Hantek

DPO8000 series

Digital oscilloscope

User Manual

202410

Warranties and Declarations

Copyright

Statement

Qingdao Hantek Electronic Co., Ltd. reserves the right to amend this document without prior notice. Qingdao Hantek Electronic Co., Ltd. promises that the information provided is correct and reliable but does not guarantee that this document is free from errors. Before using this product, please make sure that the specifications of relevant technical documents are the latest effective version. If you use documents or products of Qingdao Hantek Electronic Co., LTD and need products, patents or works of third parties to cooperate with them, you shall be responsible for obtaining the consent and authorization of the third parties. The above consent and authorization shall not be the liability of Hantek.

Product certification

Hantek certified DPO series oscilloscope to meet China's national industry standards and has passed the CE certification.

Contact us

If you have any questions when using the products of Qingdao Hantek Electronic Co., LTD., you can obtain service and support through the following ways:
Email: service@hantek.com, support@hantek.com
Website: http://www.hantek.com

Contents

Figures

Tables

1 Safety requirement

1.1 Summary of general security issues

Read the following safety precautions carefully to avoid injury and to prevent damage to this product or any product connected. To avoid possible dangers, please use this product in accordance with the regulations.

- **Only professionally authorized personnel can perform repairs.**
- **Use the right power cable.**

  Use the power cable approved by the country in which the product is used only.
- **Connect and disconnect correctly.**

  Before connecting the probe to the circuit being measured, please connect the probe to the oscilloscope first. Before disconnecting the probe from the oscilloscope, please disconnect the probe and the circuit under test first.
- **Ground the product.**

  To avoid electric shocks, the product is grounded through a grounding conductor of the power cable. The grounding conductor must be connected to the ground before connecting the input or output terminals of the product. Ensure that the product is properly grounded.
- **Connect the probe properly.**

  The ground wire of the probe is the same as the ground potential. Do not connect the ground wire to high voltage.
- **View all terminal rating values.**

  To avoid fire or excessive current, please check all rating values and signs on the product. Please consult the product manual for details of the rating values before connecting the product.
- **Do not operate with the cover open.**

  Do not run the product with the cover or panel open.
- **Avoid circuit exposure.**

  Do not touch exposed connectors and components after power is switched on.
- **Do not operate if the product is suspected to be faulty.**

  If you suspect that the product has been damaged, please ask qualified maintenance personnel to check it.

- **Maintain proper ventilation.**
- **Do not operate in a humid environment.**
- **Do not operate in inflammable or explosive environment.**
- **Please keep the product surface clean and dry.**

Warning:

Equipment that meets Class A requirements may not provide adequate protection for broadcast services in residential environments.

1.2 Security terms and signs

Security terms in this manual:

Warning:

Indicates that the operation may not cause immediate damage to you.

Note:

Indicates that the operation may cause damage to the product or other property.

Safety terms on products:

Warning:

Indicates a potential hazard may be caused to you if you do not perform this operation.

Safety signs on the product:

Hazardous Voltage	Safety Warning
	

1.3 Measurement category

Measurement category

This instrument can be used for measurement under class I.

Warning:

This instrument is only allowed to be used in the specified measurement class.

Measurement class definition

- **Class I refers to measurements taken on a circuit not directly connected to the main power supply.** For example, measurements made on circuits that are not exported from a main power supply, especially from a protected (internal) main power supply. In the latter case, the instantaneous stress will change. Therefore, the user should understand the instantaneous capacity of the instrument.
- **Class II refers to measurements taken on a circuit directly connected to low-voltage instruments.** For example, measurements made on household appliances, portable tools, and similar equipment.
- **Class III refers to measurements taken on construction equipment.** For example, measurements made on switchboards, circuit breakers, circuits (including cables, busbars, junction boxes, switches, sockets) in fixed equipment, as well as equipment for industrial use and certain other equipment (for example, fixed motors permanently connected to fixed instruments).
- **Class IV refers to measurements taken at the source of low-voltage equipment.** For example, measurements made on electricity meters, primary overcurrent protection equipment, and pulse control units.

1.4 Ventilation Requirement

This oscilloscope uses a fan to force cooling. Please make sure that the air intake and exhaust areas are free from obstructions and have free air. When using the oscilloscope in a bench-top or rack setting, provide at least 10 cm clearance beside, above and behind the instrument for adequate ventilation.

Note:

Inadequate ventilation may cause an increase of temperature in the instrument, which would cause damage to the instrument. So please keep the instrument well ventilated and inspect the air outlet and the fan regularly.

1.5 Working Environment

Temperature

Operating: 0°C to 50°C
Non-operating: -30°C to 70°C

Humidity

- Operating:
  Below +30°C: ≤95%RH (without condensation)
  +30°C to +40°C: ≤75%RH (without condensation)
  +40°C to +50°C: ≤45%RH (without condensation)
- Non-operating:
  Below 65°C: ≤95% RH (without condensation)

Warning:

To avoid short circuit or electric shock, do not operate the device in a damp environment.

Altitude

Operating: below 3 km
Non-operating: below 15 km

Installation (Overvoltage) Category

This product is powered by mains conforming to installation (overvoltage) category II.

Warning:

Ensure that no overvoltage (e.g. from lightning) reaches the product. Otherwise, the operator may be in danger of receiving electric shock.

Installation (Overvoltage) Category Definitions

Installation (overvoltage) category I refers to signal level which is applicable to equipment measurement terminals connected to the source circuit. Among these terminals, precautions are done to limit the transient voltage to a low level.
Installation (overvoltage) category II refers to the local power distribution level which is applicable to equipment connected to the AC line (AC power).

Pollution Degree

Pollution Degree 2

Pollution Degree Definition

- **Pollution Degree 1:** No pollution or only dry, nonconductive pollution occurs. The pollution has no effect. For example, a clean room or air-conditioned office environment.
- **Pollution Degree 2:** Normally only nonconductive pollution occurs. Temporary conductivity caused by condensation is to be expected. For example, indoor environment.
- **Pollution Degree 3:** Conductive pollution or dry nonconductive pollution that becomes conductive due to condensation occurs. To be found in industrial environment or construction sites (harsh environments). For example, sheltered outdoor environment.
- **Pollution Degree 4:** The pollution generates persistent conductivity caused by

conductive dust, rain, or snow. For example, outdoor areas.

Security level

Class 1 - Grounded products

1.6 Care and Cleaning

Care:

Do not expose the LCD to direct sunlight for a long time when storing or placing the oscilloscope.

Cleaning:

If the oscilloscope and probe are inspected frequently as required by operating conditions, clean the outer surface of the instrument by following the following steps:
1) Use a lint - free cloth to remove the dust outside the oscilloscope and probe. Please be careful to avoid scratching the smooth display filter material.
2) Clean the oscilloscope with a soft cloth soaked in water. For a more thorough cleaning, use a aqueous solution of 75% isopropyl alcohol.

Note:

In order to avoid damaging the surface of oscilloscope or probe, do not use any corrosive reagent or chemical cleaning reagent.

Warning:

Before powering on the device again, ensure that the device is dry enough to avoid electrical short circuit or personal injury caused by moisture.

1.7 Environmental Considerations

The following symbols indicate that the product complies with the requirements of WEEE Directive 2002/96/EC.

Equipment recovery:

Producing the device requires the extraction and use of natural resources. Some substances contained in the equipment may be harmful to the environment or human health if the product is not disposed of properly. In order to avoid the release of harmful substances into the environment and reduce the use of natural resources, it is

recommended that appropriate methods be used to recycle this product to ensure that most of the materials can be correctly reused.

2 Product features

Product features

- Set 8 independent instruments in one, including: oscilloscope /16-channel logic analyzer/spectrum analyzer/arbitrary wave generator/digital voltmeter / 6-bit frequency meter and accumulator/protocol analyzer/protocol generator;
- Real-time sampling rate up to 8GSa/s, 2G memory depth, hardware real-time waveform recording and playback up to 2 million frames;
- UART, I2C, SPI, LIN, CAN protocol generation function;
- Built in two 200MHz signal sources, with a vertical resolution of 12 bits and a maximum output frequency of 200MHz;
- 10.1-inch multi touch capacitive screen, 256 level waveform grayscale and color temperature display;
- The waveform capture rate is higher than 600000 waveforms per second;
- Rich serial protocol triggering and decoding functions;
- Up to 52 waveform parameters can be automatically measured, and full memory hardware measurement function is also provided;
- Multiple data analysis and processing functions: independent search, navigation buttons and event lists, histograms, Bode plots, power analysis, counters;
- The entire series comes standard with LAN and USB remote communication functions, and supports optional RS232 and HDMI interfaces;
- Wide range, low background noise, vertical sensitivity range: 500 μ V/div~10 V/div, all levels support full bandwidth;
- Li Shayu waveform supports dual screen display and cursor measurement in XY mode.

10.1 inch multi-touch capacitive screen, 256 level waveform gray scale and color temperature display,1GHz bandwidth,8GSa/s sampling rate, 2G memory depth, 600.000wfms/s waveform capture rate; Built in two 200MHz waveform generator, supporting arbitrary wave output; 52 kinds of automatic measurement; Rich serial protocol triggering and decoding functions; Multiple data analysis and processing functions; Integrate the functions of 8 instruments, significantly simplify the measurement system and speed up the measurement; Rich configuration interfaces are provided for more convenient use; It is a reliable oscilloscope that can provide you with professional measurement.

3 Document overview

This document describes how to quickly understand the front and back panels, user interfaces, and basic operation methods of the DPO8000 series digital oscilloscopes.

Tip:

The latest version of this manual can be downloaded at (http://www.hantek.com).

Document number:

202410

Software version:

Software upgrade may change or increase product functionalities, please pay attention to Hantek website for the latest version.

Document format conventions:

1 Virtual keys and main interface icons

Use **[name]** to represent virtual keys and main interface icons. For example, **[Utility]** is for 

.

2 Menu

Use "menu text (bold) + color" to represent a label or a menu option. For example, **I/O** means to click the "I/O" option on the current operation interface to enter the function configuration menu of "I/O".

3 Operation steps

Use "->" to represent the next step. For example, **[Utility]** -> **Language** means click **Utility** label before clicking **Language** menu.

Document content conventions:

DPO8000 series tablet oscilloscope consists of the following models. Unless otherwise specified, this manual uses DPO8054E as an example to describe the DPO8000 series and basic operations.

Model	Bandwidth	Channel	Signal source	Digital Channels
DPO8104E	1GHz	4	2	16
DPO8084E	800MHz	4	2	16
DPO8054E	500MHz	4	2	16

Model	Bandwidth	Channel	Signal source	Digital Channels
DPO8034E	350MHz	4	2	16
DPO8024E	200MHz	4	2	16
DPO8104C	1GHz	4	-	-
DPO8084C	800MHz	4	-	-
DPO8054C	500MHz	4	-	-
DPO8034C	350MHz	4	-	-
DPO8024C	200MHz	4	-	-
DPO8054U	500MHz	4	2	16

Table 3.1 Model

4 Quick start

4.1 General examination

Check the shipping package

After receiving the oscilloscope, please follow the following steps to check the instrument: Check whether there is any damage caused by transportation: If the packaging cartons or protective foam pads are seriously damaged, please keep them until the whole machine and accessories pass the electrical and mechanical testing.

Check the accessories

The details of the accessories are provided in Appendix A: Accessories at the end of the user manual. If you find any accessory missing or damaged, please contact the dealer responsible for this business.

Check the machine

If you find the instrument is damaged, not working properly, or unable to pass the performance test, please contact the dealer responsible for this business.

4.2 Appearance and dimension

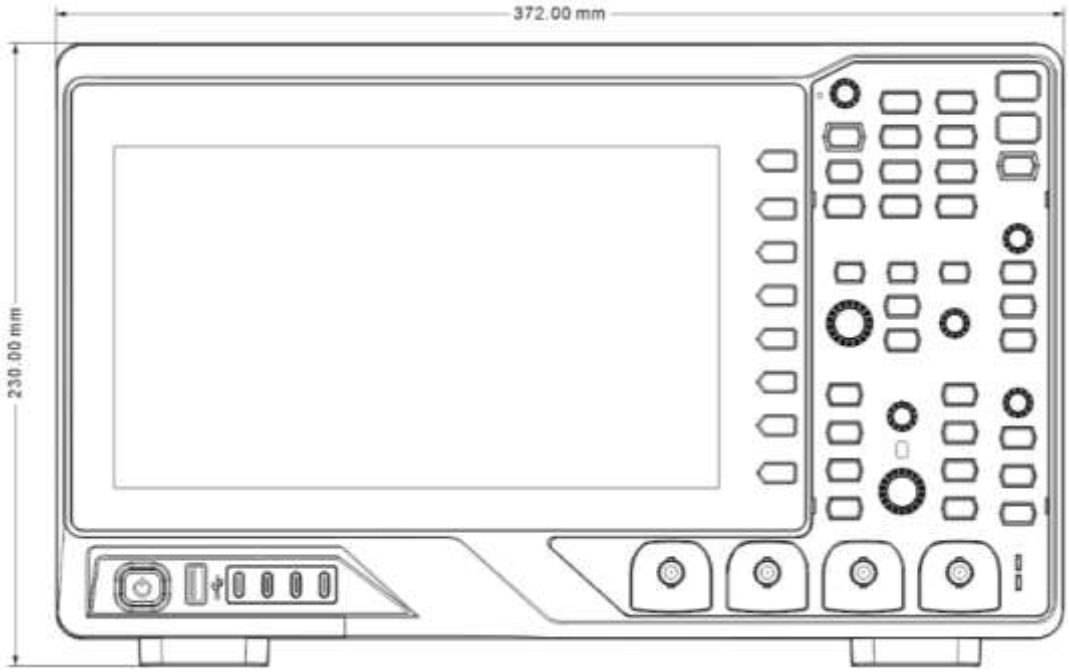


Figure 4.1 Front view

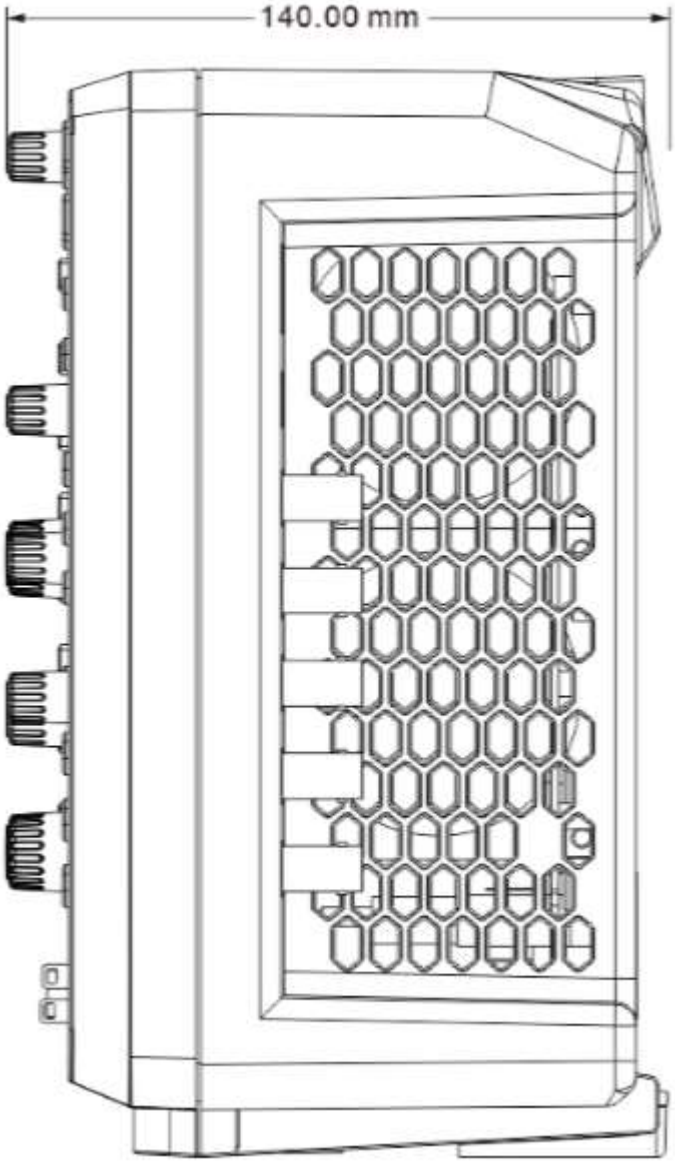


Figure 4.2 Side view

4.3 Preparation before use

1. To Adjust the Supporting Legs

There is a bracket design at the bottom of the machine, and the bracket should be adjusted appropriately to tilt the oscilloscope slightly upwards, which can better stabilize the placement of the oscilloscope and facilitate observation and operation. When not using the machine, simply close the bracket.

2. To Connect to AC Power

The specifications of the AC power supply that this oscilloscope can input are: 100-

120V, 50/60/400Hz; 100-240V, 50/60Hz. Please connect the oscilloscope to the power supply using the power cord provided in the attachment, as shown in the figure.

Figure 4.3 To Connect to AC Power

Warning:

To avoid electric shock, ensure that the instrument is correctly grounded.

3. Turn-on Checkout

When the machine is properly plugged in and the oscilloscope is powered on, press the power button in the lower left corner of the front panel to start the oscilloscope. During the startup process, all the button lights on the front panel will light up for a few seconds, and the oscilloscope will perform a series of self checks. After the self check is completed, the startup screen will appear.

4. Function Inspection

- Click on **[Default Setup]** in the button area to restore the oscilloscope to its factory settings.
- Connect the grounding crocodile clamp wire of the probe to the grounding terminal in the figure below.
- Connect the probe to the input terminal of channel 1 and the compensation signal output terminal marked in the figure below.

Figure 4.4 To Use the Compensation Signal

- Set the probe attenuation ratio to X10 and click on the button area **[Auto Scale]**.
- Observe the pictures on the waveform to see if the square wave signal is displayed normally. As shown in the following figure.

Figure 4.5 Square Waveform Signal

- Check CH2~CH4 channels using the same method. If the square wave waveform displayed in the time base does not match the figure above, you can follow the instructions in the section on Probe Compensation.

Warning:

To avoid electric shock when using the probe, please make sure that the insulated wire of the probe is in good condition. Do not touch the metallic part of the probe when the probe is connected to high voltage source.

Tip:

The probe compensation signal can only be used for probe compensation

adjustment and cannot be used for calibration.

5. Probe Compensation

When connecting the probe to any input channel for the first time, this adjustment is required to match the probe with the input channel. Uncompensated or offset probes can lead to measurement errors or errors.

- Perform the first four steps of the previous section on "Function Check".
- Compare the waveform with the figure below.

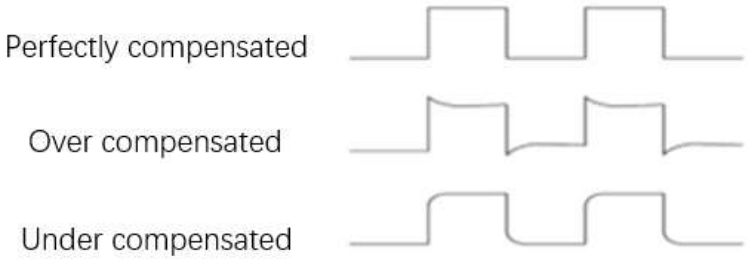


Figure 4.6 Probe Compensation

- If necessary, use a non-metallic screwdriver to adjust the variable capacitance on the probe until the waveform displayed on the screen is "Perfectly compensated" as shown in the figure above. If necessary, repeat this step. The adjustment method is shown in the following figure.

