. Typically, the input voltage for LIN is 3.30 V. If the input voltage is 1.8 V or another value, it can be adjusted using the Multipurpose rotary knob. Adjusting the bus level (V_{cc}) will affect the values of V_{IH} and V_{IL}. The relationship between V_{cc}, V_{IH}, and V_{IL} is shown in the table below.
- Input high-level/Low-level voltage (V_{IH}, V_{IL}): These input values are determined by V_{cc} and follow the relationship specified in the table. They can also be manually adjusted by using the Multipurpose rotary knob. However, modifying V_{IH} or V_{IL} will not affect the value of V_{cc}.

Symbol	Parameter	Minimum	Maximum	Unit
V_{IH}	Input high-level voltage	0.7×Vcc	Vcc+0.5	V
V_{IL}	Input low-level voltage	-0.5	0.3×Vcc	V

- Each of the remaining parameters has its own specific standard. Refer to the table below for standard parameter settings. If the users' standard differs from the one shown in the table, double-click the parameter input field to open the numeric keypad to enter a specific standard. For details on using the numeric keypad, refer to section 5.8 Parameter Setting. Alternatively, select the parameter, then use the Multipurpose rotary knob to adjust the standard value. Click Default to restore the default settings.

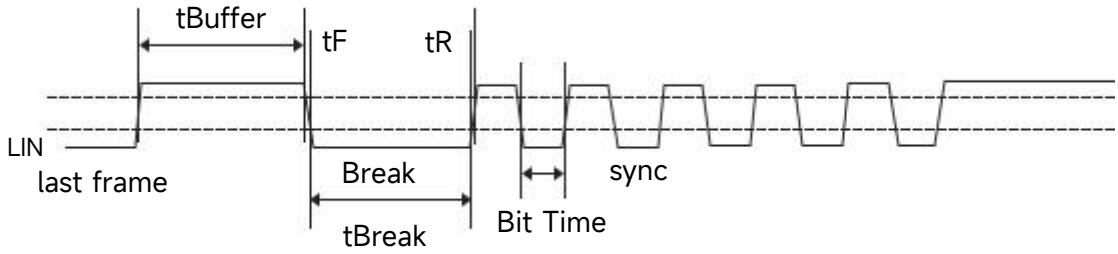
(2) LIN Timing Analysis Test Item

Symbol	Parameter	Minimum	Maximum	Unit

VIH	Input high-level voltage	0.7Vcc	Vcc + 0.5	V
VIL	Input low-level voltage	-0.5	0.3Vcc	V
tR	Rise time	---	100	ns
tF	Fall time	---	100	ns
tBreak	Break time	---	---	ns
Bit Time	Bit time	---	---	ns
tBuffer	Idle time	1	---	ms
Over+	Positive overshoot	This item is part of optional signal quality testing and is not defined in standard specifications. Users may define the acceptable test range according to their specific needs.		V
Over-	Negative overshoot			
Monot	Monotonicity (glitch)			
Glitch	Glitch			

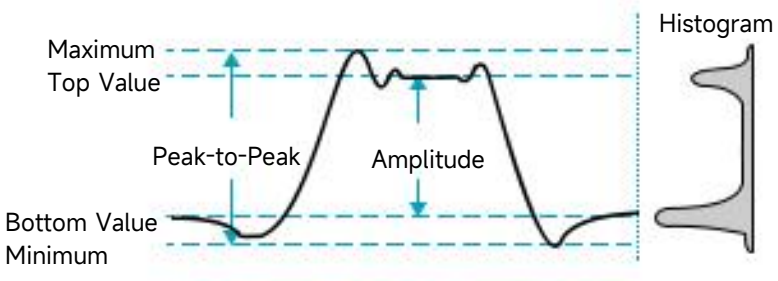
(3) Signal Position

The specific signal positions for LIN timing–related test items are shown below.



The calculation principles for voltage-related test items in LIN, VIH (high threshold) and VIL (low threshold) are illustrated in the figure below.

- VIH: The voltage measured from the flat top of the waveform to ground, representing the top value.
- VIL: The voltage measured from the flat bottom of the waveform to ground, representing the bottom value.



Calculation of Top and Bottom Value

(4) LIN Timing Analysis Test Item

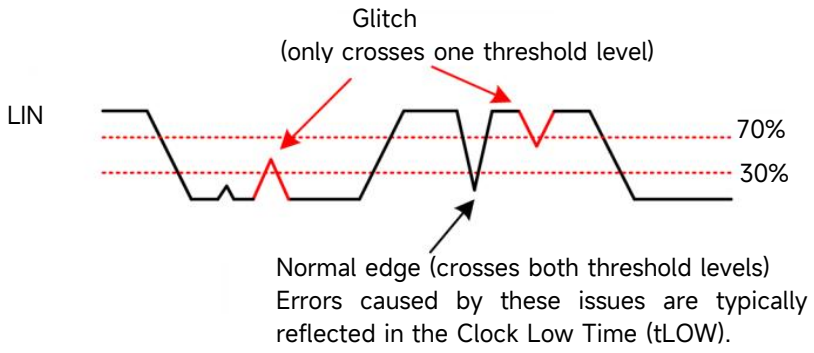
Test Item	Description
VIH	Input high-level voltage
VIL	Input low-level voltage
tR(max)	Rise time of the signal: During measurement, the system records all rise time samples on the screen (main time base) and outputs the maximum value as the result (worst-case timing).
tF(max)	Fall time of the signal: During measurement, the system records all fall time samples on the screen (main time base) and outputs the maximum value as the result (worst-case timing).
tBreak(min)	Break field length: During measurement, the system identifies all Break fields at each frame header on the screen (main time base), records their lengths, and outputs the minimum value among all samples as the result.
BitTime	Bit duration: During testing, the system records all bit time samples on the screen (main time base) and outputs the maximum and minimum values of the samples.
tBuffer(min)	Idle time: Refers to the idle duration of the signal between two frames. During measurement, the system records all idle time samples on the screen (main time base) and outputs the minimum value (worst-case timing) as the result.
Over+	Positive overshoot test item. If the positive overshoot exceeds the preset threshold, the software marks the test result as Fail.
Over-	Negative overshoot test item. If the negative overshoot exceeds the preset threshold, the software marks the test result as Fail.
Monot	Signal monotonicity (glitch) test item. Detects non-monotonic positions on the signal edge. If the measured value exceeds the preset threshold, the software marks the test result as Fail.
Glitch	Signal glitch test item. Refers to the condition where the signal crosses only one threshold level without forming a valid edge. The system records all glitches across the entire screen (main time base) and outputs the glitch with the maximum voltage difference as the result.