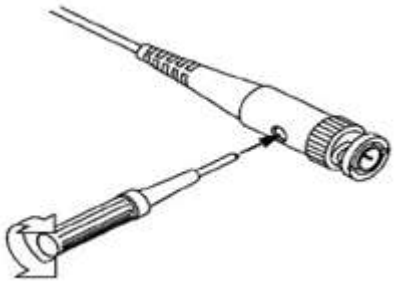


Figure 4.7 Adjusting capacitance

4.4 Front Panel Overview

Figure 4.8 Front Panel

Table 4.1 Front Panel Description

No.	Description
1	Touch Screen
2	Function Menu Operation Keys
3	Multifunction Knob
4	Default Setting Key
5	Touch Lock Key
6	Common Operation Keys
7	Horizontal Control System
8	Vertical Control System

9	Run/Stop Key
10	Single Trigger Control Key
11	Auto Waveform Display Key
12	Arbitrary Waveform Generator Setting Key
13	Trigger Control System
14	Probe Compensation Signal Output Terminal/Ground Terminal
15	Analog Channel Input Terminals
16	Digital Channel Input Interface
17	USB HOST Interface
18	Power Key

4.5 Front Panel Function Overview

1. **Horizontal:**

- **Search:**

  Press this key to enter the search settings menu, and the search function allows users to search for related events in the collected data by setting search criteria.

- **Zoom:**

Pressing this key can turn on or off the delayed scanning function.

-  :

  Indicates the horizontal time base knob. You can rotate the knob to modify the horizontal time base. Turn it clockwise to decrease the time base, and turn it counterclockwise to increase the time base. During the modification, waveforms of all the channels will be displayed in expanded or compressed form, and the time base message at the upper section of the screen would change accordingly. Press down this knob to quickly switch the horizontal time base adjustment mode between "Coarse" and "Fine".

-  :

  Indicates the horizontal position knob. You can rotate the knob to modify the horizontal position (i.e. trigger position). The trigger point would move left or right relative to the center of the screen when you rotate the knob. During the modification, waveforms of all the channels would move left or right, and the horizontal position message at the upper-right corner of the screen would change accordingly. Press down this knob to quickly reset the horizontal position (or the delayed sweep position).

- **Navigation Control Key:**

  This combination key can perform recording and playback navigation, time navigation, and event navigation.

- **Horizontal time base knob:**

  This knob is used to modify the horizontal time base. Rotate clockwise to decrease the time base, and counterclockwise to increase it. During the modification process, the waveforms of all channels are expanded or compressed for display, while the time base information above the screen changes in real-time. Press this knob to quickly switch the horizontal time base adjustment mode to "coarse adjustment" or "fine adjustment".

- **Horizontal displacement knob:**

  This knob is used to modify the horizontal displacement. Rotate the knob to move the trigger point left and right relative to the center of the screen. During the modification process, the waveforms of all channels move left and right, while the horizontal displacement information in the upper right corner of the screen changes in real-time. Press this knob to quickly reset the horizontal displacement.

2. **Vertical:**

- **CH1, CH2, CH3, and CH4 analog input channel switches:**

  The four channels are identified with different colors, and the color of the waveform also corresponds to the color of the channel.

- **Channel vertical offset knob:**

  Rotate this knob to modify the vertical offset of the current channel waveform. Rotate clockwise to increase the offset, and counterclockwise to decrease the offset. During the modification process, the waveform will move up and down, and the offset information in the corresponding status labels will change in real-time. Press this knob to quickly reset the vertical offset to zero.

- **Channel vertical gear knob:**

  Modify the vertical gear of the current channel. Rotate clockwise to decrease the gear, and counterclockwise to increase the gear. During the modification process, the waveform display amplitude will increase or decrease, and the corresponding gear information in the status label will change in real time. Press this knob to quickly switch the vertical gear adjustment mode to "coarse adjustment" or "fine adjustment".

3. **Waveform control system:**

- **REF:**

  Reference waveform key. Press this key to open the reference waveform setting menu.

- **Math:**

  Mathematical operation keys. Press this button to open the mathematical operation function menu, where you can perform A+B, A-B, A×B, A/B and FFT.

- **LA:**

  Logic analyzer key. Press this key to open the control menu of Logic analyzer.

- **Decode:**

  Decode key. Press this key to open the decoding settings menu.

4. **Wave Gen:**

- **G1:**

  Press the G1 button to open the signal generator channel of G1.

- **Burst:**

  Burst button.

- **Signal source knob:**

  Modify the frequency and amplitude of the signal source. Rotate clockwise to increase the value, and counterclockwise to decrease the value. Press this knob to select the adjustable frequency or amplitude.

5. **Trigger:**

- **Trigger knob:**

  Modify the trigger level or threshold level. Rotate clockwise to increase the level, and counterclockwise to decrease the level. During the modification process, the trigger level moves up and down, while the trigger level/threshold level value in the upper right corner of the screen changes in real-time. Press this knob to quickly set the trigger level value to 50% of the waveform peak value.

- **Trig Menu:**

  Press the button to open the trigger operation menu.

- **Force Trig:**

  Pressing this key will force the oscilloscope to generate a trigger signal.

- **Mode:**

  Press this key to switch the trigger mode to Auto, Normal, or Single.

6. **AUTO:**

Press this button to enable the automatic waveform setting function. The oscilloscope will automatically adjust the vertical gear, horizontal time base, and triggering method based on the input signal to achieve the optimal waveform display.

7. **Run/Stop:**

Press this key to set the operating status of the oscilloscope to "Running" or "Stopping". In the RUN state, the green backlight of this key lights up; In the STOP state, the red backlight of this key lights up.

8. **Single:**

Press this key to set the triggering method of the oscilloscope to Single.

9. **Multifunction Knob:**

Modify the setting parameters by rotating the knob.

10. **Default:**

Press this key to restore the oscilloscope to its factory default settings.

11. **Touch Lock Key:**

Press this key to disable the touch screen function.

12. **Function Menu:**

- **Quick Action:**

  The quick operations that can be performed by pressing this key include screenshots, waveform saving, setting saving, all measurements, and statistical resetting.

- **Analyze:**

  Press this key to enter the analysis menu.

- **Cursors:**

  Press this key to enter the cursor measurement menu. The oscilloscope provides two cursor modes: manual and tracking.

- **Save/Recall:**

  This key can perform the save recall function.

- **Utility:**

  Press this key to enter the system function settings menu. Set system related functions or parameters, such as interfaces, sound, and language. In addition, it also supports some advanced functions, such as testing, waveform recording, and self calibration.

- **Help:**

  Help provides instructions for the various function buttons and corresponding menu keys on the front panel.

- **Meas:**

  Press this key to enter the measurement settings menu.

- **Acquire:**

  Press this key to enter the sampling settings menu. The time base mode, acquisition method, and storage depth of the oscilloscope can be set.

- **Display:**

  Press this key to enter the display settings menu. The display type, afterglow time, and waveform brightness can be set for the waveform.

4.6 Rear Panel Overview

Figure 4.9 Rear Panel

Table 4.2 Rear Panel Description

No.	Description
1	Handle
2	LAN Interface
3	USB DEVICE Interface
4	Reference clock
5	EXT input terminal

6	G1 signal source output terminal
7	Trigger output and pass/fail
8	HDMI
9	RS232/485
10	AC Power Cord Connector

4.7 Rear Panel Overview

1. **Handle:**

   Users can vertically pull up the handle for easy access to the oscilloscope; When not in use, press down on the handle.

2. **LAN interface:**

   This interface is used to connect the oscilloscope to the network.

3. **USB VIEW interface:**

   This interface is used to connect the oscilloscope to the computer, and users can send SCPI commands or customize programming to control the oscilloscope through the upper computer software.

4. **Reference clock:**

   The use of a reference clock can provide a more accurate sampling clock signal for the oscilloscope, and can also be used to synchronize the clocks of two or more oscilloscopes.

5. **EXT input:**

   This interface is used to input external trigger signals to the oscilloscope.

6. **G1 signal source output terminal:**

   This interface outputs the waveform set by the signal generator.

7. **Trigger output and pass/fail:**

   Trigger output:

   When the oscilloscope generates a trigger, a signal reflecting the current capture rate of the oscilloscope can be output through this interface. The signal can be connected to the waveform display device to measure the frequency of the signal,

and the measurement result is the same as the current capture rate.

Pass/fail:

In pass/fail testing, when a pass or fail event is detected, a pulse will be output from the **[TRIG OUT]** connector on the rear panel.

8. **HDMI interface (option):**

   This interface is used to connect the oscilloscope to an external display with an HDMI interface (such as a monitor or projector), allowing for clearer waveform display. At this point, the display screen of the oscilloscope is still valid.

Note:

When using an external display screen with an oscilloscope HDMI interface, the power supply of the external display screen needs to be grounded, otherwise it will burn out the oscilloscope's HDMI module.

Suggestion: Connect the monitor to the HMDI interface before powering on the device.

9. **RS-232/485 interface (option):**

   This interface is the serial communication interface of the oscilloscope(Cross serial cable).

10. **Power socket:**

    Power input terminal. Please connect the oscilloscope to the power supply using the power cord provided in the attachment.

4.8 User Interface

Figure 4.10 User Interface

1. **Model and time:**

   Display the specific model and time of the machine.

2. **Hantek logo, function navigation:**

   Click on the icon using the touch screen function to open the function navigation.

3. **Running status:**

   The operating status of the oscilloscope includes: RUN, STOP, TD, WAIT, and AUTO.

4. **Horizontal time base:**

   Represents the length of time represented by each grid on the horizontal axis of the screen. The horizontal knob can be used to modify this parameter.

5. **Horizontal displacement:**

   Adjust the parameters through the horizontal knob to display the specific horizontal position value.

6. **Sampling rate/storage depth:**

   Display the current sampling rate and storage depth of the analog channel. The sampling rate and storage depth will change with changes in the horizontal time base.

7. **Horizontal position sign:**

   Display the current horizontal position.

8. **Waveform memory:**

   Provide a schematic diagram of the position of the waveform in the current screen in the memory.

9. **Trigger location:**

   Display waveform memory and trigger settings for waveforms on the screen.

10. **Trigger source:**

    Display the current trigger source.

11. **Trigger type:**

    Display the currently selected trigger type and trigger condition settings.

12. **Trigger level/threshold level:**

    The trigger source selection is CH1-CH4, and an appropriate trigger level needs to be set. Select D0-D15 as the trigger source and set an appropriate threshold level. The trigger level or trigger threshold will be displayed in the upper right corner.

13. **Operation menu:**

    Press the menu button in the button area to open the corresponding menu.

14. **Notification area:**

    Display icons for time, sound, USB drive, and LAN interface connections.

15. **Arbitrary wave generator (option):**

    Display the open status of any wave generator.

16. **Arbitrary wave generator (option):**

    Display the open status of any wave generator

17. **Digital channel status area:**

    Display the current status of the digital channel.

18. **Message box:**

Display prompt messages.

19. **CH4 status label:**

    Display the status of CH4. Display the disposal gear and offset information of CH4. Display the current channel settings: channel coupling and bandwidth limitation.

20. **CH3 status label:**

    Display the status of CH3. Display the disposal gear and offset information of CH3. Display the current channel settings: channel coupling and bandwidth limitation.

21. **CH2 status label:**

    Display the status of CH2. Display the disposal gear and offset information of CH2. Display the current channel settings: channel coupling and bandwidth limitation.

22. **Analog channel labels/waveforms:**

    Different channels are identified with different colors, and the channel labels and waveform colors are consistent.

23. **CH1 status label:**

    Display the status of CH1. Display the disposal gear and offset information of CH1. Display the current channel settings: channel coupling and bandwidth limitation.

4.9 Touch Screen Controls

This series of oscilloscopes provides a 10.1 inch super capacitive touch screen, which supports multi-touch and gesture operation with high sensitivity. The functions supported by the touch screen include touch, drag, pinch, and rectangle drawing.

Tip:

The menus displayed on the oscilloscope screen and the enabled label buttons on the screen can both use the touch screen function.

1. **Tap:**

Tap a menu or function on the screen with one finger, and the functions that can be achieved by touching include:

- Touch the menu displayed on the screen to operate the menu.
- Touch the pop-up numeric keyboard to set parameters.
- Touch the virtual keyboard to set the tag name and file name.

- Touch the close button in the upper right corner of the pop-up menu to close the pop-up box.
- Touch other windows displayed on the screen to operate them.

Figure 4.11 Tap Gesture

2. Drag:

After holding down the target, drag to the desired position. The functions that can be achieved by dragging include:

- Drag the waveform to change its displacement or offset.
- Drag the window to change its position.

Figure 4.12 Drag Gesture

3. Pinch & Stretch:

Bring two fingers together or apart. Pinching gestures can amplify or reduce the waveform. Enlarge the waveform and slide the close fingers apart; Reduce the waveform and slide the separated fingers closer. As shown in the following figure. The functions that can be achieved by kneading include:

- Vertical direction kneading can adjust the vertical gear of the waveform.
- Horizontal direction pinch can adjust the horizontal gear of the waveform.

Figure 4.13 Pinch & Stretch Gesture

4. Rectangle Drawing:

Click on **[Hantek]**, then click on the **AreaSel** to switch to rectangular drawing mode. Drag your finger on the screen to draw a rectangle,. Move your finger away from the screen, and a menu will appear on the screen. At this time, you can touch to select the functions of "Histogram", "Enlarge Level", "Enlarge Vertical", or "Enlarge Waveform".

- Select "Histogram":
  - ➢ Draw the range of the histogram;
  - ➢ Open the 'Histogram' menu.
- Select "Enlarge Level ": horizontally expand the waveform; Select 'Horizontal Zoom Out': compress the waveform horizontally.
- Select "Enlarge Vertical ": vertically expand the waveform; Select 'Vertical Zoom Out': Compress the waveform vertically.
- Select "Enlarge Waveform": expand the waveform in both horizontal and vertical directions simultaneously; Select 'Waveform Reduction': compress the waveform in both the horizontal and vertical directions simultaneously.

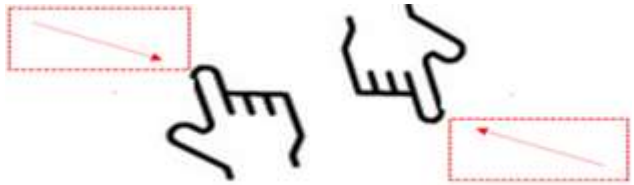


Figure 4.14 Rectangle Drawing Gesture

4.10 Parameter Setting Method

The parameter setting method of this series of oscilloscopes supports two methods: numeric keyboard input and multi-functional knob setting of values.

1. For parameters on the menu, rotate the multi-functional knob V0 in the button area to select parameter items or modify parameter values.
2. After clicking on the parameters on the menu, a numeric keyboard will pop up on the screen, as shown in the following figure.

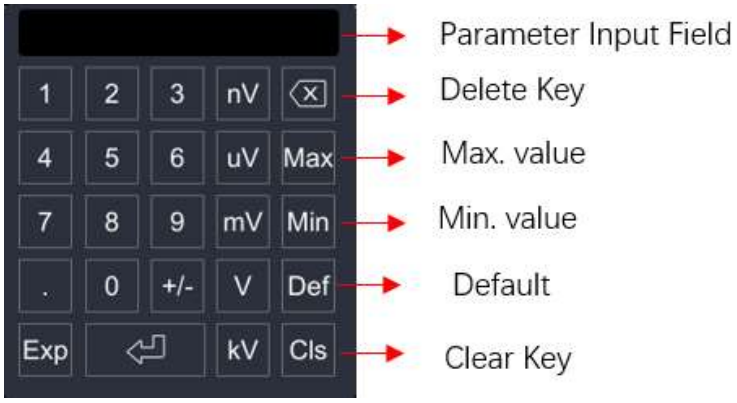


Figure 4.15 Numeric Keypad

In the numeric keypad, you can use the touch screen function to enter values or units by clicking on them. After entering all numerical values and selecting the desired units, the numeric keypad will automatically close to complete the parameter settings. In addition, after completing the numerical input, you can also directly click the  in the numeric keyboard to close it, and the units of the parameters are the default units. In the numeric keypad, you can also perform the following operations:

- Delete the entered parameter values.
- Set the parameters to maximum or minimum values.
- Set the parameters to default values.
- Clear the parameter input box.

3. For parameters on the menu, click the menu label button directly to switch settings.

4.11 To Use the Built-in Help System

To Use the Built-in Help System

The help system of this oscilloscope provides instructions for various function buttons on the front panel and corresponding menu keys. The operation steps are:

- Click the 

button in the button area to enter the help module;
- Press other keys to obtain help information;
- Press the 

button again to exit the help module.

For example:

After clicking the **[Help]** button, click the **[QuickAction]** button again, and the explanation menu for **[QuickAction]** will pop up on the screen.

5 To Set the Vertical System

This series of oscilloscopes provides four input channels CH1-CH4, each with an independent vertical control system. The control systems for the four channels are the same, and this chapter mainly introduces various setting methods for CH1. Please read this chapter carefully to understand the setup functions and operation of the vertical system of the tablet oscilloscope.

Contents in this chapter:

5.1 To Enable or Disable the Analog Channel

1. Enable analog channel

For example, after connecting a signal to the CH1 channel, the channel can be opened as follows.

- Click on the channel status label at the bottom of the screen, open the channel, and the label will light up. At the same time, the corresponding button in the button area **[CH1]** will also light up, and the color will be consistent.
- Press the **[CH1]** button directly to open the channel, and the light on the corresponding channel knob will also light up.

If CH1 is turned on and currently selected, the label is shown in the following figure.

If CH1 is turned on but not currently selected, the label is shown in the following figure.

The information displayed in the channel status label is related to the current channel settings (independent of the channel's on/off status).

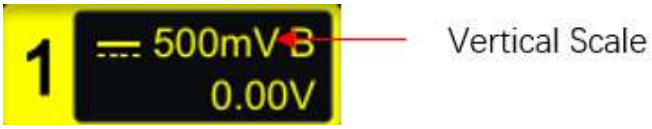
2. Disable analog channel

The simulation channel can be closed through the following methods:

- If the CH1 channel is already open and currently selected, press **[CH1]** once to close the CH1 channel.
- If the CH1 channel is open but not in the selected state, press **[CH1]** once to make CH1 in the selected state, and then press the CH1 button again to close the CH1 channel.
- Use the touch screen function to close the channel by clicking on the number on the channel menu at the bottom of the screen.

5.2 To Adjust the Vertical Scale

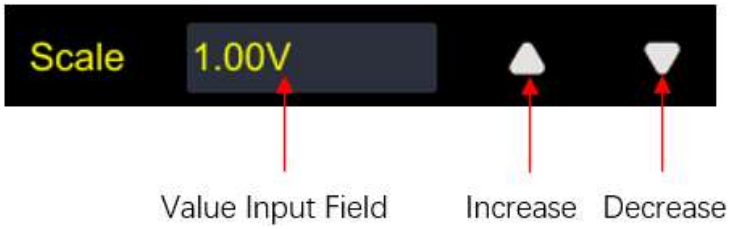
Vertical gear refers to the voltage value represented by each grid in the vertical direction of the display screen, usually expressed as V/div. When adjusting the vertical gear, the waveform display amplitude will increase or decrease, and the gear information in the channel status label will also change in real time.



The probe ratio and input impedance of the vertical gear and channel settings are related. By default, the probe ratio is 1X and the input impedance is 1M Ω. The vertical gear range is: 500uV/div~10V/div.

When the CH1 channel is open, the vertical gear can be adjusted through the following methods:

- Rotate the knob corresponding to CH1 to adjust the vertical gear within an adjustable range. Rotate clockwise to decrease the gear, and counterclockwise to increase the gear.
- Using the touch screen function, click on the CH1 channel menu at the bottom of the screen, and in the pop-up menu, set the vertical gear.

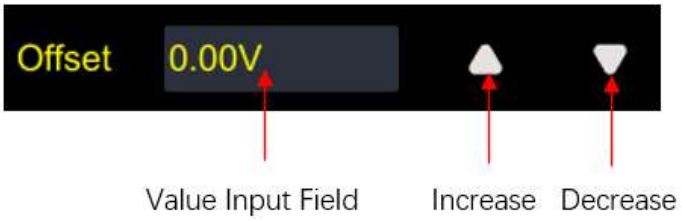


5.3 To Adjust the Vertical Offset

Vertical offset refers to the offset of the zero point position of the channel signal of the waveform in the vertical direction relative to the center of the screen. The unit is consistent with the currently selected amplitude unit. When adjusting the vertical offset, the waveform of the corresponding channel moves up and down, and the offset information in the channel status label also changes in real-time.