(5) Glitch Determination Principle

- LIN Signal Glitch Detection

  When the LIN signal changes state, if the signal only crosses one threshold level, it is identified as a glitch, as illustrated in the figure below.

  Note: If a signal crosses both threshold levels in a single transition, it will be treated as a normal edge by the master controller. However, this may cause errors in other test items.



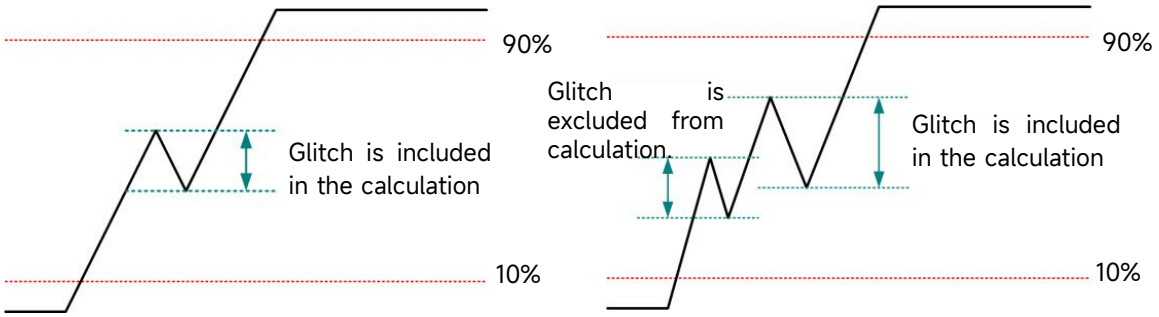
LIN waveform only crosses one threshold level

■ Monotonicity Determination Principle

Monotonicity, also referred to as glitch, describes the phenomenon in which a signal exhibits non-monotonic behavior during its rising or falling edge transitions, typically due to interference

The evaluation of glitch voltage amplitude is as follows:

If the measured glitch voltage amplitude exceeds the voltage reference threshold specified in the "Monotonicity" parameter settings, the test result is marked as Fail.



Glitch on rising edge of waveform

Note: The 90% and 10% values in the figure above correspond to the user-defined high and low thresholds, respectively.

(6) LIN Timing Analysis Configuration – Test Item

Symbol	Parameter	Description
Level	Bus Level	The unit is volts (V).
Vih(max)	Maximum input high - level voltage	If the bus voltage is Vcc, then: Vcc<Vih(max)<=1010 0.5×Vcc<= Vih(min)<Vcc 0.01<=Vil(max)<0.5×Vcc -10<=Vil(min)<=-0.01
Vih(min)	Minimum input high - level voltage	
Vil(max)	Maximum input low - level voltage	
Vil(min)	Minimum input low - level voltage	
tR(max)	Maximum rise time	The unit is nanoseconds (ns), with a range

		from 1 to 1,000,000,000.
tF(max)	Minimum rise time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
tBuffer(min)	Minimum idle time	The unit is nanoseconds (ns), with a range from 1 to 1,000,000,000.
Overshoot	Default overshoot value	The unit is volts (V), with a range from 0.01 to 1,000.
Monot	Default monotonicity value	The unit is volts (V), with a range from 0.01 to 1,000.
ThresholdH	High threshold	The unit is percent (%), with a range from 20 to 90. The setting must satisfy the condition: Low Thresh < High Thresh − 10.
ThresholdL	Low threshold	The unit is percent (%), with a range from 10 to 80. The setting must satisfy the condition: Low Thresh < High Thresh − 10.

24. 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 marker navigation.