Adjust vertical offset:

- Using the touch screen function, adjust the vertical offset by dragging gestures. Please refer to the Drag section for specific operations.
- Rotate the offset knob in the button area to adjust the vertical offset.
- Using the touch screen function, click on the CH1 channel menu at the bottom of the screen, and in the pop-up menu, set the offset.

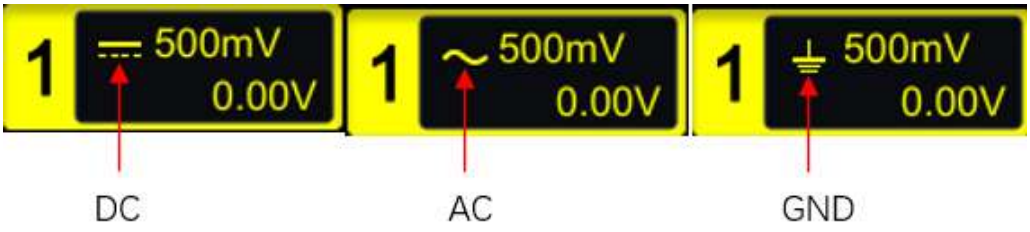


5.4 Channel Coupling

Setting the coupling method can filter out unwanted signals. Open the vertical menu of the channel and click on the **Coupling** menu to select the coupling method. As shown in the following figure.

- When the coupling method is "DC": both the DC and AC components contained in the measured signal can pass through.
- When the coupling method is "AC": the DC component contained in the measured signal is blocked.
- When the coupling method is "GND": both the DC and AC components contained in the measured signal are blocked.

After setting the coupling method, the current coupling method will be displayed at the channel label.



Tip:

When the input impedance is selected as 50 ohms, the coupling mode of the channel can only be set to DC and cannot be changed.

5.5 Bandwidth Limit

This oscilloscope supports bandwidth limiting function. Setting bandwidth limits can reduce noise in the displayed waveform.

- When the bandwidth limit is turned off, the high-frequency components contained in the measured signal can pass through.
- If the bandwidth limit is turned on and limited to 20MHz, the high-frequency components greater than 20MHz contained in the measured signal will be attenuated.

Click to open **[CH1]**, and then click on the **BW Limit** menu to select the restricted bandwidth. You can choose to close, 20M, 100M, 200M, 350M, and the default is 20M. After opening the bandwidth limit, the color B at the channel label matches the channel color.

Tip:

Bandwidth limitation not only reduces noise, but also attenuates or eliminates high-frequency components in the signal.

5.6 Probe Ratio

Open **[CH1]**, then click on the **Attenuation** to set the probe.
The oscilloscope allows users to manually set the probe attenuation ratio, and users must set the probe ratio correctly to obtain accurate measurement results. The default probe ratio is 1X, and the probe range is 0.01X-50000X.
To match the actual attenuation ratio of the probe, it is necessary to adjust the channel attenuation ratio accordingly under the channel menu. Whenever the attenuation ratio of the probe changes, it is necessary to set the corresponding attenuation ratio in the channel menu to ensure the correctness of the waveform amplitude and measurement results displayed on the oscilloscope.

Table 5.1 Probe Ratio

Menu	Attenuation Ratio(display amplitude of the signal: actual amplitude of the signal)
0.01X	0.01:1
0.02X	0.02:1
0.05X	0.05:1
0.1X	0.1:1
0.2X	0.2:1
0.5X	0.5:1
1X	1:1
2X	2:1
5X	5:1

10X	10:1
20X	20:1
50X	50:1
100X	100:1
200X	200:1
500X	500:1
1000X	1000:1
2000X	2000:1
5000X	5000:1
10000X	10000:1
20000X	20000:1
50000X	50000:1

5.7 Input Impedance

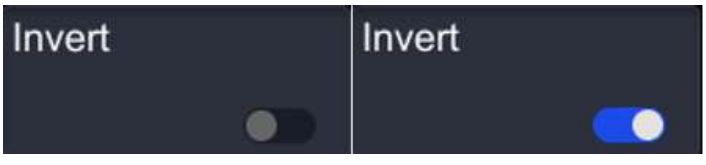
This oscilloscope provides two input impedance modes: 1M Ω and 50 Ω to reduce the circuit load caused by the interaction between the oscilloscope and the circuit under test.

Open **[CH1]**, click on the **Impedance** menu tab, and select an input impedance of 1M Ω or 50 Ω.

- 1M Ω: At this point, the input impedance of the oscilloscope is very high, and the current flowing into the oscilloscope from the tested circuit can be ignored.
- 50 Ω: Match the oscilloscope with a device with an output impedance of 50 Ω.

5.8 Waveform Invert

Open **[CH1]**, click on the inverting menu, select the inverting switch flag to turn it on or off, and the flag will light up to turn on the waveform inverting function. The default is to turn off inverting.



When the waveform inversion is turned off, the waveform is displayed normally; When

the waveform inversion is turned on, the waveform voltage value is inverted. Opening waveform inversion will also change the results of mathematical operations, waveform measurement, and other operations.

Figure 5.1 "Invert" Off

Figure 5.2 "Invert" On

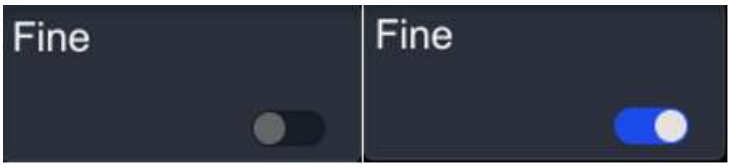
5.9 Amplitude Unit

Open **[CH1]**, click on the **Unit** menu, and set the unit. The optional units are [W], [A],

[V], [U], and the default unit is [V].
After modifying the amplitude unit, the units of functions related to the channel will also change.

5.10 Fine tuning

Open **[CH1]**, click on the **Fine** adjustment menu, and the label will light up to indicate fine adjustment. The default setting is coarse adjustment.



- Fine adjustment: The vertical gear will be adjusted within a small range, changing the vertical resolution to facilitate the observation of waveform details.
- Coarse adjustment: Set the vertical gear in steps of 1 to 2 to 5.

5.11 Channel Delay

When using an oscilloscope for actual measurements, the transmission delay of the probe cable may cause significant errors (zero offset). This series of oscilloscopes supports users to set a delay time to correct the zero offset of the corresponding channel. Zero offset is defined as the offset of the intersection point between the waveform and the trigger level line relative to the trigger position.
Click on **[CH1]**, click on the **Channel Delay** menu, and a numeric keyboard will pop up to set the delay. The delay time can be set from -100ns to 100ns.

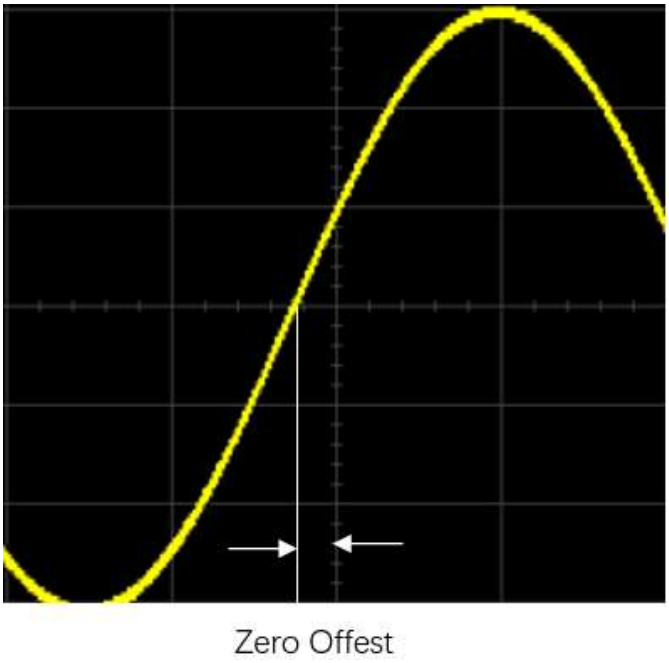


Figure 5.3 Zero Offset

5.12 Offset Cal

When using an oscilloscope for actual measurement, the temperature drift characteristics of the device or external environmental interference cause a small deviation in the zero voltage of the channel, which affects the measurement results of vertical parameters. This series of oscilloscopes supports users to set a zero elimination voltage to correct the zero point of the corresponding channel, thereby improving the accuracy of measurement results.

Click on **[CH1]**, click on the **Offset Cal** menu, and a numeric keyboard will pop up to set the numerical range.

5.13 Vertical Expansion

This series of oscilloscopes supports two vertical expansion methods: center or GND, with the default being center:

- Screen: When changing the vertical gear, the mathematical operation waveform will expand or compress around the center of the screen.
- GND: When changing the vertical gear, the mathematical operation waveform will expand or compress around the zero position (grounding) of the waveform.

5.14 Channel Label

This series of oscilloscopes can have additional channel labels set.
Click on **[CH1]**, click on the **Label** menu, and enter the settings menu.

- **Display:**

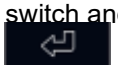
Click on the **Display** the menu label, choose to open or close the channel label, open the label, and the set label will be displayed on the left side of the waveform. The label defaults to 1.

- **Library:**

Click on the **Library** menu tab to select tags such as ADO, ADDR, BIT, CAS, CLK, etc.

- **Label:**

Click on the **Label** menu label, and the label editing keyboard will automatically pop up. Through the touch function, enter the label name on the keyboard.

In the virtual keyboard, click the Select Case switch and rotate the multi-functional knob V0 to select letters. After entering, press the  key to end editing. You can also use the touch screen function to operate through touch gestures. If the **Display** is turned on, the set label will be displayed on the left side of the CH1 waveform.
If the inputted characters are deleted or modified, they can be achieved through the multi-functional knob V0 or by using the touch screen function:

- Using a multifunctional knob: You can only delete the character until it needs to be deleted or modified by pressing the delete button. If you modify characters, you need to re-enter the required characters.
- Using the touch screen function: simply position the cursor to the right of the character that needs to be deleted or modified through a touch gesture, and press the delete button to delete the character. If you modify characters, you need to re-

enter the required characters.

In addition, the "Input Selection Area" is only available when selecting the Chinese input method.

6 To Set the Horizontal System

This chapter contains detailed information about the oscilloscope horizontal system. We suggest that you read carefully to understand the setting functions and operation of the oscilloscope horizontal system.

Horizontal adjustment can be achieved through the following methods:

- Click on the horizontal time base label at the top of the screen to adjust the horizontal setting.

- Adjust the time base knob and displacement knob in the button area to adjust the horizontal setting.

- Use the touch function to adjust the horizontal setting.

This chapter includes:

6.1 To Adjust the Horizontal Position

Horizontal displacement refers to the displacement of the waveform trigger points of all channels in the horizontal direction relative to the center of the screen. When the waveform trigger point is located on the left (right) side of the screen center, the horizontal displacement is positive (negative).
When changing the horizontal displacement, the waveform trigger points and displayed waveforms of all channels move left and right; The horizontal displacement information above the screen changes in real-time.

The horizontal displacement can be adjusted by the following methods:

- Click on the horizontal label at the top of the screen, and in the pop-up menu, enter the displacement value directly or click the increase/decrease knob on the right to set the displacement, as shown in the following figure.

- Adjust the horizontal displacement by rotating the horizontal displacement knob.

- Adjust the horizontal displacement by dragging the gesture, and the horizontal trigger position icon on the screen will move left and right with the sliding of the hand. Please refer to the Drag section for specific operations.

6.2 To Adjust the Horizontal Time Base

The horizontal time base is the time value represented by each grid in the horizontal direction of the display screen, generally expressed as s/div. The adjustable range of the horizontal time base is 500ps/div~1000s/div. The default value is 1us/div.
Change the horizontal time base, and the horizontal time base information will be displayed in real-time on the screen.

The horizontal time base can be adjusted by the following methods:

- Click on the horizontal label at the top of the screen, and in the pop-up menu, enter the time base value directly or click the increase/decrease knob on the right to set the time base, as shown in the following figure.

- Adjust the horizontal time base by rotating the horizontal time base knob.
- Adjust the horizontal time base through the pinching gesture of the touch function. Please refer to the Pinch section for specific operations.

6.3 Pan or zoom single acquisition or stopped acquisition

After the oscilloscope stops, the stopped display may contain several collected data with useful information, but only the last collected data can be panned and scaled. Pan and zoom the data collected in a single or stopped collection.

6.4 Delayed Sweep

Delay scanning is used to horizontally amplify a section of waveform in order to view waveform details.
Click the **[Zoom]** button in the button area, and the screen is divided into two display areas.

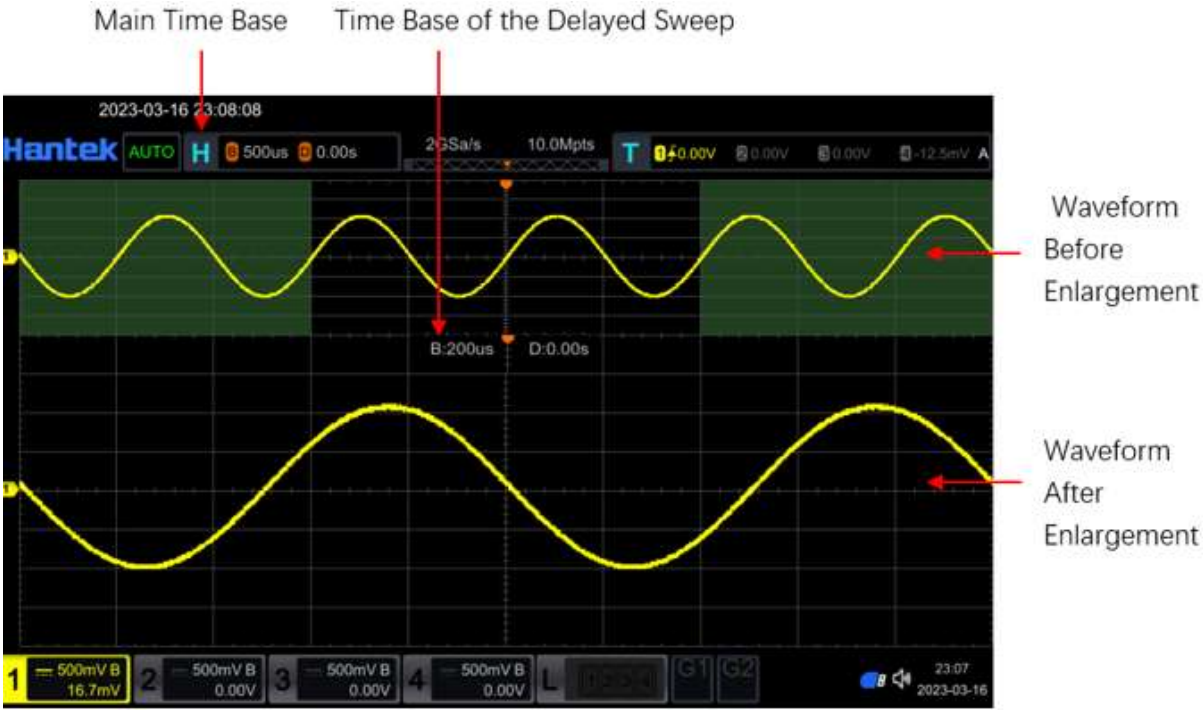


Figure 6.1 Delayed Sweep Mode

Waveform before amplification:

The upper half of the screen is the waveform before amplification, and its horizontal time base (known as the main time base) is displayed in the upper left corner of the screen. You can move the area left and right by adjusting the horizontal displacement, or expand or decrease the area by adjusting the base gear when horizontal.

Amplified waveform:

The lower part of the screen is a horizontally extended delayed scanning waveform, with its horizontal time base (referred to as the delayed scanning time base) displayed on the screen. Delayed scanning time base improves resolution compared to the main time base.

Tip:

The delay scanning time base should be less than or equal to the main time base.

7 To Set the Sample System

Click on the button area **[Acquire]** to enter the settings menu.

This chapter includes:

7.1 Timebase Mode

Open **[Acquire]**, click on the **Timebase Mode** menu, and select time base modes: YT mode, XY mode, and ROLL mode.

7.1.1 YT Mode

In this mode, the Y-axis represents the usual voltage and the X-axis represents the time.
In YT mode, when the horizontal time base is set to 100 ms/div or slower, the oscilloscope enters scanning mode. In this mode, the oscilloscope first collects data on the left side of the trigger point, then waits for the trigger condition, and continues to complete the waveform on the right side of the trigger point after the trigger condition occurs. At the same time, the waveform data currently collected is displayed.

Note:

Scanning mode observation signal frequency is low, it is recommended to set the "Channel Coupling" mode to "DC".

7.1.2 XY Mode

This series of oscilloscopes supports the waveform display window "XY Window" in XY mode, where both the X-axis and Y-axis represent the voltage level.
When the time base mode is selected as XY, the oscilloscope automatically opens channels 1 and 2, and channels 3 and 4 are automatically closed.

Phase difference measurement:

The Lissajous method can conveniently measure the phase difference between two signals of the same frequency. The following figure shows the measurement principle diagram of phase difference.

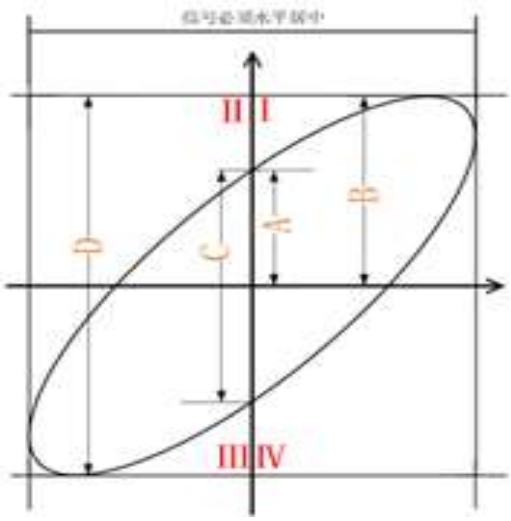


Figure 7.1 Measurement Schematic Diagram of Phase Deviation

According to or, where θ the definition of A, B, C, and D is the phase difference angle between channels, as shown in the figure above. Therefore, the difference angle can be obtained, which is:
If the main axis of the ellipse is within the I and III quadrants, the obtained phase difference angle should be within the I and IV quadrants, that is, within (0 to π/2) or (3 π/2 to 2 π). If the main axis of the ellipse is within the II and IV quadrants, the obtained phase difference angle should be within the II and III quadrants, that is, within (π/2 to π) or (π to 3 π/2). The X-Y function can be used to test the phase change generated by a signal passing through a circuit network. Connect the oscilloscope to the circuit and monitor the input and output signals of the circuit.

Note:

In XY mode, the oscilloscope will forcibly open CH1 and CH2 channels and close CH3 and CH4 channels. In general, longer sampling waveforms can achieve better display results, but due to storage depth limitations, longer waveform lengths mean a need to reduce the sampling rate. Therefore, in this measurement process, appropriately reducing the sampling rate can obtain a better display effect of the Lissajous pattern.
In XY mode, "Scan Mode", "Vector Display", "Protocol Decoding", "Acquisition Mode", "Pass/Fail Test", "Digital Channel", and "Afterglow Time" do not work.

Using the Lissajous method:

1. Connect a sine signal to CH1, and then connect a sine signal with the same frequency, amplitude, and phase difference of 90 ° to CH2.
2. Click on **[AUTO Scale]**, select the XY mode, and rotate the knob to adjust the sampling rate appropriately to obtain a better Lissajous pattern for better observation and measurement.
3. Adjust the vertical knobs corresponding to CH1 and CH2 channels to make the signal easy to observe. At this point, the circle shown in the following figure should

be obtained.

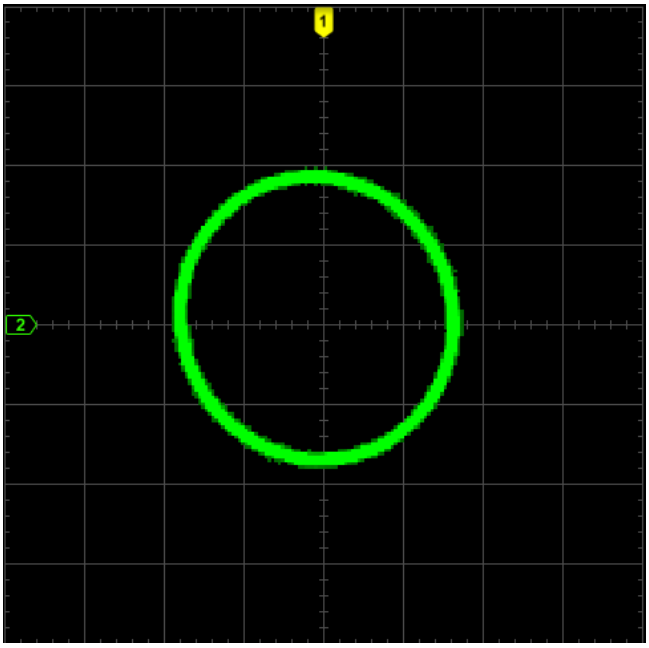


Figure 7.2 XY mode waveform

4. Observing the measurement results in the above figure and based on the phase difference measurement principle diagram (Figure 7.1), it can be concluded that A/B (C/D)=1, that is, the difference angle between the input signals of the two channels is θ=± arcsin1=90 °

7.1.3 ROLL Mode

Select the ROLL mode, in which the waveform scrolls from right to left to refresh the display.

- If the current delay scanning is turned on, the delay scanning will automatically turn off when the "ROLL" time base mode is enabled. When the "YT" time base mode is reactivated, delayed scanning will be turned back on.
- The following functions cannot be set in ROLL mode: "Adjust horizontal displacement" (this function is available when the oscilloscope is running in "STOP"), "Delay scanning", "Trigger oscilloscope", "Protocol decoding", "Pass/fail testing", "Waveform recording and playback", "Set afterglow time".

7.2 Acquisition Mode

Open **[Acquire]**, click on the **Acquisition** method menu, select Normal, Average, Peak, High Res, and the default acquisition method is Normal.

7.2.1 Normal

In this mode, the oscilloscope samples the signal at equal time intervals to reconstruct the waveform. For most waveforms, using this mode can produce the best display effect.

7.2.2 Average

In this mode, the oscilloscope averages the waveforms sampled multiple times to reduce random noise on the input signal and improve vertical resolution. The higher the average frequency, the smaller the noise and the higher the vertical resolution, but the displayed waveform also responds slower to waveform changes.

After selecting the "average" mode, the sampling system menu will display the average number of times. Press the **Averages** key and rotate the multi-functional knob to set it to 2, 4, 8, 16, 32, 64, 128, 256, 512, or 1024. The default is 2.

Figure 7.3 Waveforms before Averaging

Figure 7.4 Waveforms after Averaging

7.2.3 Peak

In this mode, the oscilloscope collects the maximum and minimum values of the signal within the sampling interval to obtain the envelope of the signal or narrow pulses that may be lost. Using this mode can avoid signal aliasing, but the displayed noise is relatively high.
In this mode, the oscilloscope can display all pulses at least as wide as the sampling period.

7.2.4 High Resolution

This mode adopts a supersampling technique that averages adjacent points of the sampled waveform, reducing random noise on the input signal and producing a smoother waveform on the screen. Usually used when the sampling rate of a digital converter is higher than the storage rate of the acquisition memory.

Note:

The averaging method used in "average" and "high resolution" modes is different, with the former being "multiple sample averaging" and the latter being "single sample averaging".

7.3 Memory Depth

Storage depth refers to the number of waveform points that an oscilloscope can store in a single trigger acquisition. It reflects the storage capacity of the acquisition memory.

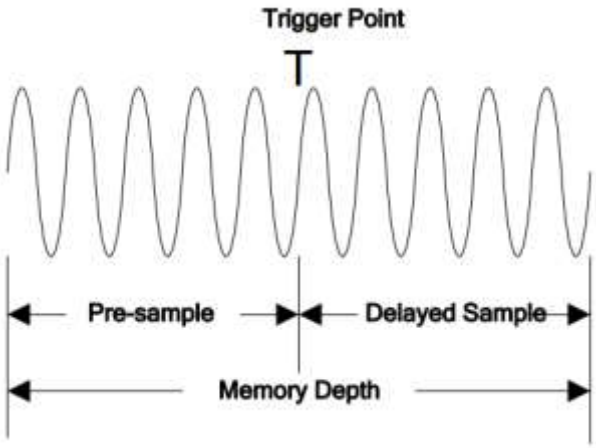


Figure 7.5 Memory Depth

The following equation shows the relations among memory depth, sample rate, and horizontal time base scale:

MDepth = SRate * TScale * HDivs

MDepth ——indicates the memory depth. The unit is pts.

SRate——indicates the sample rate. The unit is Sa/s.

TScale ——indicates the horizontal time base scale. The unit is s/div.

HDivs ——indicates the number of grids in the horizontal direction. The unit is div.

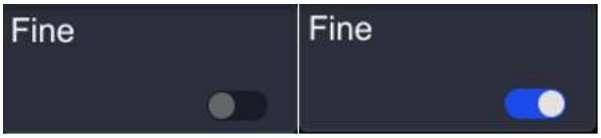
Therefore, under the same horizontal time base scale, a higher memory depth can ensure a higher sample rate.

Open **[Acquire]**, click on the **Mem Depth** menu, and select the storage depth.

- Single channel mode: Storage depth options: Auto, 25K, 250K, 2.5M, 25M, 50M, 100M, 250M, 500M, 2G.
- Dual channel mode: Storage depth options: Auto, 12.5K, 125K, 1.25M, 12.5M, 25M, 50M, 125M, 250M, 1G.
- Full channel mode: Storage depth options: Auto, 6.25K, 62.5K, 625K, 6.25M,12.5M, 25M, 62.5M, 125M, 500M.

7.4 Fine

Open **[Acquire]**, click on the **Fine** menu, and the label will light up to indicate fine tuning. The default setting is coarse tuning.



Fine adjustment: The time base will be adjusted within a small range, changing the time base to facilitate observation of waveform details.
Coarse adjustment: Set the horizontal time base in steps of 1-2- 5.

7.5 Horizontal Expansion

Horizontal expansion refers to the reference position on which the screen waveform is horizontally expanded or compressed when adjusting the horizontal time base. Open **[Acquire]**, click on the **Expand** menu, and select the open expansion method. The extensions supported by this oscilloscope include center, left, right, trigger point, and customization. The default is' Center '.

- Center: When changing the horizontal time base, the waveform expands or compresses horizontally around the center of the screen. Press the multi-functional knob to quickly restore the reference position to 0.
- Left: When changing the horizontal time base, the waveform expands or compresses horizontally around the leftmost side of the screen.
- Right: When changing the horizontal time base, the waveform expands or compresses horizontally around the far right side of the screen.
- Trigger: When changing the horizontal time base, the waveform expands or compresses horizontally around the trigger point.
- User: When changing the horizontal time base, the waveform expands or compresses horizontally around the user-defined reference position.

After selecting " User ", the menu will display "Expand User ". Click on the " **Expand User** " menu label, rotate the multi-functional knob to set the extension benchmark, and the range can be set from the rightmost side of the screen to the leftmost side of the screen. The default is the center of the screen.

7.6 Segmented acquisition

Open **[Acquire]**, click on the **Segmented** menu, and enter the settings menu. Alternatively, click on the navigation icon **[Hantek]** in the upper left corner of the screen and select segmented acquisition. Entering the settings menu and lighting up the label means opening segmented collection. The segmented collection results appear in the middle of the screen.
Click on the **Segmented Set** menu tab and set the segmented settings by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

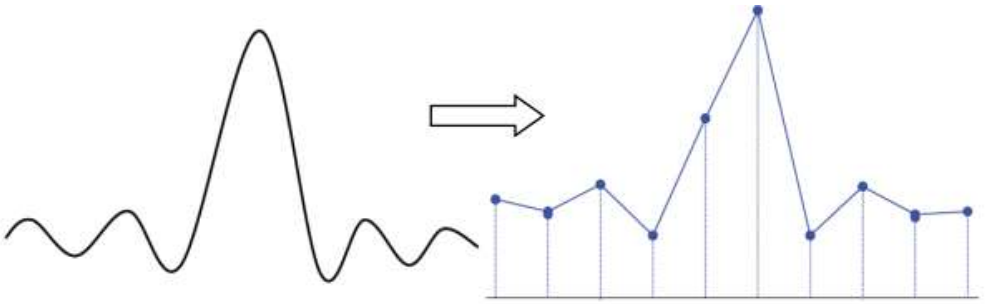
7.7 Sample Rate

Sampling refers to the process in which an oscilloscope converts analog signals into digital signals at certain time intervals and stores them in sequence. The sampling rate is the reciprocal of the time interval.

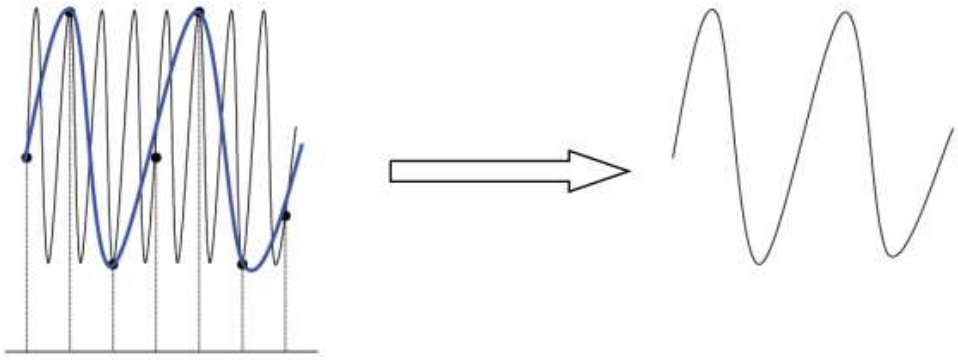
Open **[Acquire]** and the sampling rate menu will display the current sampling rate value. The sampling rate is displayed in the status bar at the top of the screen, and can be indirectly changed by adjusting the horizontal time base or modifying the storage depth.

The impact of low sampling rate on waveform:

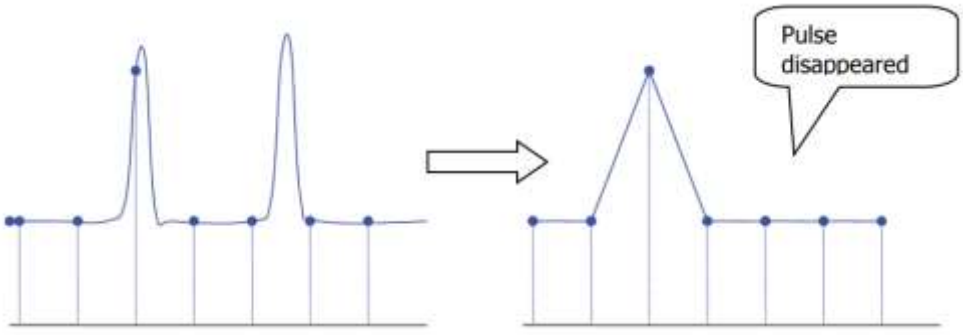
- Waveform distortion: Due to the low sampling rate, some waveform details are missing, resulting in significant differences between the waveform displayed by the oscilloscope sampling and the actual signal.



- Waveform aliasing: since the sampling rate is lower than 2 times of the actual signal frequency (Nyquist frequency), the waveform frequency when reconstructing the sampled data is lower than the frequency of the actual signal.



- Waveform leakage: Due to the low sampling rate, the waveform reconstructed from the sampled data did not reflect all the actual signals.



8 To Trigger the Oscilloscope

The so-called triggering refers to setting certain triggering conditions according to requirements. When a certain waveform in the waveform flow meets this condition, the oscilloscope immediately captures the waveform and its adjacent parts, and displays them on the screen. The trigger determines when the oscilloscope starts acquiring data and displaying waveforms. Once the trigger is set correctly, the oscilloscope can convert unstable displays or blank screens into meaningful waveforms. Here are some basic concepts of triggers.

You can enter the trigger menu by:

- Click on the button area **[Trig Menu]** to enter the trigger menu.
- Click on the trigger tab at the top of the screen (as shown in the figure below), and a simple trigger menu will pop up on the screen.

This chapter includes:

8.1 Trigger LEVEL

The trigger level is the signal voltage corresponding to the set trigger point, which is related to the type of trigger signal source.

- The trigger flag and trigger level line move up and down as the trigger level changes. The trigger flag and channel color remain consistent. When changing the trigger level, a trigger level line will temporarily appear on the screen to tell you the position of the level (the specific value of the trigger level is displayed in the trigger menu label at the top right of the screen). After about 2 seconds of stopping modifying the trigger level, the trigger level line disappears.
- For slope triggering, under amplitude triggering, and over amplitude triggering, it is necessary to set the triggering levels: the level values of level A and level B. Through the **Level Select** menu in the **[Trig Menu]** menu, select the current adjustable levels as level A, level B, and level AB.

The trigger level can also be adjusted through the touch function.

- Using the touch screen function, touch the trigger setting label in the upper right corner of the screen to pop up the trigger setting box as shown in the following figure. Touch the right side of the level value input box to increase or decrease the level value. You can also touch the value input box and directly input the specific value through the pop-up numeric keyboard.

8.2 Trigger Sensitivity

The trigger sensitivity indicator shows the sensitivity of the trigger signal recognition. By adjusting the trigger sensitivity, it can effectively filter out the noise that may be superimposed on the trigger signal and prevent false triggering.

Click on the trigger label in the upper right corner of the screen to pop up the trigger setting box and set the sensitivity.

Set trigger **Sens**:

- Click on the sens box in the menu to pop up the numeric keypad and set the sensitivity value.
- Click the up and down arrow keys on the right to set the sensitivity value.

8.3 Trigger Source

Click on the button area **[Trig Menu]** to enter the **Source** menu. The available data sources include: CH1~CH4, EXT.

External trigger sources can be used to trigger on the 5th channel while collecting data on all 4 channels. Trigger signals (such as external clocks or signals from the circuit to be tested, etc.) will be connected to the EXT trigger source through the external trigger input terminal **[EXT In]** connector.

8.4 Trigger Mode

The following is a schematic diagram of the acquisition memory. To facilitate the understanding of triggering events, the collection memory can be divided into pre triggered buffer and post triggered buffer.

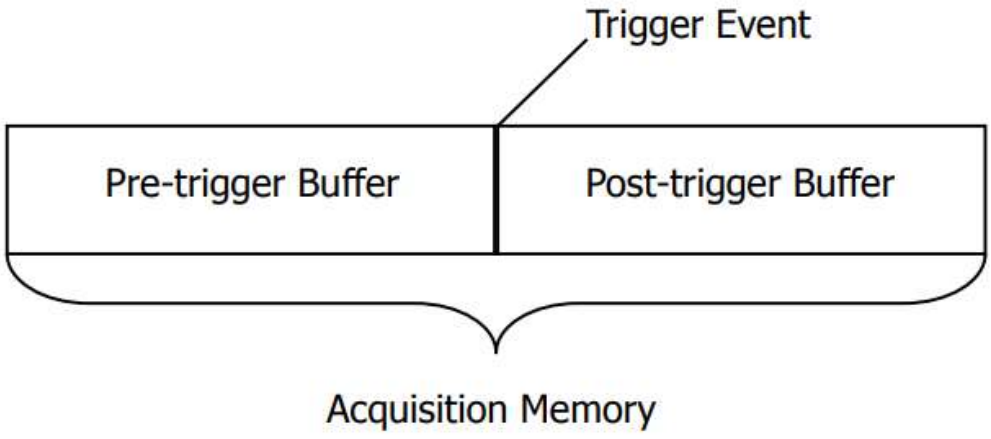


Figure 8.1 Schematic Diagram of the Acquisition Memory

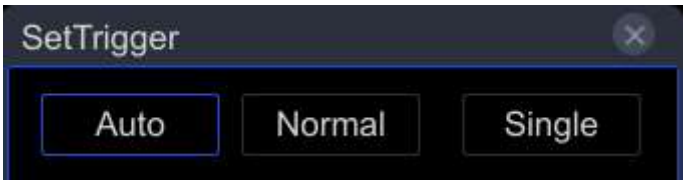
After starting operation, the oscilloscope will first fill the pre triggered buffer. After filling, the oscilloscope will start searching and triggering; During the search period, the sampled data will continue to be transmitted to the pre triggered buffer (new data will continuously overwrite existing data). After searching for a trigger, the pre trigger buffer will contain the events that occurred before the trigger. Then, the oscilloscope will fill the trigger buffer and display the data in the acquisition memory. If the collection is initiated through the **[Run/Stop]** button, the process will be repeated; If the acquisition is

initiated through the **[Single]** key, it will stop after completing a single acquisition (the current displayed waveform can be panned and scaled).
The oscilloscope trigger mode provides Normal, Auto, and Single trigger. The trigger mode defaults to automatic mode.

- Normal mode: Only when the oscilloscope has a valid trigger will the displayed waveform be updated. Before replacing the original waveform with a new waveform, the oscilloscope will display the original waveform. Use the "normal" mode only when you want to view the waveform that is effectively triggered. When using this mode, the oscilloscope only displays the waveform after the first trigger.
- Auto mode: can freely run collection without effective triggering. This mode allows for untriggered scanning waveforms to occur at a time base setting of 100 milliseconds/grid or slower. When the oscilloscope detects a valid triggering condition, complete a triggered acquisition. When the oscilloscope detects that there are no valid triggering conditions, complete a non triggering acquisition.
- Single mode: Only when the oscilloscope has a valid trigger can the collection end and enter the stop state.
- Forced trigger: In both normal and single trigger modes, pressing the forced trigger button in the trigger menu can forcibly generate a trigger signal.

Using the touch screen function, touch the trigger setting label 

in the upper right corner of the screen to pop up the trigger setting box as shown in the following figure. You can switch the trigger method by touching "Auto", "Normal", and "Single " with the touch gesture.



When you are not familiar with a signal feature, the oscilloscope should be set in the "Auto" mode to ensure that the waveform is displayed even if other trigger settings are incorrect. Although the waveform may not be stable, it can provide us with intuitive judgment for further adjusting the oscilloscope.
When we set specific triggering conditions for a specific signal, especially when the time interval between meeting the triggering conditions is relatively long, we need to set the triggering mode to "Normal" to prevent the oscilloscope from automatically forcing triggering.

8.5 Trigger Holdoff

The trigger release function can be used to generate stable and complex waveforms (such as amplitude modulation columns) for display. 'Release suppression' refers to the time difference between the oscilloscope detecting one trigger and preparing to detect

another trigger. During the suppression period, the oscilloscope will not trigger. For a pulse train, the suppression time can be adjusted so that the oscilloscope only triggers on the first pulse of the train.

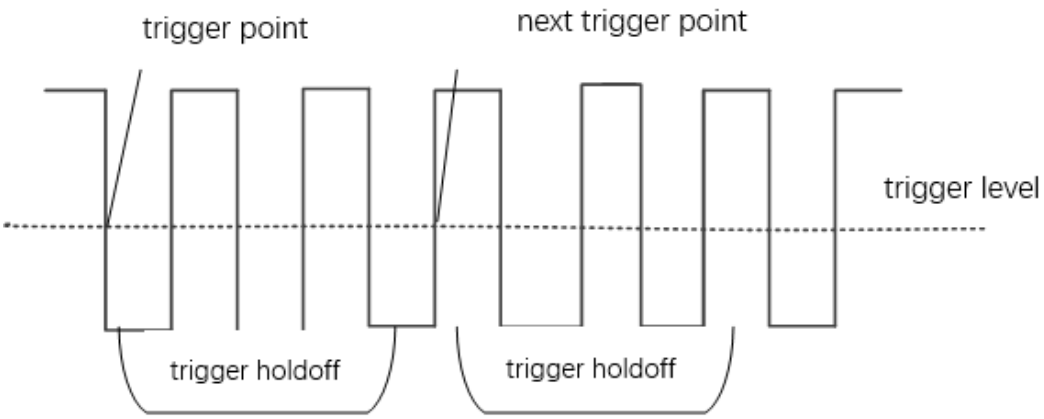


Figure 8.2 Schematic Diagram of Trigger Holdoff

Set the Holdoff:

- Open the **[Trig Menu]**, click on the **Holdoff** menu label, and rotate the multi-functional knob V0 to adjust the trigger release value.
- Click to **Holdoff**, and a numeric keyboard will appear. Set the value directly on the keyboard.