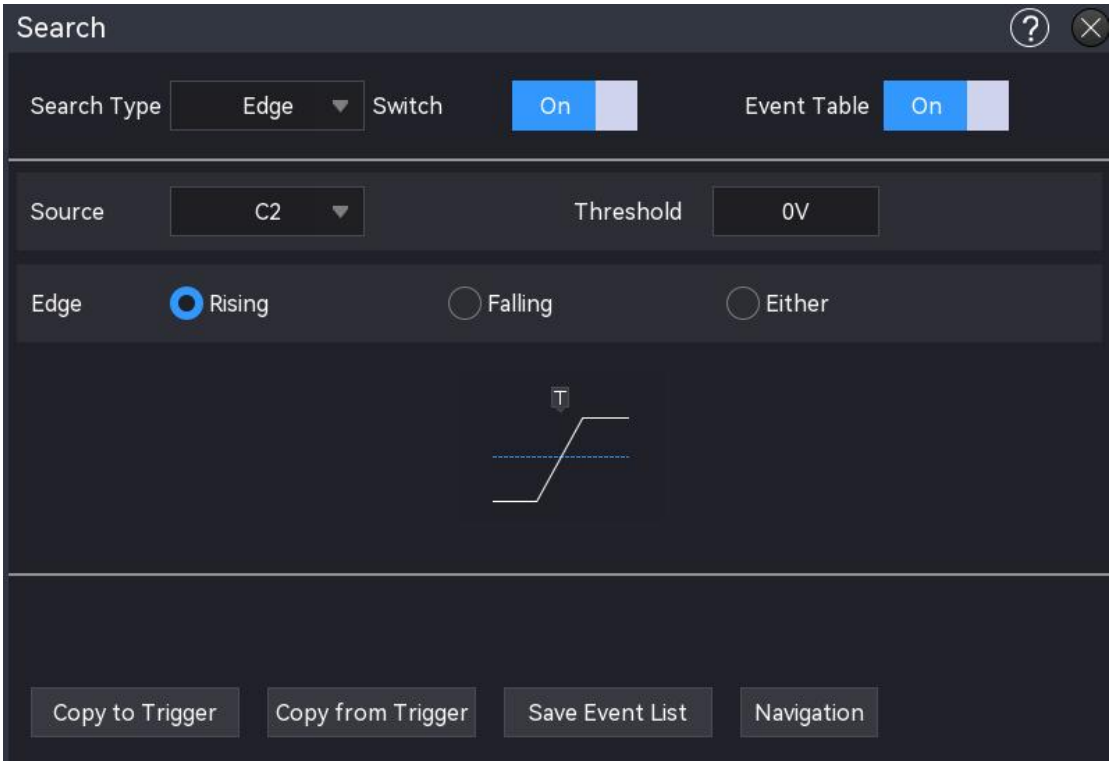
24.1. Search

Access the Search menu using the following methods.

- Click Home icon , select the search icon to open the search function.
- If the search function is added to the toolbar, click search icon in the top-right corner of the toolbar 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.

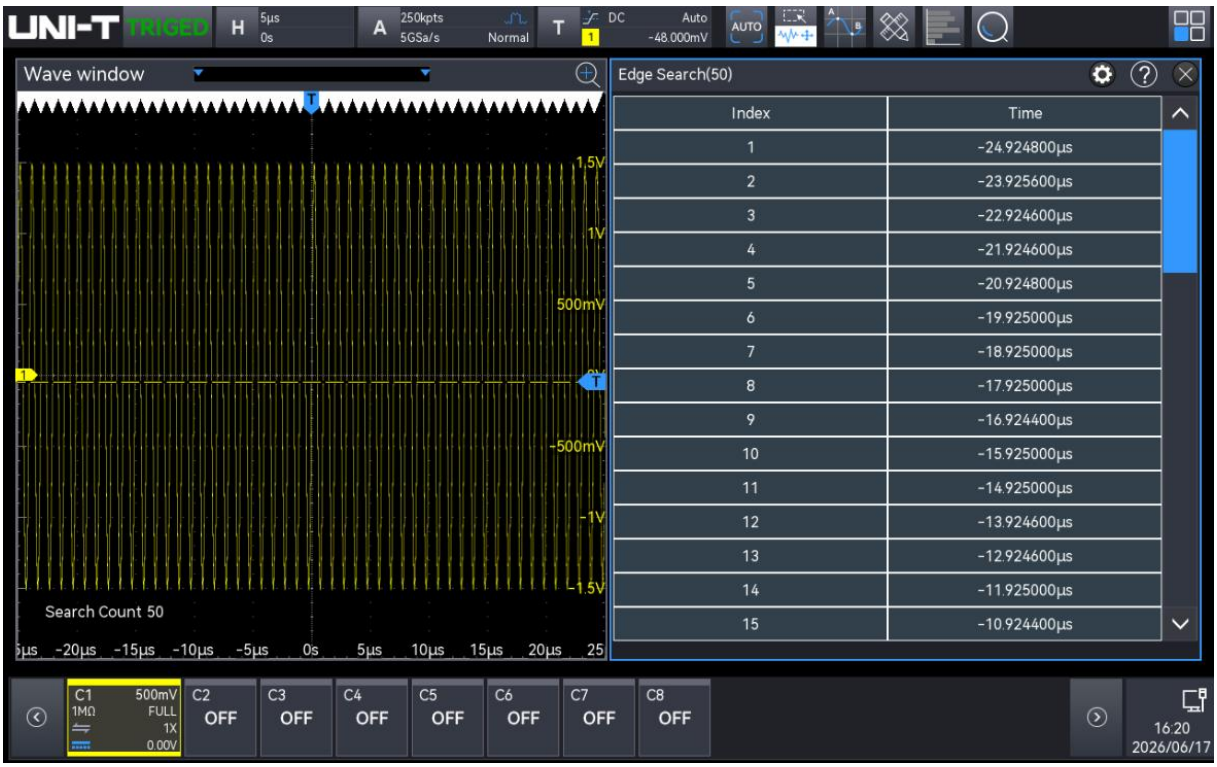
   - Edge search: Click Search Type and select Edge, for the edge type setting (source, trigger coupling, edge type, trigger level). Refer to Edge Triggering for more details.
   - Pulse width search: Click Search Type and select Pulse width, for the pulse width setting (source, polarity, upper limit, lower limit). Refer to Pulse Width Tiggering 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 Triggering 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 Triggering for more details.
   - Window search: Click Search Type and select Window, for the window setting (source, edge type, search position, over-amplitude time). Refer to Window Triggering 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 Triggering for more details.
   - Timeout search: Click Search Type and select Timeout, for the timeout setting (source, edge type, timeout type). Refer to Timeout Triggering 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 Triggering 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 Triggering 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 Triggering for more details.
   - Pattern search: Click Search Type and select Code pattern, for the code pattern setting (source, pattern). Refer to Pattern Triggering 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. After opening the event table, the interface appears as shown in the figure below. The event table lists all waveform search points corresponding to the waveforms currently displayed. When the waveform is zoomed or adjusted, the search results in the event table update accordingly.

The following operations can be performed on the event table:

- When a large number of search results exist, the search points are displayed in blocks. Expand a block to view detailed information for each search point.
- Click any row in the event 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 event 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 to open the Help document.
- Click icon in the top-right corner to close the event 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. Copy from trigger

   Click from trigger 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 Copy from trigger will copy the edge trigger settings to the Edge search settings.

   To use the Copy from trigger function, the search type should be set first. Then, copy the corresponding trigger settings from the trigger menu.