The adjustable range of release time is 8ns to 10s.

8.6 Trigger Type

This series of machines has multiple triggering functions. Open the **[Trig Menu]**, click on the **Type** menu, and select the trigger type.

- Edge Trigger
- Pulse Trigger
- Video Trigger
- Slope Trigger
- Overtime Trigger
- Window Trigger
- Runt Trigger
- Superamp Trigger
- Pattern Trigger
- Delay Trigger
- Setup/Hold Trigger
- UART Trigger(Option)
- LIN Trigger (Option)
- CAN Trigger (Option)
- SPI Trigger (Option)

- I2C Trigger (Option)

8.6.1 Edge Trigger

The edge trigger type identifies the trigger by searching for the specified edge (Rising, Falling, Either) and voltage level on the waveform.

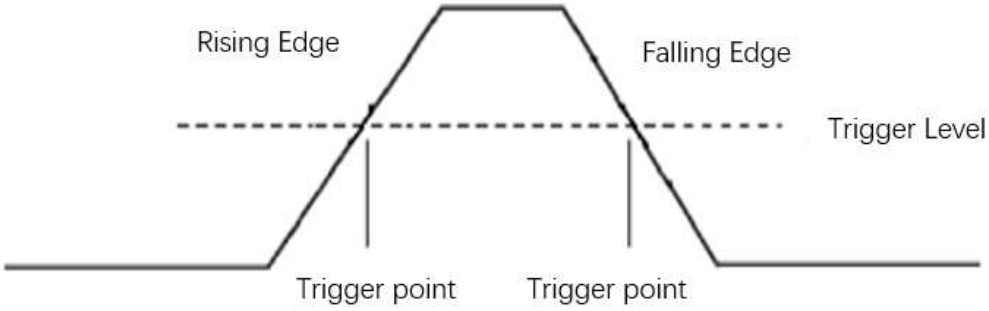
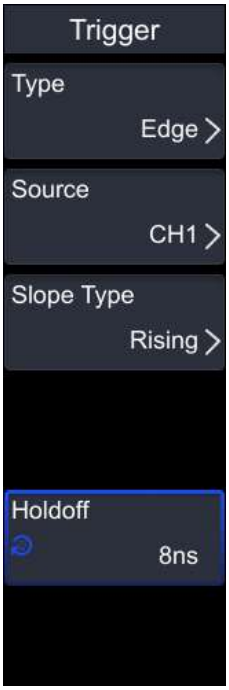


Figure 8.3 Rising/Falling

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label, select "Edge ", and set the trigger settings.
After selecting the trigger type, the current trigger setting information (including trigger type, trigger signal source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab to select sources such as CH1, CH2, CH3, CH4, and

EXT.

3. Slope Type:

Click on the **Slope Type** menu label, and the edge types that can be selected include: rising edge, falling edge, or any edge.

- Rising: Set the signal's rising edge to trigger.
- Falling: Set the signal falling edge trigger.
- Either: Set the signal to trigger the rising or falling edge.

4. Holdoff:

Click on the **Holdoff** menu tab, and set the trigger suppression by selecting the multi-functional knob V0 or directly setting the trigger suppression with the pop-up numeric keyboard.

5. Noise Reject:

Please refer to the introduction in the section on Noise Rejection for specific settings related to Noise Reject.

Note:

The automatic setting button will set the trigger type to Edge trigger and the trigger slope to rising edge.

8.6.2 Pulse Trigger

Pulse width trigger sets the oscilloscope to trigger on a specified width of positive or negative pulses. You can set the trigger source, polarity (positive pulse width, negative pulse width), limiting conditions, and pulse width in this menu.

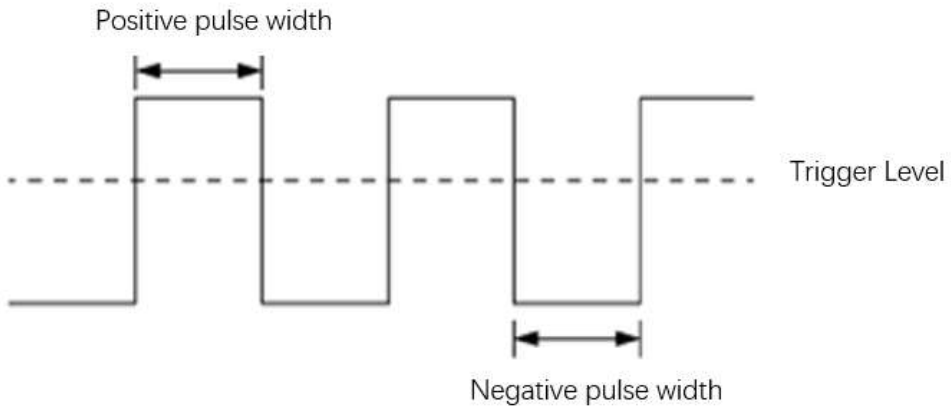


Figure 8.4 Positive Pulse Width/Negative Pulse Width

Click on **[Trig Menu]** in the button area to open the trigger menu.

1. Type:

Click on the **Type** menu label and select "Pulse" to set the trigger.After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab to select sources such as CH1, CH2, CH3, CH4, and EXT.

3. Polarity:

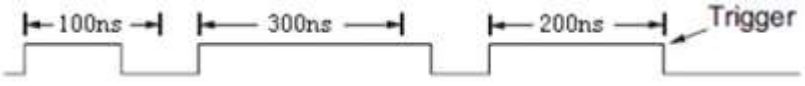
Click on the **Polarity** menu tab to select either positive or negative polarity.

- Positive polarity: triggered when the positive polarity pulse width of the signal is set.
- Negative polarity: triggered when the signal negative polarity pulse width is set.

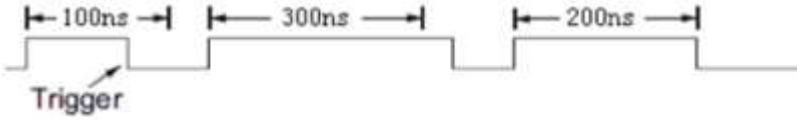
4. When:

Click on the **When** menu tab and select Trigger Condition.

- =(Equal to time value): Only when the positive or negative pulse width of the input signal is equal to the set pulse width can it be triggered.
  For example, for a positive pulse, if t (actual pulse width) is set to 200ns, the waveform is triggered.



- != (Not equal to time value): Only when the positive or negative pulse width of the input signal is not equal to the set pulse width can it be triggered.



- > (greater than the time value): Only when the positive or negative pulse width of the input signal is greater than the set pulse width can it be triggered.
  For example, for a positive pulse, if t (actual pulse width) is set to>100ns, the waveform is triggered.

- <(less than the time value): Only when the positive or negative pulse width of the input signal is less than the set pulse width can it be triggered.
  For example, for a positive pulse, if t (actual pulse width) is set to<100ns, the waveform is triggered.

5. Width:

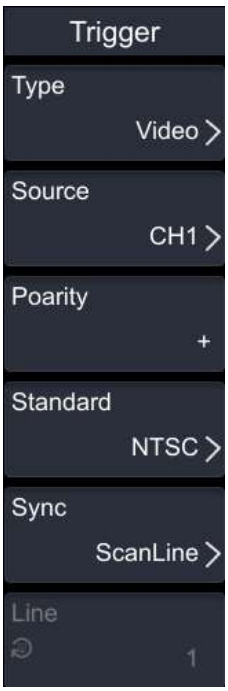
Click on the **Width** menu tab and select the multi-functional knob V0 to set the trigger release or the pop-up numeric keyboard to directly set the trigger release.

6. Holdoff:

Click on the **Holdoff** menu tab, and set the trigger suppression by selecting the multi-functional knob V0 or directly setting the trigger suppression with the pop-up numeric keyboard.

8.6.3 Video Trigger

Video triggering can be used to capture complex waveforms of most standard analog video signals and high-definition video signals. The trigger circuit can detect the vertical and horizontal intervals of the waveform and generate a trigger based on the selected video trigger setting. This series of oscilloscopes supports NTSC (National Television Standards Committee) and PAL.
Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu tab, select "Video", and set the trigger settings.
After selecting the trigger type, the current trigger setting information (including trigger type, trigger signal source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab to select sources such as CH1, CH2, CH3, CH4.

3. Polarity:

Click on the **Polarity** menu tab to select either positive or negative polarity.

- Positive polarity: triggered when the positive polarity pulse width of the signal is set.
- Negative polarity: triggered when the signal negative polarity pulse width is set.

4. Standard:

Click on the **Standard** menu tab and select the desired video standard. The video standards supported by this series of oscilloscopes include NTSC, PAL/SCEAM, 480p/60Hz, and other video standards.

Table 8.1 Video Standard

Video Standard	Frame Frequency	Scan Type	TV Scan Line

NTSC	30	Interlaced Scan	525
PAL/SECAM	25	Interlaced Scan	625

5. Sync:

Click on the **Sync** menu tab and select the desired field or line (ScanLine, Line, Odd, Even, AllFields) to trigger the signal.

- ScanLine: displays a complete line, including a portion of the previous and next lines. The oscilloscope is triggered at any line.
- Line: Display a complete line, including a portion of the previous and next lines. Select a specified number of rows for the oscilloscope to trigger based on user selection.
- Odd: displays multiple fields and the oscilloscope only triggers on odd fields.
- Even: display multiple fields and the oscilloscope only triggers on even numbered fields.
- AllFields: Display multiple fields and trigger the oscilloscope on any field.

6. Line:

Click on the **Line** number menu tab and set the line number in the field to be triggered. When selecting the synchronization method as the number of lines, you can specify the number of lines. The range of line count settings is related to the currently selected video standard, with settings ranging from 1 to 525 (NTSC) and 1 to 625 (PAL/SECAM).

8.6.4 Slope Trigger

Slope triggering sets the positive or negative slope triggering of the oscilloscope from one level to another within a specified time.
As shown in the figure below, we define the time difference between the two points (A and B) where the high and low trigger levels intersect with the rising (falling) edge of the waveform as the positive (negative) slope time.

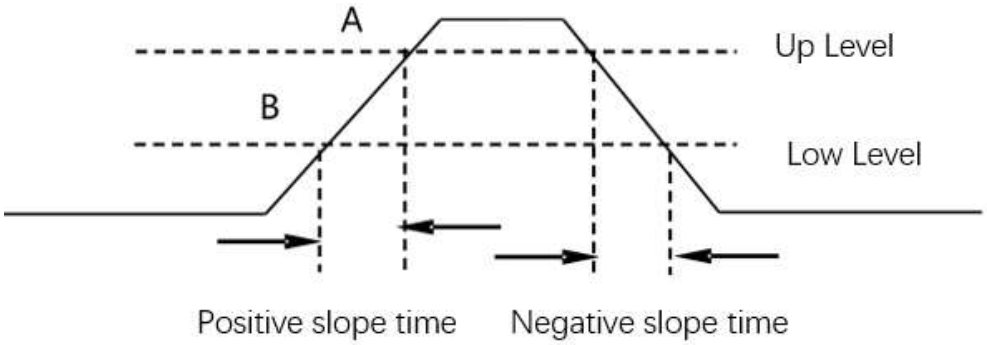
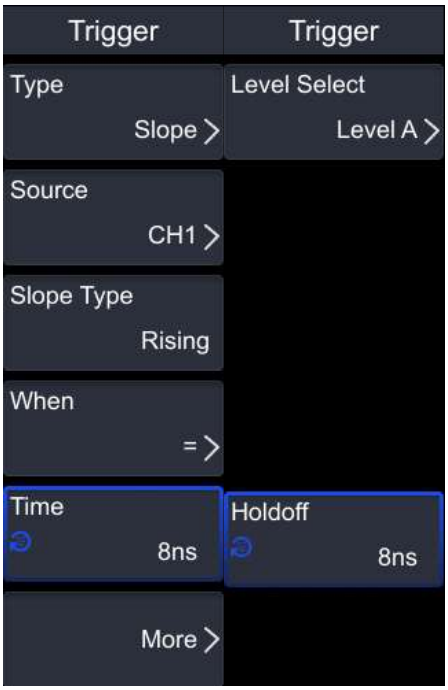


Figure 8.5 Positive Slope Time/Negative Slope Time

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "Slope" to set the trigger.
After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab to select sources such as CH1, CH2, CH3, CH4.

3. Slope Type:

Click on the **Slope Type** menu tab to select rising and falling edges.

- Rising: Set the signal slope condition to trigger with a positive slope.
- Falling: Set the signal slope condition to trigger with a negative slope.

4. When:

Click on the **When** menu tab and select Trigger Condition.

- <(less than time value): Only when the positive or negative slope time of the input signal is less than the set time value can it be triggered.
- >(Greater than time value): It can only be triggered when the positive or negative slope time of the input signal is greater than the set time value.
- != (Not equal to time value): When the positive or negative slope time of the input signal is not equal to the set time value.

- =(Equal to time value): When the positive or negative slope time of the input signal is equal to the set time value.

5. Time:

Click on the **Time** menu tab and set the trigger release by selecting the multi-functional knob V0 or directly set it on the pop-up numeric keyboard.

6. Level Select:

Click on the **Level Select** menu tab, and the level selections are: Level A, Level B, and Level AB.

7. Holdoff:

Click on the **Holdoff** menu tab, and set the trigger suppression by selecting the multi-functional knob V0 or directly setting the trigger suppression with the pop-up numeric keyboard.

8.6.5 Overtime Trigger

Triggered when the time interval (△T) from the rising edge (or falling edge) of the input signal to the end of the adjacent falling edge (or rising edge) through the triggering level is greater than the set timeout time. As shown in the following figure:

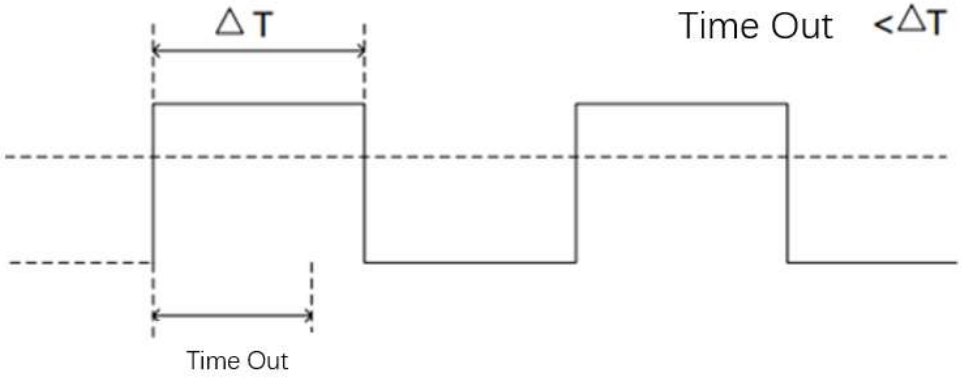
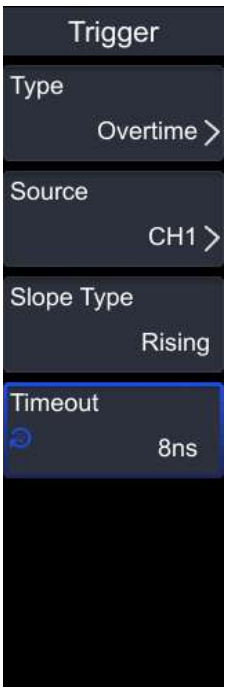


Figure 8.6 Overtime Trigger

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "Overtime" to set the trigger.
After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab to select sources such as CH1, CH2, CH3, CH4, and EXT.

3. Slope type:

Click on the **Slope Type** menu tab to select rising and falling edges.

- Rising: Start timing by triggering the level on the rising edge of the input signal.
- Falling: Start timing by triggering the level at the falling edge of the input signal.

4. Timeout:

Click on the **Timeout** time menu tab, and set the trigger release by selecting the multi-functional knob V0 or the pop-up numeric keyboard to directly set it.

8.6.6 Window Trigger

Window triggering provides high and low trigger levels. When the input signal passes the high or low trigger level, the oscilloscope triggers.

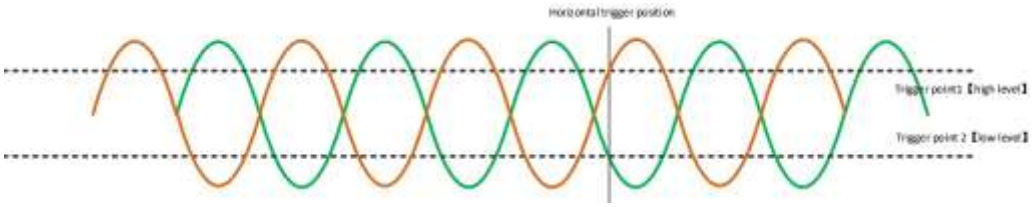
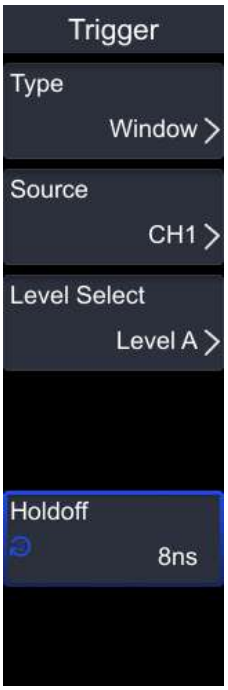


Figure 8.7 Window Trigger

- If both the high and low levels are within the waveform range, the waveform is triggered on the rising or falling edge at the same time.
- If the high level is within the waveform range and the low level is outside the waveform range, the waveform will only trigger on the rising edge.
- If the high level is outside the waveform range and the low level is within the waveform range, the waveform will only trigger on the falling edge.

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "Window trigger" to set the trigger. After selecting the trigger type, the current trigger setting information (including trigger type, trigger signal source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab and select CH1-CH4. Please refer to the section on triggering data sources for details. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. Level Select:

After setting the triggering conditions, it is necessary to adjust the triggering level to correctly trigger the signal and obtain a stable waveform.
Select the **Level Select** type from the level selection menu tab.

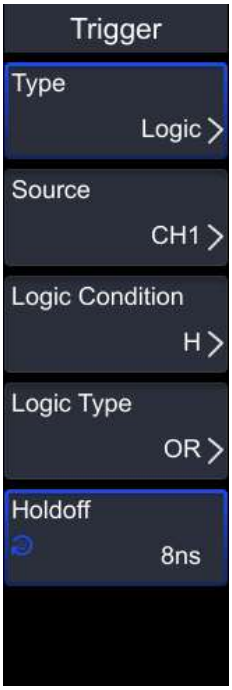
- Level A: Only adjust the upper limit of the trigger level, while keeping the lower limit of the trigger level unchanged.
- Level B: Only adjust the lower limit of the trigger level, while keeping the upper limit of the trigger level unchanged.
- Level AB: Adjust both the upper and lower trigger levels simultaneously, keeping the trigger level difference (i.e. the difference between the upper and lower trigger levels) unchanged.

4. Holdoff:

Please refer to the introduction in the section on Trigger Holdoff for specific settings related to triggering release.

8.6.7 Logic Trigger

Triggered by searching for specified simulation channel logic conditions. Each channel can be set as H, L, X, rising edge, falling edge, or any other edge. The oscilloscope will trigger on the last edge of the logic set to 'true'. If all channel codes are set to 'ignore', the oscilloscope will not trigger.
Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "Logic Trigger" to set the Logic trigger. After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab and select CH1-CH4. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. Code:

Click on the **Code** menu tab and select: H, L, X, Rising, Falling, or Either.

- H: Set the selected channel code value to "1", which means the voltage level is higher than the trigger level/threshold level of the channel.
- L: Set the selected channel code value to "0", which means the voltage level is lower than the trigger level/threshold level of the channel.
- X: Set the selected channel code value to "X", which means that the channel is not part of the code. When all channels in the code are set to "ignore", the oscilloscope will not trigger.
- Rising edge: Set the code pattern to the rising edge on the selected channel.
- Descending edge: Set the code pattern to the descending edge on the selected channel.
- Arbitrary edge: Set the code pattern to any edge on the selected channel.

4. Logic type:

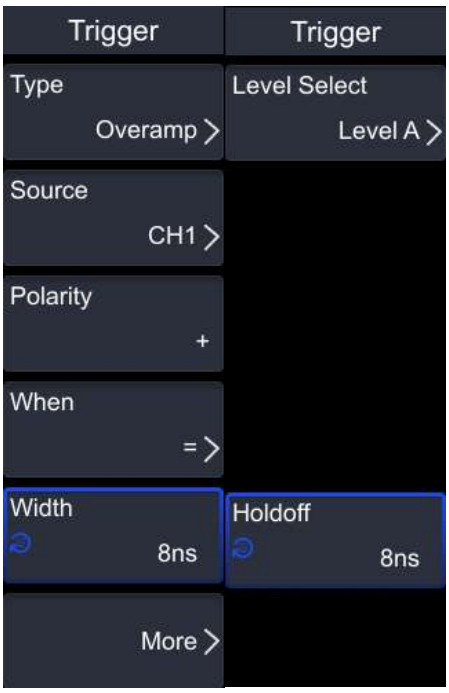
Click on the **Logic type** menu tab to select: OR, AND.

5. Holdoff:

Please refer to the introduction in the section on Trigger Holdoff for specific settings related to Holdoff.

8.6.8 Overamp Trigger

Overamplitude triggering provides a high trigger level and a low trigger level, which are triggered when the input signal rises above the high trigger level or falls below the low trigger level and the timeout pulse width meets the user's set width.
Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "Overamp trigger" to set the over amplitude pulse trigger. After selecting the trigger type, the current trigger setting information (including trigger type, trigger signal source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab and select CH1-CH4. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. Polarity:

Select the **Polarity** of the pulse that can trigger the under amplitude pulse in the polarity menu tab.

- Positive polarity: triggered on a forward under amplitude pulse.
- Negative polarity: triggered on a negative under amplitude pulse.

4. When:

Select the trigger limit condition for under amplitude pulse triggering in the **When** menu tab.

- >[Greater than the set width value]: Only when the negative or positive pulse width is greater than the set width can it be triggered (with a pulse width error of 5%).
- <[Less than the set width value]: Only when the negative or positive pulse width is less than the set width can it be triggered (with a pulse width error of 5%).

- =[Equal to the set width value]: Only when the negative or positive pulse width is equal to the set width can it be triggered (with a pulse width error of 5%).
- != [Not equal to the set width value]: Only when the negative or positive pulse width is not equal to the set width can it be triggered (with a pulse width error of 5%).

5. Width:

Click on the **Width** menu tab and set the width value directly by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

6. Level Select:

After setting the triggering conditions, it is necessary to adjust the triggering level to correctly trigger the signal and obtain a stable waveform.
Select the **Level Select** type from the level selection menu tab.

- Level A: Only adjust the upper limit of the trigger level, while keeping the lower limit of the trigger level unchanged.
- Level B: Only adjust the lower limit of the trigger level, while keeping the upper limit of the trigger level unchanged.
- Level AB: Adjust both the upper and lower trigger levels simultaneously, keeping the trigger level difference (i.e. the difference between the upper and lower trigger levels) unchanged.

7. Holdoff:

Please refer to the introduction in the section on Trigger Holdoff for specific settings related to triggering release.

8.6.9 Runt Trigger

Under amplitude trigger is used to trigger pulses that cross one trigger level but do not cross another trigger level, as shown in the following figure:

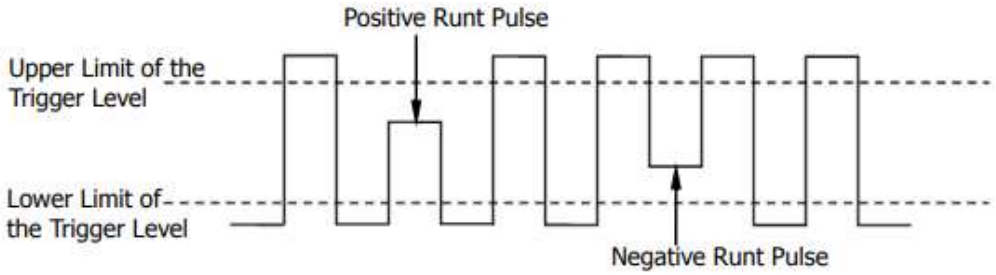
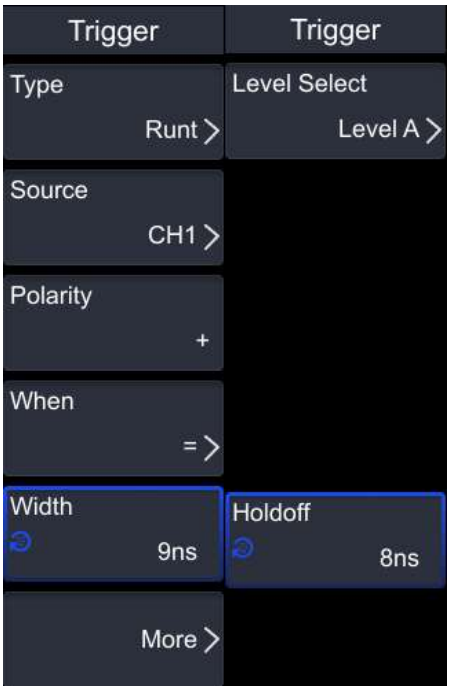


Figure 8.8 Runt Trigger

- Forward under amplitude: The pulse spans the low level but not the high level.
- Negative under amplitude: The pulse crosses the high level without crossing the

low level.

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "Runt trigger" to set the under amplitude pulse trigger. After selecting the trigger type, the current trigger setting information (including trigger type, trigger signal source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab and select CH1-CH4. Please refer to the section on triggering data sources for details. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. Polarity:

Select the **Polarity** of the pulse that can trigger the under amplitude pulse in the polarity menu tab.

- Positive polarity: triggered on a forward under amplitude pulse.
- Negative polarity: triggered on a negative under amplitude pulse.

4. When:

Select the trigger limit condition for under amplitude pulse triggering in the **When** menu tab.

- >[Greater than the set width value]: Only when the negative or positive pulse width is greater than the set width can it be triggered (with a pulse width error of 5%).

- <[Less than the set width value]: Only when the negative or positive pulse width is less than the set width can it be triggered (with a pulse width error of 5%).
- =[Equal to the set width value]: Only when the negative or positive pulse width is equal to the set width can it be triggered (with a pulse width error of 5%).
- != [Not equal to the set width value]: Only when the negative or positive pulse width is not equal to the set width can it be triggered (with a pulse width error of 5%).

5. Width:

Click on the **Width** menu tab and set the width value directly by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

6. Level Select:

After setting the triggering conditions, it is necessary to adjust the triggering level to correctly trigger the signal and obtain a stable waveform.
Select the **Level Select** type from the level selection menu tab.

- Level A: Only adjust the upper limit of the trigger level, while keeping the lower limit of the trigger level unchanged.
- Level B: Only adjust the lower limit of the trigger level, while keeping the upper limit of the trigger level unchanged.
- Level AB: Adjust both the upper and lower trigger levels simultaneously, keeping the trigger level difference (i.e. the difference between the upper and lower trigger levels) unchanged.

7. Holdoff:

Please refer to the introduction in the section on Trigger Holdoff for specific settings related to triggering release.

8.6.10 Delay Trigger

Delay triggering requires setting source A and source B separately. When the time difference (△T) between the edge set by source A (edge A) and the edge set by source B (edge B) meets the preset time limit, the oscilloscope triggers, where edge A and edge B must be adjacent edges.

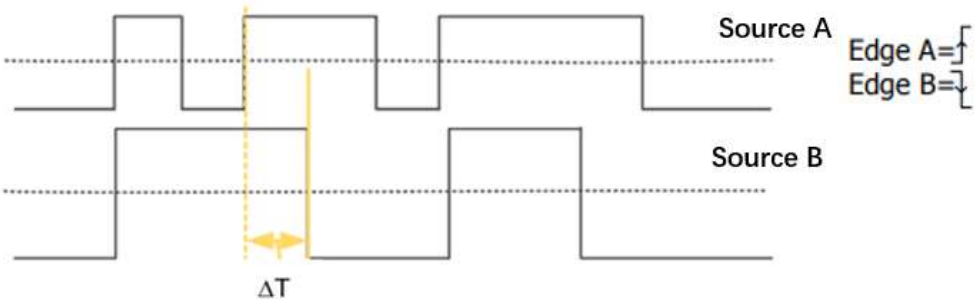
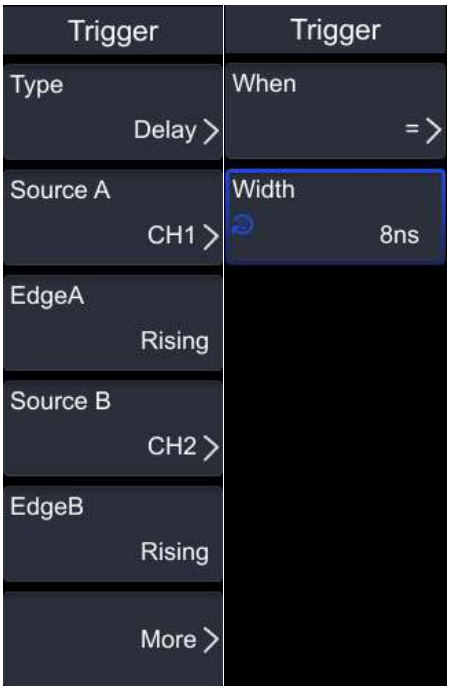


Figure 8.9 Delay Trigger

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "Delay Trigger" to set the delay trigger. After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source A and Source B:

- Source A:

  Click on the menu label of **Source A** and select CH1-CH4 or EXT as the trigger source for Source A. Only by selecting the channel that has already been connected to the signal as the source can stable triggering be obtained.

- Source B:

  Click on the menu label of **Source B** and select CH1-CH4 or EXT as the trigger

source for Source B. Only by selecting the channel that has already been connected to the signal as the source can stable triggering be obtained.

3. **Edge A and Edge B:**

- Edge A:

  Click on the **Edge A** menu label to select the trigger edge type of source A during delayed triggering, and you can choose either the rising or falling edge.

- Edge B:

  Click on the **Edge B** menu label to select the trigger edge type of source B during delayed triggering, and you can choose either the rising or falling edge.

4. **When:**

Select the time limit condition for delayed triggering in the **When** menu tab.

- = [Equal to the set width value]: Only when the time difference between the set edges is equal to the set width can it be triggered (with a pulse width error of 5%).
- != [Not equal to the set width value]: Only when the time difference between the set edges is not equal to the set width can it be triggered (pulse width error is 5%).
- >[Greater than the set width value]: Only when the time difference between the set edges is greater than the set width can it be triggered (with a pulse width error of 5%).
- < [Less than the set width value]: Only when the time difference between the set edges is less than the set width can it be triggered (with a pulse width error of 5%).

5. **Width:**

Click on the menu label for **Width**, and set the width value directly by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

8.6.11 Setup/Hold Trigger

Establishing a hold trigger requires setting the clock channel and data channel separately. The establishment time starts from the time when the data channel crosses the power generation level and ends when the designated clock channel edge arrives; The holding time starts when the designated clock channel edge arrives and ends when the data channel crosses the touch power generation level again (as shown in the following figure). When the establishment time or holding time is less than the preset time, the oscilloscope will trigger.

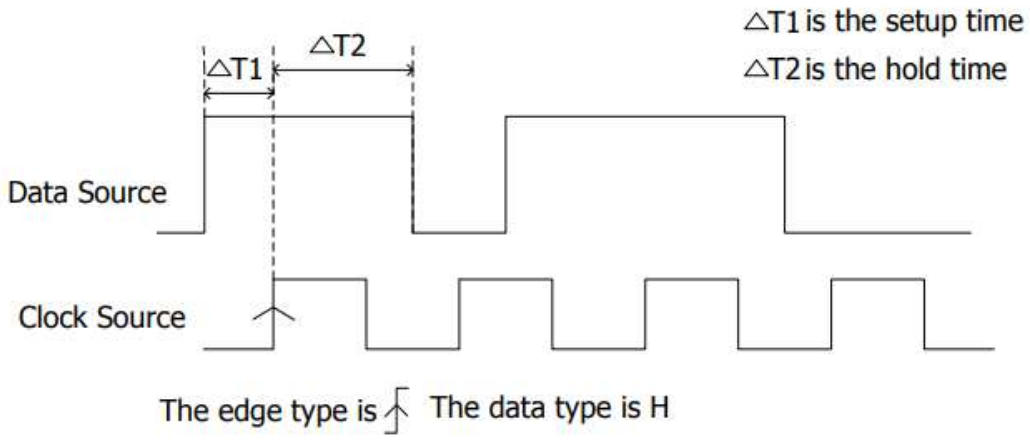


Figure 8.10 Setup/Hold Trigger

Click on **[Trig Menu]** in the button area to open the trigger menu.

1. Type:

Click on the **Type** menu label and select " Setup/Hold " to set the trigger for establishing hold. After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, and it will change with the change of the trigger setting.

2. Clock source:

Click on the **SCL** menu tab and select CH1-CH4 or EXT as the signal source. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. **Slope Type:**

Click on the **Slope** menu tab to select the desired clock edge type, which can be selected as either the rising or falling edge.

4. **Data source:**

Click on the **SDA** menu tab and select CH1-CH4 or EXT as the source. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

5. **Data Type:**

Click on the **Data Type** menu tab to set the effective code type of the data signal. It can be set to H (high level) or L (low level).

6. **When:**

In the **When** menu tab, select Create Hold Trigger Condition.

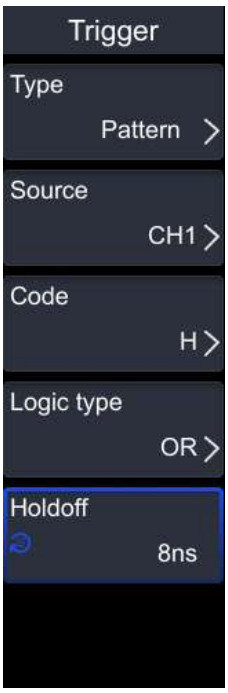
- Setup: When the establishment time is less than the set value, the oscilloscope triggers.
- Hold: When the hold time is less than the set value, the oscilloscope triggers.

7. **Width:**

Click on the menu label for **Width**, and set the width value directly by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

8.6.12 Pattern Trigger

Pattern triggering requires setting the logical values of each channel and the logical relationships between channels (AND, OR, NOT, etc.). When the logical relationship is met and the set time conditions are met, triggering occurs when the edges of any channel change.
The logical values of each channel can be set to: high (higher than the trigger level), low (lower than the trigger level), and none (unrelated).
Click on **[Trig Menu]** in the button area to open the trigger menu.



6. Type:

Click on the **Type** menu label and select " Pattern Trigger" to set the Pattern trigger. After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

7. Source:

Click on the **Source** menu tab and select CH1-CH4. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

8. Code:

Click on the **Code** menu tab and select: H, L, X, Rising, Falling, or Either.

9. Logic type:

Click on the **Logic type** menu tab to select: OR, AND.

10. Holdoff:

Please refer to the introduction in the section on Trigger Holdoff for specific settings related to Holdoff.

8.6.13 Nth Edge

Triggered on the Nth edge after the specified idle time. For example, in the waveform shown in the following figure, if it is necessary to trigger on the second

rising edge after a specified idle time (the time between adjacent rising edges), the idle time should be set to P<idle time<M. Where M is the time between the first rising edge and the previous rising edge, and P is the maximum time between the rising edges involved in counting.

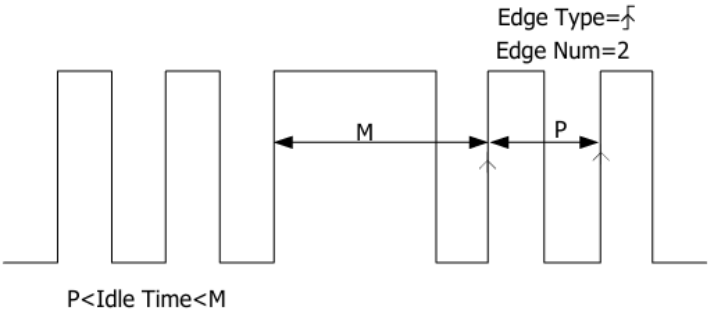
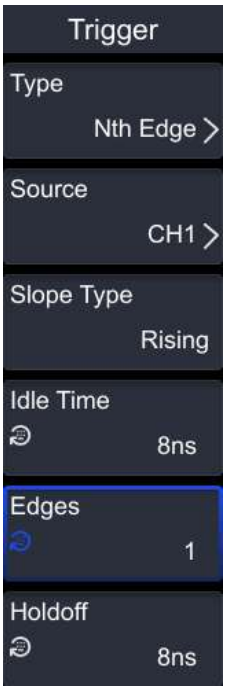


Figure 8.11 Schematic Diagram of Nth Edge Trigger

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. **Type:**

Click on the **Type** menu tab, select "Nth Edge", and set the trigger.
After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, and will change with the change of trigger settings.

2. **Source:**

Click on the **Source** menu tab and select CH1-CH4,EXT, Digital channels (D1.0-D1.3, D2.0-D2.3, D3.0-D3.3, D4.0-D4.3).
Digital channel: Only digital channels that have been inserted and enabled can be selected.

Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. **Slope Type:**

   Click on the **Slope Type** menu tab and select either rising edge or falling edge.
   Rising edge: Triggered at the rising edge of the input signal and when the voltage level meets the set triggering level.
   Falling edge: Triggered at the falling edge of the input signal and when the voltage level meets the set triggering level.

4. **Idle Time:**

   Click on the **Idle Time** menu tab to set the idle time before starting edge counting in the Nth edge trigger, and set the idle time by rotating the multifunction knob V0 or the pop-up numeric keypad.

5. **Edges:**

   Click on the **Edges** menu tab to set the specific value of "N" in the Nth edge trigger. Set the idle time by rotating the multifunction knob V0 or the pop-up numeric keypad, with a range of 1 to 65535.

6. **Triggering inhibition:**

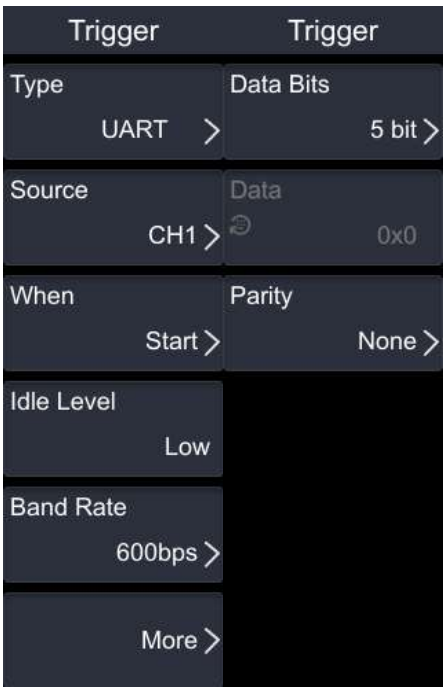
   Please refer to the introduction in the section on Trigger Holdoff for specific settings related to Holdoff.