6. Save Event List

   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 (only when a USB flash drive is detected). For the setting steps, refer to the section of Storage 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.

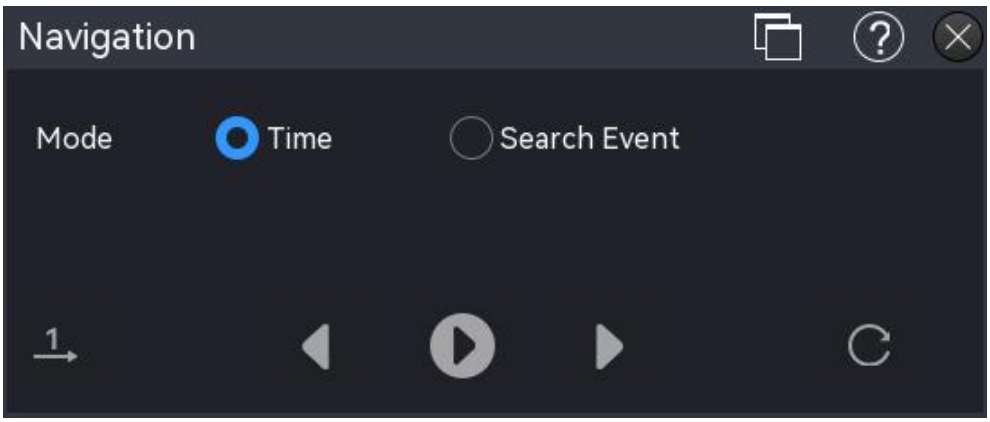
7. Navigation

   Click Navigation key to jump to the navigation module.

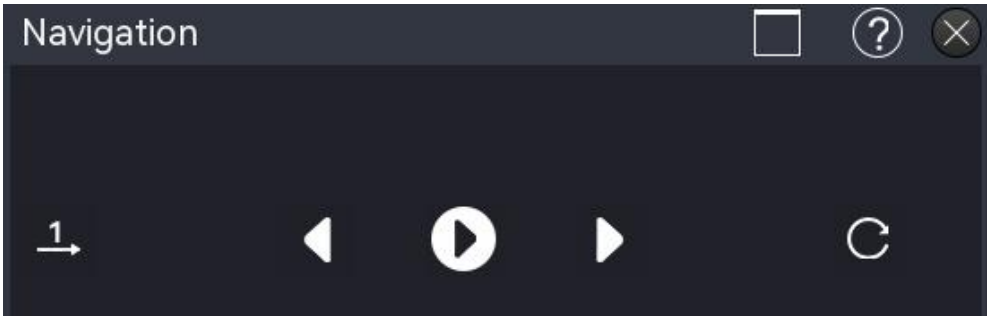
24.2. Navigation

Access the Navigation menu using the following methods.

- Click Home icon, select the navigation icon to open the navigation function.
- If the navigation function is added to the toolbar, click navigation icon in the top-right corner of the toolbar to open the navigation function.
- In the search menu, click navigation 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

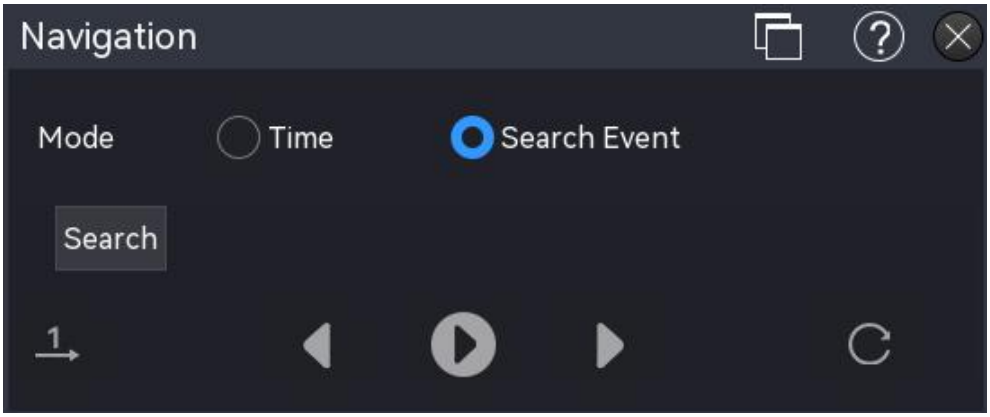
  The playback order can be set to sequential playback or reverse playback. Click icon in the lower right corner of the menu to switch the playback order.

  : 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. For details on the search

settings, refer to *Search* Function.

- 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.

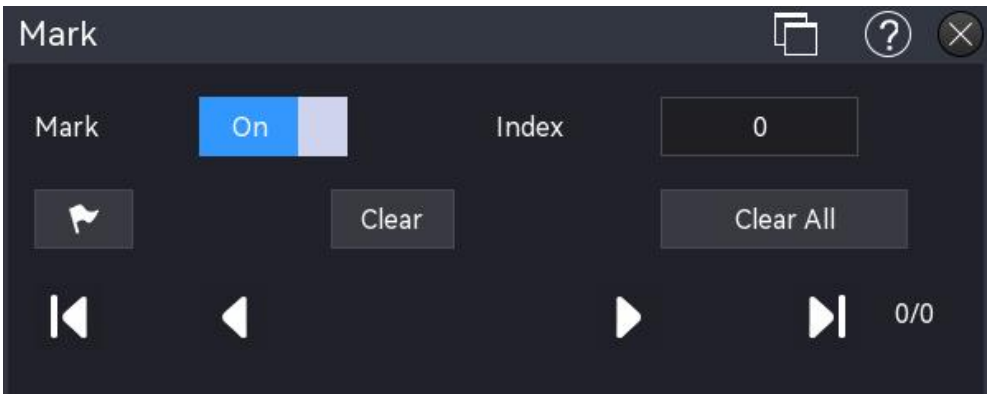
  : Play from the rightmost search point to the leftmost search point.