8.6.14 UART Trigger(Option)

RS232 bus is a serial communication method used for data transmission between computers or between computers and terminals.
The RS232 serial protocol transmits a character as a frame of data, with a frame structure consisting of 1 bit start bit, 5-8 bits data bit, 1 bit check bit, and 1-2 bits stop bit. The format is shown in the following figure. This series of oscilloscopes can trigger when detecting frame start, frame end, data, verification errors, and errors of RS232 signals.

Figure 8.12 Schematic Diagram of UART Protocol

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "RS232 trigger" to set RS232 trigger.
After selecting the trigger type, the current trigger setting information (including trigger type, trigger signal source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **Source** menu tab and select CH1-CH4 or EXT. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. When:

Select the desired trigger condition in the **When** menu tab.

- Start: Triggered at the start position of the frame.
- Stop: Triggered at the end of the frame position.
- Data: Triggered when the data reception is completed and the received data matches the data set by the user.
- Check Error: Triggered when the data reception is completed and the verification result does not match the data set by the user.
- Error: Triggered when an error frame is detected.

4. Polarity:

Click on the **Polarity** menu tab to select the polarity for data transmission, which can be either positive or negative.

5. Baud Rate:

Click on the **Baud Rate** menu tab and select the preset baud rate from the pop-up sub options. Alternatively, click again to pop up the numeric keypad and directly enter the set baud rate.

6. Data Bits:

Click on the **Data Bits** menu tab and select the desired data width. The data width refers to the number of bits of data per frame. The data width can be selected as 5 bits, 6 bits, 7 bits, or 8 bits.

7. Data:

Click on the **Data** menu tab and set the data directly by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

8. Parity:

Click on the **Parity** menu tab, select polarity, and you can choose None, Odd, or Even.

8.6.15 LIN Trigger (Option)

This oscilloscope can be triggered on the synchronous field of LIN signals, as well as on designated identifiers, data, or frames.
The format of the LIN bus data frame is shown in the following figure.

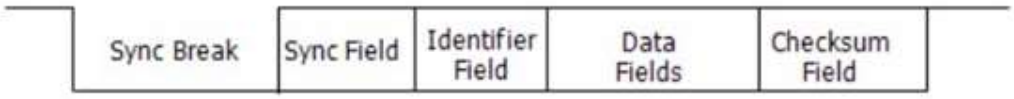
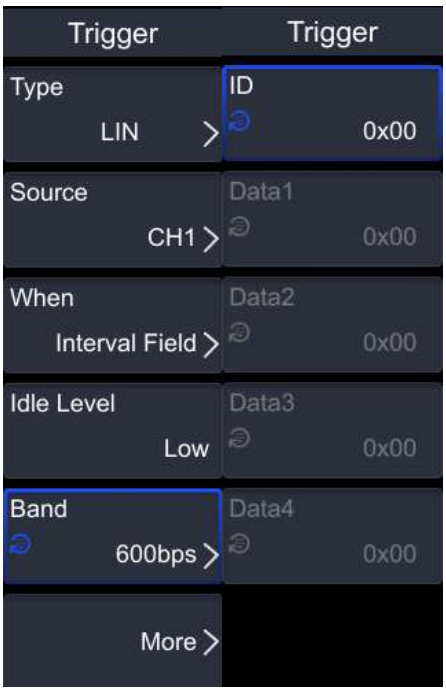


Figure 8.13 Data Frame Format of the LIN Bus

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "LIN trigger" to set the LIN trigger.
After selecting the trigger type, the current trigger setting information, including trigger type, trigger source, and trigger level, will be displayed at the top of the screen, and will change with the change of the trigger setting.

2. Source:

Click on the **Source** menu tab and select CH1-CH4 or EXT as the specified source. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. When:

Click on the **When** menu tab to select the desired trigger condition.

- Interval Field: Triggered by the edge after the end of LIN interval.
- Sync Field: Triggered upon completion of LIN sync field data reception.
- ID Field: triggered after receiving data from LINID field.
- Sync Code Error: Triggered when the LIN synchronization field data reception is completed but the synchronization field data is not equal to 0x55.
- Identifier: Triggered when the LINID field data is received and the ID data is equal to the user set ID.
- ID And Data: LIN data is received normally, and both ID and data are triggered by the user's set data.

4. Baud:

Click on the **Baud** menu tab and select the preset rate from the pop-up sub options.

5. ID:

Click on the **ID** menu tab and set the ID value by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

6. Data:

Click on the **Data** menu tab and set Data 1, Data 2, Data 3, and Data 4 by rotating the multifunctional knob V0 or the pop-up numeric keypad.

8.6.16 CAN Trigger (Option)

This oscilloscope can be triggered at the beginning of the CAN signal frame, at a specified type of frame (such as remote frame, data frame, etc.), or at a specified type of error frame. The format of the CAN bus data frame is shown in the following figure.

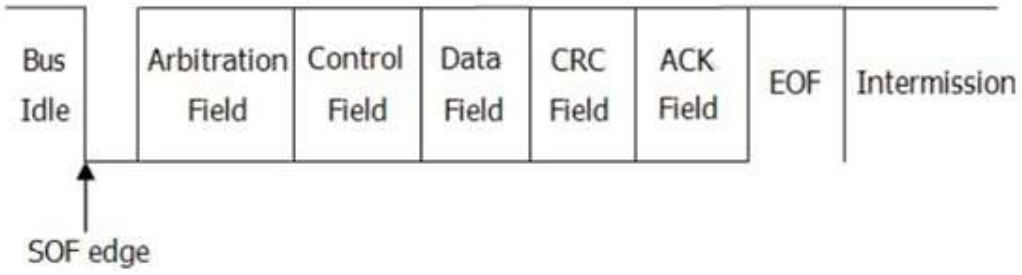
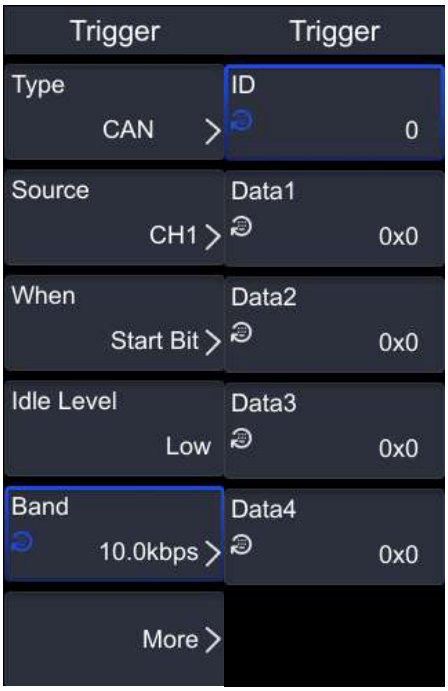


Figure 8.14 Data Frame Format of the CAN Bus

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. **Type:**

Click on the **Type** menu label and select "CAN trigger" to set the CAN trigger.
After selecting the trigger type, the current trigger setting information (including trigger type, trigger signal source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. **Source:**

Click on the **Source** menu tab and select CH1-CH4 or EXT as the specified source. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. **When:**

Click the **When** menu button to select the desired trigger condition.

- Start Bit: Triggered at the beginning of a data frame.
- Remote Frame ID: Triggered on the remote frame with the specified ID.
- Data Frame ID: Triggered on the data frame with the specified ID.
- Frame ID: Triggered on the frame with the specified ID.
- Frame ID and Data: Triggered on the data frame of the specified ID and the data frame of the specified data. After selecting the frame ID and data, click on the definition menu tab and select either data or ID.
- Error Frame: Triggered on the error frame.
- All Error: Triggered on all errors.
- Ack Error: Triggered on confirmation error.
- Overload Frame: Triggered on an overload frame.

4. **Baud:**

Click on the **Baud** menu tab and select the preset rate from the pop-up menu.

5. **ID:**

Click on the **ID** menu tab and set the ID value by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

6. **Data:**

Click on the **Data** menu tab and set Data 1, Data 2, Data 3, and Data 4 by rotating the multifunctional knob V0 or the pop-up numeric keypad.

8.6.17 SPI Trigger (Option)

Under the SPI trigger type, when the chip selection or timeout conditions are met, the oscilloscope triggers when it searches for the specified data. SPI trigger is a high-

speed, full duplex, synchronous communication bus. When using SPI trigger, it is necessary to specify the serial clock line (CLK) and serial data line (MISO).

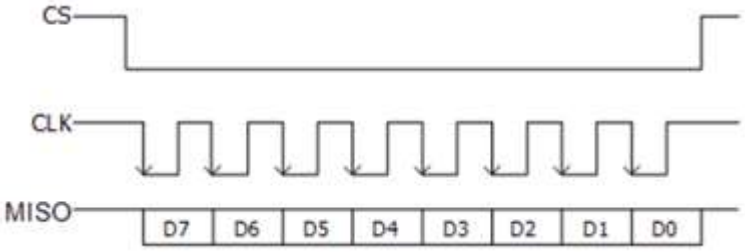
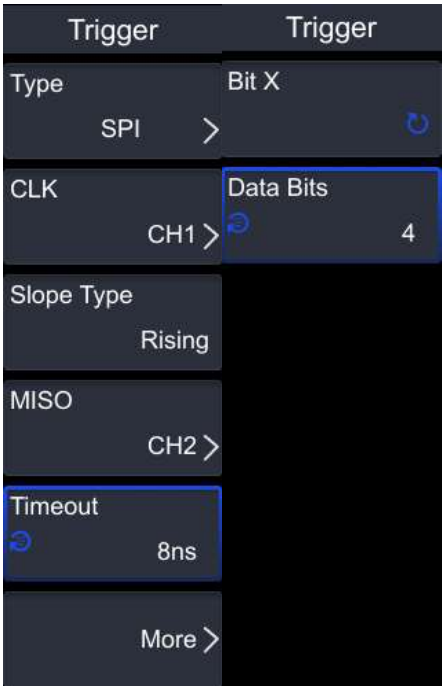


Figure 8.15 Sequential Chart of SPI Bus

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "SPI trigger" to set the SPI trigger. After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Source:

Click on the **CLK** and **MISO** menu labels and select CH1-CH4 or EXT to specify the signal source for the serial clock line (CLK) and serial data line (MISO), respectively. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. Slope type:

Click on the **Slope type** menu tab to select the desired clock edge type.

- Rising: Sampling MISO data at the rising edge of the clock.
- Falling: Sampling MISO data at the falling edge of the clock.

4. Timeout:

Click on the **Timeout** menu tab and enter the set value directly by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

5. Bit X:

Click on the **Bit X** menu tab to set the data bits that need to be operated on.

6. Data bits:

Click on the **Data bits** key and use the pop-up numeric keypad to set the number of digits in the serial data string. The number of digits in a string can be set to any integer between 4 and 32.

8.6.18 I2C Trigger (Option)

I2C (Inter Integrated Circuit Bus) integrated circuit bus, designed by NXP (formerly PHILIPS) company, is mainly used for master-slave communication between master controllers and slave devices. It is used in small data volume situations, has short transmission distance, and can only have one host at any time. I2C address addressing mode is divided into 7-bit addressing mode and 10 bit addressing mode.
7-bit addressing: In the 7-bit addressing process, the slave address starts transmission from the first byte after the start signal. The first 7 bits of this byte are the slave address, and the 8th bit is the read write bit, where 0 represents write and 1 represents read.
10 bit addressing: The 10bit and 7bit addressing of the I2C bus are compatible, allowing devices that use both 7bit and 10bit address modes on the same bus. When transmitting a 10bit address, the first byte is a special reserved address to indicate that the current transmission is a 10bit address.

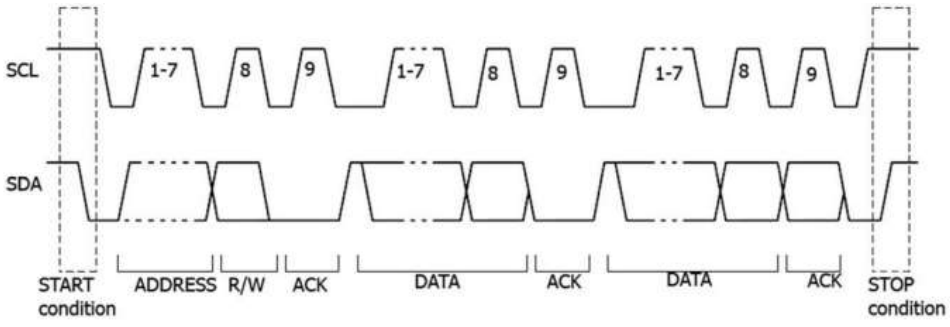
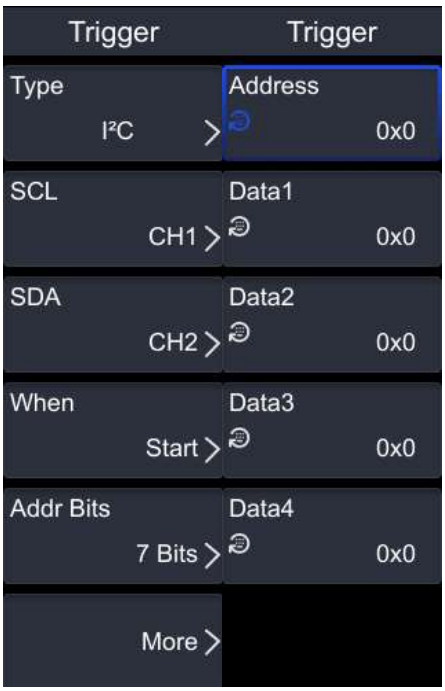


Figure 8.16 Schematic Diagram of I2C Protocol

Click on **[Trig Menu]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu label and select "I2C trigger" to set the I2C trigger.
After selecting the trigger type, the current trigger setting information (including trigger type, trigger source, and trigger level) will be displayed at the top of the screen, which changes with the trigger setting.

2. Clock source:

Click on the **SLK** menu tab and select CH1-CH4 or EXT as the specified clock source. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

3. Data source:

Click on the **SDA** menu tab and select CH1-CH4 or EXT as the specified data source. Only by selecting the channel that has been connected to the signal as the trigger source can stable triggering be obtained.

4. When:

Click on the **When** menu tab to select the desired trigger condition.

- Start: IIC start condition trigger detected. (Triggered when SCL is high and SDA data jumps from high level to low level.)
- Stop: IIC stop condition triggered detected. (Triggered when SCL is high and SDA data jumps from low to high.)
- Restart: Triggered when a new start condition appears before the stop condition.
- MissedAck: Triggered if SDA data is high during any SCL clock bit.

- Address: Triggered to search for the set address value, triggered on the read/write bit. After selecting the address as the trigger condition:
  - Click the read/write direction key and select write, read, or read/write. When "8 Bits" is selected for the address bit width, this setting is not available.
  - Click on the address bit width to select the desired address bit width. The address bit width can be selected from 7 bits, 8 bits, or 10 bits.
  - Click on the address key and set the address value triggered by I2C through the pop-up numeric keypad.
- Address and Data: Triggered when the IIC address is received and the data (4 bytes of data) is equal to the user set value. (The oscilloscope simultaneously searches for the set address value and data value, and triggers when both address and data meet the conditions.)

5. Address Bits:

Click on the **Addr Bits** width menu tab to select the desired address bit width, which can be selected as either 7Bits or 10Bits.

6. Address:

Click on the **Address** menu label and set the address directly by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

7. Data:

Click on the **Data** menu tab and set Data 1, Data 2, Data 3, and Data 4 by rotating the multifunctional knob V0 or the pop-up numeric keypad.

9 Math Operation

This series of oscilloscopes supports algebraic, FFT, logical, and functional operations between various analog channel waveforms
And digital filtering.
You can enter the **Math** menu through the following methods:

- Click **[Math]** in the button area to enter the mathematical operation menu.
- Click on the navigation icon **[Hantek]** in the upper left corner of the screen, select FFT, and enter the FFT menu.

This chapter includes:

9.1 Arithmetic Operations

The algebraic operations supported by this series of oscilloscopes include: A+B, A-B, A*B, A/B.

- Addition: Add the signals from source A and source B point by point and display the calculation results.
- Subtraction: Subtract the signals from source A and source B point by point and display the calculation results.
- Multiplication: Multiply the signals of source A and source B point by point and display the calculation results.
- Division: Divide the signals of source A and source B point by point and display the calculation results.

1. Operator:

Click on the **Operator** menu tab and select algebraic operations such as A+B addition, A-B subtraction, A * B multiplication, and A/B division.

2. Operation:

Click on the **Operation** menu tab and select on or off.

3. Source A, Source B:

After clicking on **Source A** or **Source B**, a menu will appear where you can select the source as CH1, CH2, CH3, CH4, or REF reference waveform, respectively. Channels in closed state cannot be selected.

4. Offset:

Click on the **Offset** menu label and set the offset by selecting the multi-functional knob V0 or directly setting the vertical offset for the calculation result display window using the pop-up numeric keyboard.

5. Auto Setting:

Click the **AutoSetting** button, and the instrument will automatically adjust the vertical gear and offset of the calculation results to the optimal value based on the current configuration, for the convenience of users' observation.

6. Scale:

Click on the **Scale** menu label, set the gear by selecting the multi-functional knob V0 or directly set the vertical gear for the calculation result display window using the pop-up numeric keyboard.

7. Invert:

Click on the **Invert** menu tab to select whether to turn on or off the invert function, which is used to display waveform invert in the calculation result display window.

8. Vertical expansion:

Click on the **Expand** menu tab and select either GND or center.

- GND: When changing the vertical gear, the mathematical operation waveform will expand or compress around the waveform zero point position.
- center: When changing the vertical gear, the mathematical operation waveform will expand or compress around the center of the screen.

9. Label:

Click on the **Label** menu label and set it to display waveform labels in the calculation result display window. Please refer to the Label Setting section.

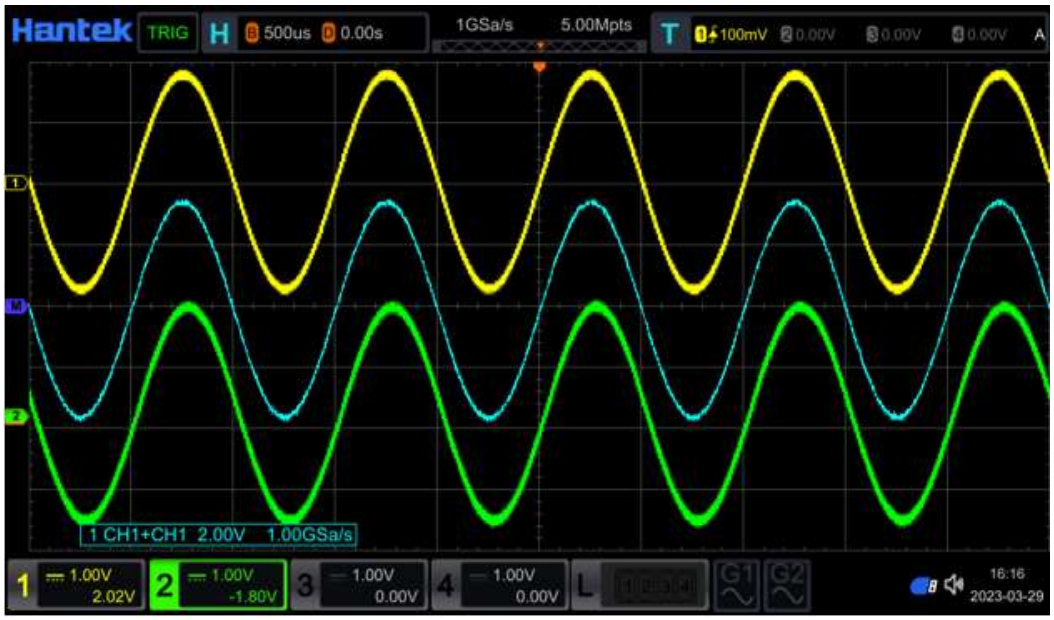
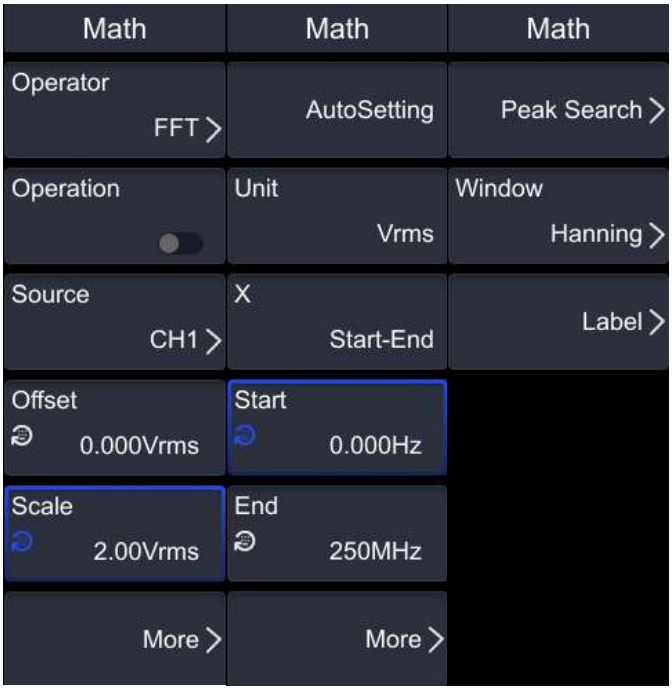


Figure 9.1 Addition

9.2 FFT



1. Operator:

Click on the **Operator** menu tab and select FFT operation.