24.3. Marker Navigation

Markers allow you to manually place marker points and navigate between them. The marker navigation interface is shown in the figure below.

Access the Marker Navigation menu using the following methods.

- Click Home icon, select the marker icon to open the marker navigation menu.
- If the marker navigation function is added to the toolbar, click marker navigation icon in the top-right corner of the toolbar to open the marker navigation function.
- In the search menu, click navigation to open the 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 5.8 Parameter Setting. Alternatively, click Index input field, then use the Multipurpose rotary knob to adjust the index.

(3) : Mark the midpoint of the current waveform area with the marking symbol ▽.

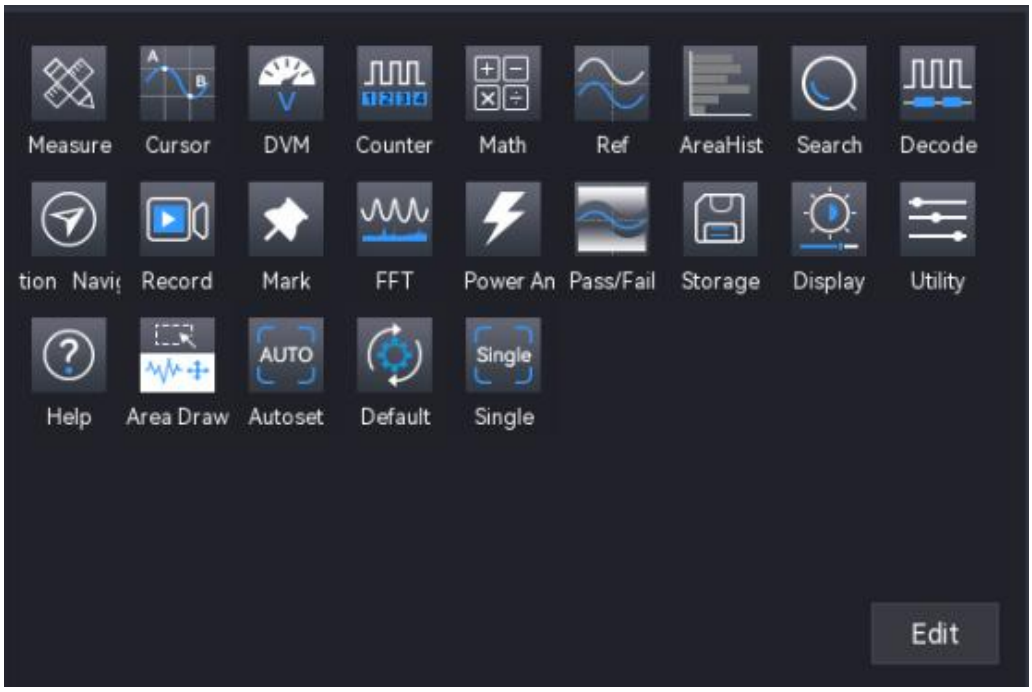
(4) Clear: Clear the midpoint marker in the waveform area.

(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.

25. 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, UPO6000HD toolbar displays 5 icons: Area Draw, Cursor, Measure, Area Histogram, and Search.

(4) Delete

Delete all the menus from the toolbar.

26. Area 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.

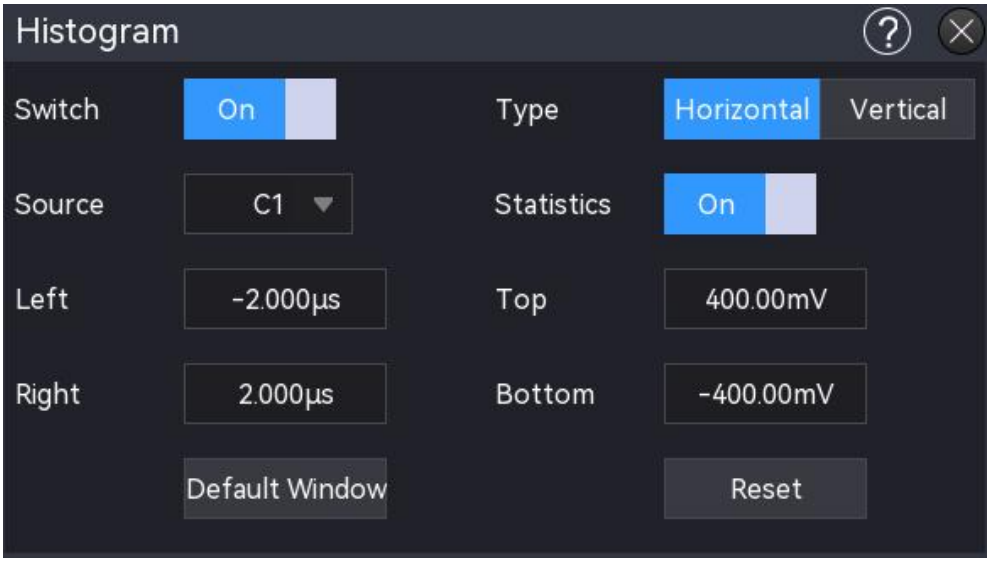
Jitter analysis: The histogram is essential for jitter analysis, enabling statistical evaluation of the value range and distribution of many data samples. This allows verification of product performance and quality, as well as identification and diagnosis of intermittent issues. It is particularly valuable 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 methods.

- Click Home icon , select the histogram icon to open the histogram menu.
- If the histogram function is added to the toolbar, click histogram icon in the top-right corner of the toolbar to open the histogram menu.
- If histogram statistical window appears, 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 list to select the source for histogram statistical analysis. The available source channels are analog channels C1 - C8.

(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.

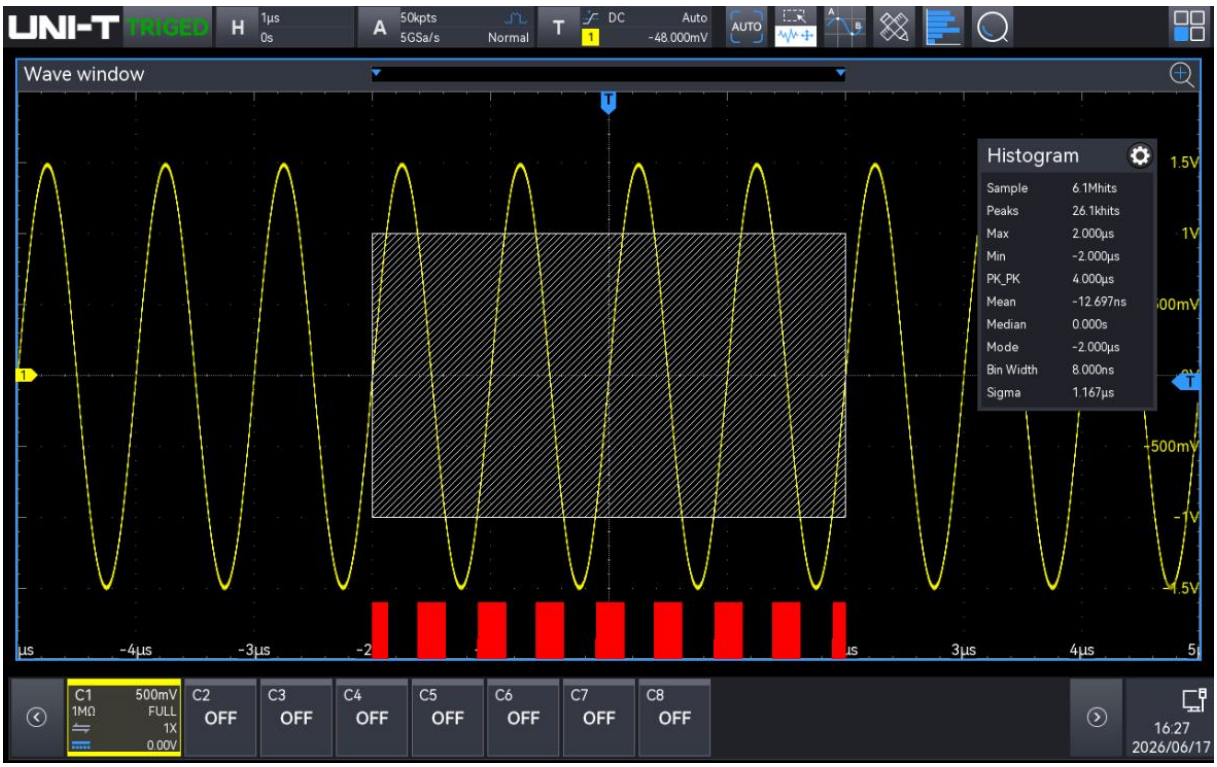
- In the histogram window, select the Left Boundary, Right Boundary, Upper Boundary, or Lower Boundary input field, then use the Multipurpose rotary knob to adjust the boundaries. Boundaries can be set to the screen area, with fine tuning available for precise adjustments.
- Select and drag the four 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 consists of bar graphs on the display. The graph appears on the left side of the grid area for vertical waveform histograms, or at the bottom for horizontal waveform histograms. As waveforms are acquired and displayed, or measurements are performed, the size of the bar graph changes to indicate the peak number of triggers at the specified histogram scale.

(8) Histogram Statistical Results

Histogram data results are displayed in the histogram statistics 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.

27. System Upgrade

27.1. Access Version

1. Navigate to the Utility menu via Home > Utility, 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.

27.2. Upgrade Method

This series supports USB disk upgrade and online firmware upgrade, offering a convenient and flexible upgrade method.

1. USB Disk Upgrade at Power-On

   The oscilloscope automatically detects the upgrade file on a USB flash drive during power-on.
2. USB Disk Upgrade via Menu

   Navigate to Home > Utility > Upgrade, detect the upgrade file on the USB flash drive, and select it to start the upgrade.
3. Online Upgrade

   Upgrade the oscilloscope firmware directly through the online upgrade function.

27.3. Power-On Upgrade

① Insert the USB flash drive when the oscilloscope is powered off. Press the power button. The oscilloscope automatically detects the upgrade file and enters the upgrade interface.

② Select the upgrade version and press Upgrades to start the process.

③ The upgrade process takes approximately 5 minutes. After completion, the oscilloscope restarts automatically. Remove the USB flash drive during the restart process.

④ Power on the oscilloscope and check the system information. If the displayed software version matches the upgrade version, the upgrade was successful.

27.4. 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.

① Upgrade File: Click Upgrade File drop-down list. 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.

② Upgrade: 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.

③ The upgrade process takes approximately 5 minutes. After completion, the oscilloscope restarts automatically. Remove the USB flash drive during the restart process.

④ Power on the oscilloscope and check the system information. If the displayed software version matches the upgrade version, the upgrade is successful.

27.5. Online Upgrade

Perform an online upgrade only after ensuring that the LAN port on the rear panel of the instrument is connected to the network. If network access is restricted, enable the required network permissions accordingly.

Press Online 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.

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.

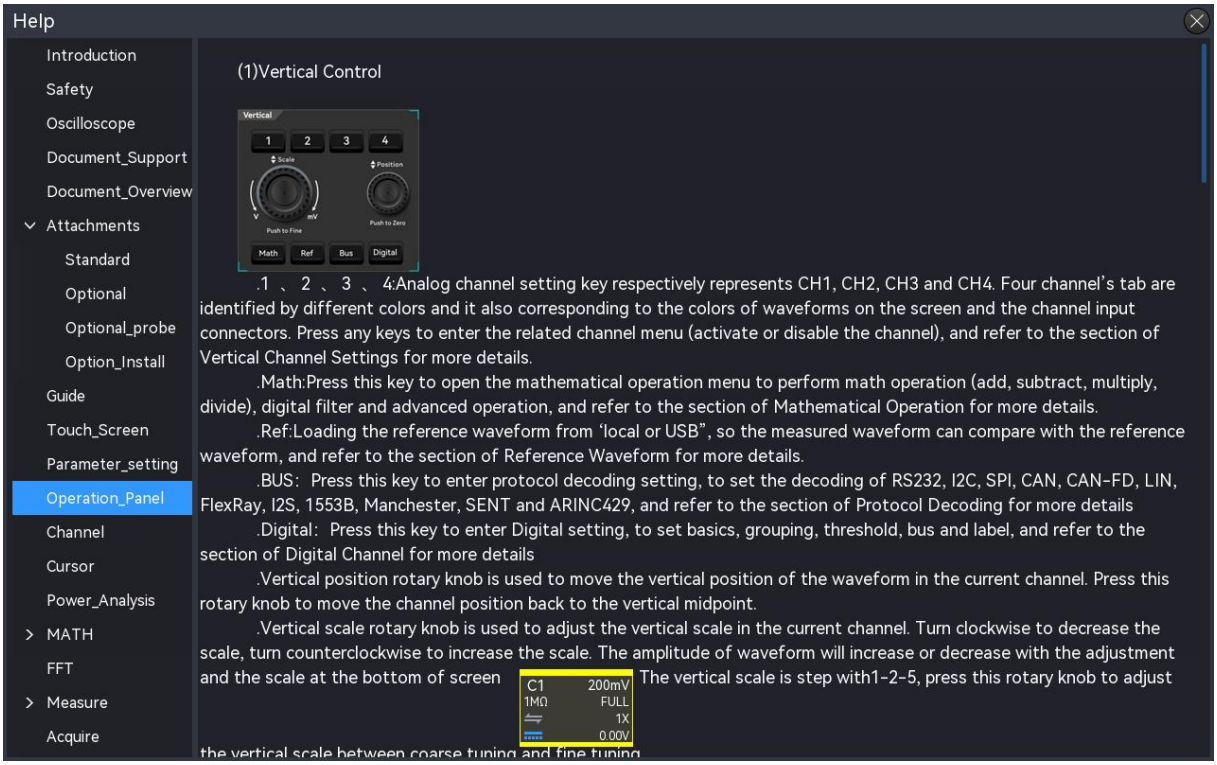
28. Help System

The help system provides information about the function keys (including menu keys) on the front panel.

Access the Help menu using the following methods.

- Click Home icon , select the help icon to open the help menu.
- In each function menu window, click the help icon in the top right to open the relevant help menu.

The help screen is divided into two parts, the left side is Help Options, and the right side is Help Display Area. By selecting a help option, the user can view all related help content on the right.



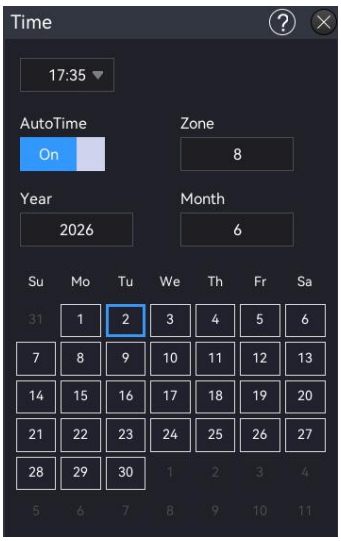
29. Notification Setting

Click notification area at the bottom of the screen to open the time settings and file browser settings.



29.1. 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

Click Zone input field, then use the Multipurpose rotary knob to adjust the time zone. Double-click Zone input field to open the numeric keypad and set the time zone. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. The time zone can be set from -11 to 12.

(3) Year

Click Year input field, then use the Multipurpose rotary knob to adjust the year. Double-click Year input field to open the numeric keypad to enter the year. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. The year can be set from 2000

to 2050.

(4) Month

Click Mont input field, then use the Multipurpose rotary knob to adjust the month. Double-click Month input field to open the numeric keypad to enter the month. For details on the use of the numeric keypad, refer to the section of 5.8 Parameter Setting. The month 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

Click Time to display the two dial wheels for setting the current time. Set the hour (left) and minute (right) values by touching and scrolling the wheels up or down. Click blank area to confirm the time setting.

29.2. 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.

30. Auxiliary Key

- Automatic Setting
- Run/Stop
- Factory Setting

30.1. 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. Press the Autoset button on the front panel enable the automatic setting.

The automatic setting is only suitable under the following conditions.

a. Signals must be simple, single-frequency waveforms. Complex or combined waveforms cannot be effectively adjusted automatically.

b. The measured signal must have a frequency of at least 10 Hz and an amplitude of at least 12 mVpp. For square waves, the duty cycle must be greater than 5%.

30.2. Run/Stop

Use the Run/Stop button on the front panel to set the oscilloscope to Run or Stop state.

- Run: If the green light is on after pressing the key, the oscilloscope is in Run state.
- Stop: If the red light is on after pressing the key, the oscilloscope is in Stop state.

In Run state, the oscilloscope continuously acquires waveforms, and AUTO is displayed at the top of the screen. In Stop state, acquisition stops, and STOP is displayed at the top of the screen.

Press the Run/Stop key to toggle waveform acquisition between Run and Stop states.

30.3. Clear

Press the Clear  button on the front panel to reset the oscilloscope. This clears all reloaded waveforms, and parameter statistics.

30.4. Factory Setting

Navigate to Utility > About and tap System Reset to quickly restore the oscilloscope to its factory default settings. This operation will erase all locally stored files. The factory default state of the UPO6000HD series high-resolution oscilloscope is 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, C3, C4, C5, C6, C7, C8	OFF
	MATH, FFT, REF	OFF
Horizontal System	Extension window	OFF
	Fine tuning	Coarse tuning
	Horizontal extension	Center
	Auto roll mode	Yes
	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
	Memory depth	Auto

	Interpolation	sinc
Display	Display format	Vector
	Grid display	Full display
	Transparence of pop-up window	50%
	Backlight brightness	50%
	Persistence	Auto
	Stop persistence	Not check
	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	Hamming
	FFT unit	dBV
	FFT count	20 k
	Waterfall curve	OFF
Measurement	Parameter snapshoot	OFF
	Parameter measurement	OFF
	Measurement statistical	OFF
	Voltmeter	OFF
	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 output	1 kHz
	Synchronized output	Idle
	IP	DHCP
	Language	Current language
	Cursor type	OFF
	Horizontal cursor	Fixed delay
	Synchronized move	OFF
	Automatically set channel	Autoset
	Automatically set sampling	Autoset
	Automatically set trigger	Autoset
	Automatically set signal	Autoset
	Select channel	C1
	RUN/STOP	RUN

31. System Prompt and Troubleshooting

- System Prompt
- Troubleshooting

31.1. System Prompt

This chapter describes the system prompts, with detailed explanations provided in the table below.

Touch function locked.	This prompt appears when the touchscreen is locked.
Touch function unlocked.	This prompt appears when the touchscreen is unlocked.
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, or decoding list 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 successful.	This prompt appears when the system upgrade is completed successfully.
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 memory depth while opening FFT.
Load successful.	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 templates 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.
This prompt appears when loading a REF file that contains errors.	This prompt appears when opening FFT and entering SCAN or ROLL modes.
Gen channel overloaded; please check the circuit!	This prompt appears when the Gen is overloaded with a power signal.
Analog channel overloaded; please check the circuit!	This prompt appears when analog channels C1 - C8 are overloaded with a power signal.
Maximum length reached!	This prompt appears when the input exceeds the length limit on the alphabetic or numeric keyboard.
10 MHz synchronization clock input not detected!	This prompt appears when the 10 MHz synchronization signal is not connected.
Signal source not opened; please open it and try again!	This prompt appears when the signal source is not opened during waveform saving operations.

USB drive is not inserted!	This prompt appears when the USB drive is not connected for option activation or full activation operations.
The current fundamental wave does not support the selected modulation; switching to a supported basic wave!	This prompt appears when a non-supported modulation basic 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.
The current fundamental wave does not support the selected modulation; switching to a supported fundamental wave!	This prompt appears when a non-supported modulation fundamental wave is selected in continuous wave mode.
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.	Upgrade failed.
Basic wave type error.	This prompt appears when an unsupported basic wave type is selected for amplitude modulation, frequency modulation, amplitude shift keying, frequency shift keying, or sweeping.
Loading failed.	This prompt appears when loading a template 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 box.
Please pause before saving the waveform in ROLL/SCAN mode!	This prompt appears when attempting to save a waveform in ROLL/SCAN mode.
Already the latest version!	This prompt appears when attempting to upgrade while already on 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.
Mouse connected	This prompt appears when a mouse is connected.
Mouse disconnected	This prompt appears when the mouse is removed.
Function disabled!	This prompt appears when enabling XY mode during average sampling, or tapping Single in ROLL mode.
No signal triggered!	This prompt appears when no signal is triggered during inrush current or power on/off tests.
Invalid file!	This prompt appears when importing an incorrect file for SOA analysis.
File does not exist.	This prompt appears when loading a deleted setup file.
Path does not exist.	This prompt appears when the selected save path no longer exists.
Save failed. Check if template points form a closed area.	This prompt appears when the edited template does not form a closed area during SOA analysis.
Timing analysis unavailable while FFT is enabled!	This prompt appears when attempting to enable timing analysis while FFT is active.
Memory depth limited for timing analysis!	This prompt appears when memory depth is set to 50 M or higher with timing analysis enabled.

31.2. Troubleshooting

(1) If the oscilloscope remains on a black screen without any display when pressing the soft power button.

a. Check if the power plug is properly connected and the power supply is normal.

b. Check if the power switch is turned on. If the power switch is turned on, the soft power button on the front panel should be green. When the soft power button is enabled, the soft power button should be blue, and the oscilloscope will make an active sound. There should be a normal relay rattle when the soft power button is pressed.

c. If the relay has sound, it indicates that the oscilloscope is normal boot-up. Press the Default key and press the Yes key, if the oscilloscope returns to normal, indicating that the backlight brightness is set too low.

d. Restart the oscilloscope after completing the above steps.

e. 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 probe and DUT are connected properly.

b. Check whether the signal output channel is open.

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 whether the channel's probe attenuation coefficient setting matches the attenuation rate of the connected probe.

(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 be displayed stably only when the trigger mode is correctly set.

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 Autoset 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.

32. Appendix

32.1. 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. Please disconnect the power supply, then wipe the instrument with a damp but not dripping soft cloth. Do not use any abrasive chemical cleaning agent 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.

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.

UNI-T is the licensed trademark of UNI-TREND TECHNOLOGY (CHINA) CO., Ltd.
UNI-T products are protected under patent laws in China and internationally, covering both granted and pending patents. Licensed software products are the properties of UNI-Trend and its subsidiaries or suppliers, all rights reserved. This manual contains information that replaces all earlier published versions. The product information in this document subject to update without notice. For more information on UNI-T Test & Measure Instrument products, applications, or service, please contact UNI-T instrument for support, the support center is available on www.uni-trend.com ->instruments.uni-trend.com
https://instruments.uni-trend.com/ContactForm/

Headquarter

UNI-TREND TECHNOLOGY (CHINA) CO., Ltd.
Address: No.6, Industrial North 1st Road, Songshan Lake Park, Dongguan City, Guangdong Province, China
Tel: (86-769) 8572 3888

Europe

UNI-TREND TECHNOLOGY EU GmbH
Address: Steinerne Furt 62, 86167 Augsburg, Germany
Tel: +49 (0)821 8879980

North America

UNI-TREND TECHNOLOGY US INC.
Address: 2692 Gravel Drive, Building 5, Fort Worth, Texas 76118
Tel: +1-888-668-8648

Copyright © 2026 by UNI-Trend Technology (China) Co., Ltd. All rights reserved.