2. Source:

Click on the dropdown box of the information **Source** item to select the information source as CH1, CH2, CH3, or CH4. Channels in closed state cannot be selected.

3. **Offset:**

Click on the **Offset** menu label and set the offset by selecting the multi-functional knob V0 or directly setting the vertical offset for the calculation result display window using the pop-up numeric keyboard.

4. **Scale:**

Click on the **Scale** menu label, set the gear by selecting the multi-functional knob V0 or directly set the vertical gear for the calculation result display window using the pop-up numeric keyboard.

5. **Auto settings:**

Click the **AutoSetting** button, and the instrument will automatically adjust the vertical gear and offset of the calculation results to the optimal value based on the current configuration, for the convenience of users' observation.

6. **Unit:**

Click on the **Unit** menu tab to select dBm/dBV or Vrms.

7. **X frequency range:**

In **X**, select the mode to determine the frequency range and set the frequency range.

- Span Center: The frequency range refers to the screen width. Divide the frequency range by 10 to obtain the frequency of each grid. Click on the input box of the center frequency item to set the frequency of the frequency domain waveform corresponding to the horizontal center of the screen. The minimum value range is 5Hz~5GHz, and the default value is 5MHz. Click on the input box of the frequency range item to set the frequency range of the frequency domain waveform, with a value range of 10Hz~5GHz and a default value of 10MHz.
- Start End: The starting frequency refers to the frequency on the left side of the screen. Click on the input box of the starting frequency item to set the starting frequency of the frequency domain waveform. The value range is from 0Hz to the value of the ending frequency -10Hz, with a default value of 0Hz. The termination frequency refers to the frequency on the right side of the screen. Click on the input box of the termination frequency item to set the termination frequency of the frequency domain waveform. The value range is the starting frequency value+10Hz~5GHz, and the default value is 10MHz.

8. **Peak Search:**

Click on the **Peak Search** menu tab to enter the peak search settings menu.

- Peak search: turned on or off
- Peak number: Click on the input box for the **Peak Number** item and set the peak number using the pop-up numeric keypad. The value range is 1 to 15, with a

default of 5.

- Threshold: Click on the input box of the **Threshold** item and set the peak threshold through the pop-up numeric keyboard. The range of threshold values is related to the current FFT gear and offset.
- Excursion: Click on the input box of the **Excursion** item to set the peak offset threshold. The minimum value of the offset threshold is 0, in dB.
- Table Order: Click to select the **Table Order** to be sorted by peak size or frequency size. Sort by peak size by default.
- Export: the interface jumps to the Save Settings menu, selects the file type, sets the file name, sets the window function, and saves.

9. Window:

The window function can effectively reduce the spectral leakage effect. This series of oscilloscopes provide the following table, which shows five kinds of FFT window function. Each kind of window function has different characteristics and waveforms suitable for measurement. The selection needs to be based on the measured waveform and its characteristics. Click the menu of **Window** function item to select.

Table 9.1 Window Function

Window Function	Characteristics	Waveforms Applicable to the Window Function
Rectangular	Best frequency resolution. Poorest amplitude resolution. Similar to the situation when no window is applied.	Transient or short pulse, the signal levels before and after the multiplication are basically the same. Sine waveforms with the same amplitudes and rather similar frequencies. Wide band random noise with relatively slow change of waveform spectrum.
Hanning	Better frequency resolution and poorer amplitude resolution compared with Rectangular.	Sine, periodic, and narrow band random noise.
Hamming	A little bit better frequency resolution than Hanning.	Transient or short pulse, the signal levels before and after the multiplication are rather different.
Blackman-Harris	Best amplitude resolution. Poorest frequency resolution.	Single frequency signal, searching for higher order harmonics.

Triangle	Better frequency resolution.	Measure the narrow band signal and that has strong noise interference.
Flattop	Measure the signals accurately.	Measure the signal that has no accurate reference and requires an accurate measurement.

10. Label:

Click on the **Label** menu label and set it to display waveform labels in the calculation result display window. Please refer to the label section.

9.3 Logic Operation

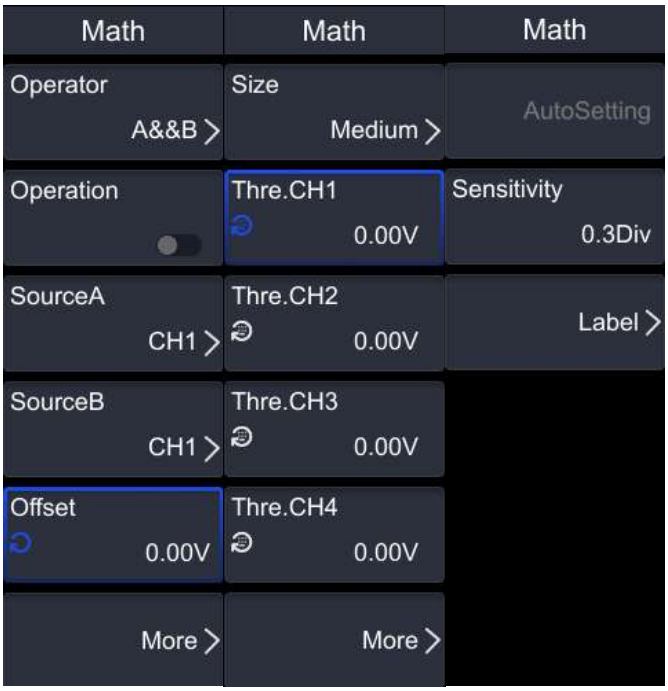
Click on the **[MATH]** menu, and the logical operations include: A&&B (AND), A | | B (OR), A ^ B (XOR), ! A (NOT).

- A&&B: Perform logical AND operations on the waveform of the specified source point by point and display the results. When the voltage value of the source channel is greater than the threshold set by the corresponding channel during operation, it is determined as a logical "1", otherwise it is determined as a logical "0". The logical AND operation of two binary bits.
- A | | B: Perform logical OR operations on the waveform of the specified source point by point and display the results. When the voltage value of the source channel is greater than the threshold set by the corresponding channel during operation, it is determined as a logical "1", otherwise it is determined as a logical "0". The logical OR operation of two binary bits.
- A ^ B: Perform a logical XOR operation on the waveform of the specified source point by point and display the results. When the voltage value of the source channel is greater than the threshold set by the corresponding channel during operation, it is determined as a logical "1", otherwise it is determined as a logical "0". The logical XOR operation of two binary bits.
- ! A: Perform logical 'not' operations on the waveform of the specified source point by point and display the results. When the voltage value of the source channel is greater than the threshold set by the corresponding channel during operation, it is determined as a logical "1", otherwise it is determined as a logical "0". The logical 'not' operation of a binary bit.

Table 9.2 Logic Operation

A	B	A&&B	A||B	A^B	!A

0	0	0	0	0	1
0	1	0	1	1	1
1	0	0	1	1	0
1	1	1	1	0	0



1. Operator:

Click on the **Operator** menu tab and select logical operation.

2. Source A, Source B:

After clicking on **Source A** or **Source B**, a menu will appear where you can select the source as CH1, CH2, CH3, CH4, or Ref reference waveform, respectively. Channels in closed state cannot be selected.

3. Offset:

Click on the **Offset** menu label and set the offset by selecting the multi-functional knob V0 or directly setting the vertical offset for the calculation result display window using the pop-up numeric keyboard.

4. Size:

Click on the **Size** menu label and select the "Small", "Medium", or "Large" display mode.

5. CH1 threshold, CH2 threshold, CH3 threshold, CH4 threshold:

Set the threshold corresponding to the source channel, click on the

Thre.CH1(Thre.CH2\ Thre.CH3\Thre.CH4) input box, and the setting method can be achieved by adjusting the multi-functional knob or directly setting the value on the numeric keyboard.

6. Auto Setting:

Click the **AutoSetting** button, and the instrument will automatically adjust the vertical gear and offset of the calculation results to the optimal value based on the current configuration, for the convenience of users' observation.

7. Sensitivity:

Set the **Sensitivity** of converting the analog signal on the signal source into a digital signal. Click on the sensitivity input box, and the setting method can be achieved by adjusting the multi-functional knob or directly setting the value on the digital keyboard. Please refer to the parameter setting method for specific operations.

8. Label:

Used to display waveform labels in the calculation result display window. Display labels, label libraries, and labels. Please refer to the Label Setting section for the setting method.

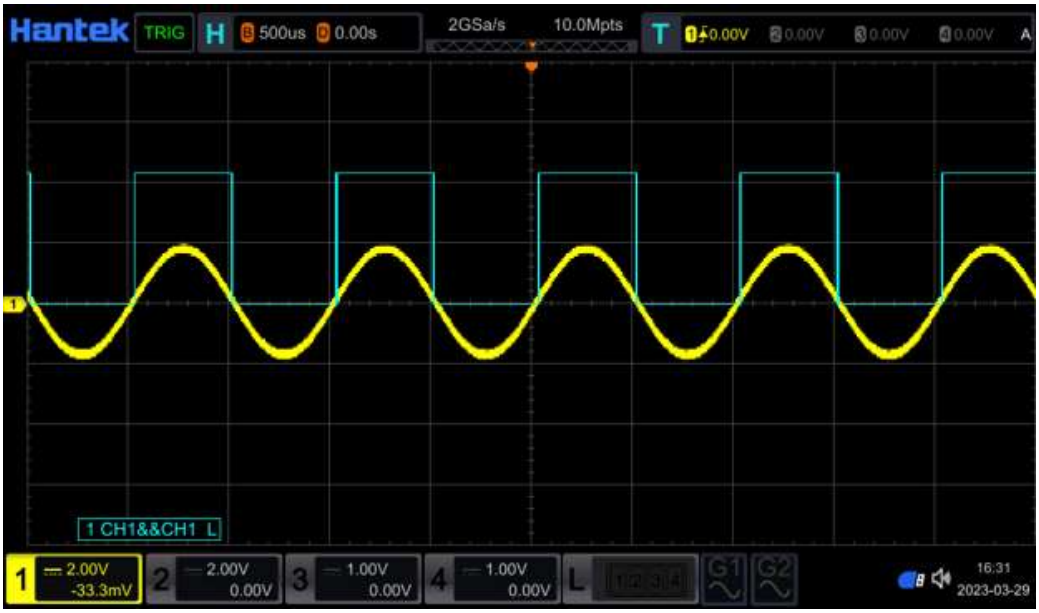


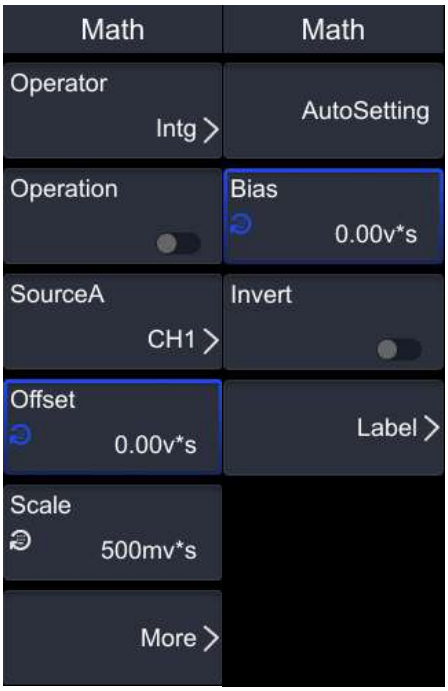
Figure 9.2 AND Operation

9.4 Function operation

In the **Math** Operation menu, select from the drop-down menu of the **operator** item. The functions supported by this series of oscilloscopes include: Intg (integral), Diff (differential), Sqrt (square root), Lg (logarithm based on 10), Ln (natural logarithm), Exp (exponent), Abs (absolute value), AX+B (linear function).

Intg: Calculate the integral of a specified source. For example, integration can be used to calculate the energy of a pulse or measure the area under a waveform.

- Diff: Calculate the discrete time derivative of a specified source. For example, differentiation can be used to calculate the instantaneous slope of a waveform.
- Sqrt: Calculate the square root of the specified source waveform point by point and display the results.
- Lg: Calculate the base 10 logarithm of the specified source waveform point by point and display the results.
- Ln: calculate the natural logarithm of the specified source waveform point by point and display the results.
- Exp: Calculate the index of the specified signal source waveform point by point and display the results.
- Abs: Take the absolute value of the waveform of the specified signal source and display the result.
- AX+B: calculate the linear function of the specified source waveform point by point and display the results.



1. Operator:

Click on the **Operator** menu tab and select logical operation.

2. Source A:

Click on the menu of **Source A**, and you can select channels with source CH1, CH2, CH3, CH4, or Ref reference waveform closed status that cannot be selected.

3. Offset:

Click on the **Offset** menu label and set the offset by selecting the multi-functional knob V0 or directly setting the vertical offset for the calculation result display window using the pop-up numeric keyboard.

4. Scale:

Click on the **Scale** label, set the gear by selecting the multi-functional knob V0 or directly set the vertical gear for the calculation result display window using the pop-up numeric keyboard.

5. Auto Setting:

Click the **AutoSetting** button, and the instrument will automatically adjust the vertical gear and offset of the calculation results to the optimal value based on the current configuration, for the convenience of users' observation.

6. Integral Offset:

Click on the **Integral Offset** menu tab and set the gear by selecting the multi-functional knob V0 or directly set the integral offset using the pop-up numeric keypad.

7. Invert:

Click on the **Invert** menu tab to select whether to turn on or off the invert function, which is used to display waveform invert in the calculation result display window.

8. Label:

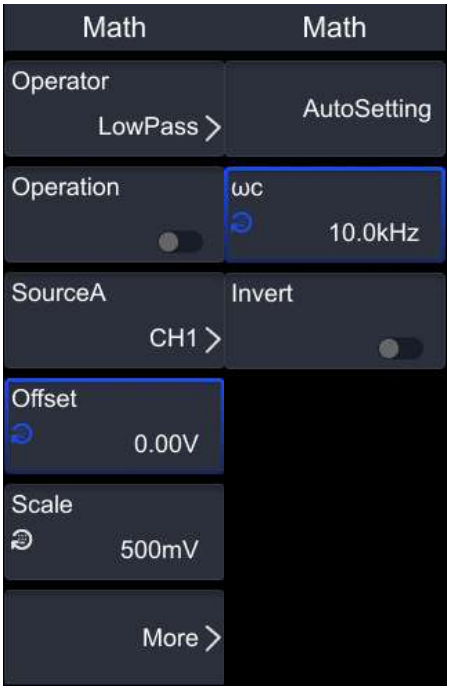
Click on the Label menu label and set it to display waveform labels in the calculation result display window. Please refer to the Label Setting section.

Figure 9.3 Lg Operation

9.5 Digital Filtering

In the **Math** Operations menu, click on the **Operator** to select from the drop-down menu. The digital filtering supported by this series of oscilloscopes includes: LowPass, HighPass, BandPass, and BandStop.

- LowPass: Only signals with frequencies lower than the current frequency limit are allowed to pass through.
- HighPass: Only signals with frequencies higher than the current frequency lower limit are allowed to pass through.
- BandPass: Only signals with frequencies higher than the current frequency lower limit and lower than the current frequency upper limit are allowed to pass through.
- BandStop: Only signals with frequencies lower than the current frequency lower limit or signals higher than the current frequency upper limit are allowed to pass through.



1. Operator:

Click on the **Operator** menu tab and select logical operation.

2. Source A:

Click on the menu of **Source A** to select a reference waveform with source CH1, CH2, CH3, CH4, or Ref. Channels in closed state cannot be selected.

3. Offset:

Click on the **Offset** menu label and set the offset by selecting the multi-functional knob V0 or directly setting the vertical offset for the calculation result display window using

the pop-up numeric keyboard.

4. Scale:

Click on the **Scale** menu label, set the gear by selecting the multi-functional knob V0 or directly set the vertical gear for the calculation result display window using the pop-up numeric keyboard.

5. Auto Setting:

Click the **AutoSetting** button, and the instrument will automatically adjust the vertical gear and offset of the calculation results to the optimal value based on the current configuration, for the convenience of users' observation.

6. Wc:

Click on the **wc** menu tab and set it directly by selecting the multi-functional knob V0 or the pop-up numeric keyboard.

7. Invert:

Click on the **Invert** menu tab to select whether to turn on or off the invert function, which is used to display waveform invert in the calculation result display window.

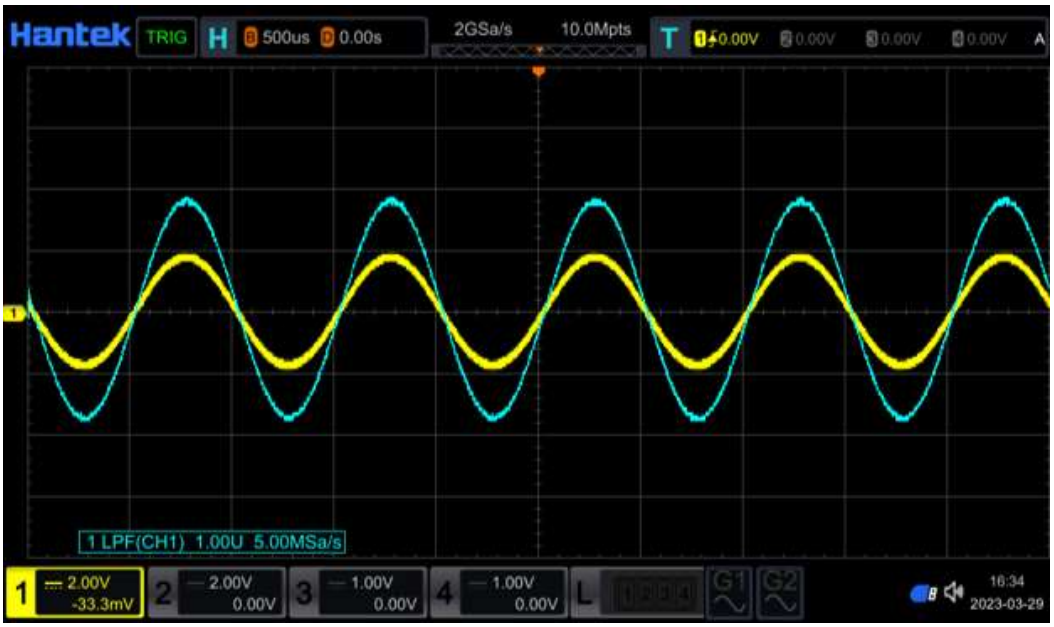
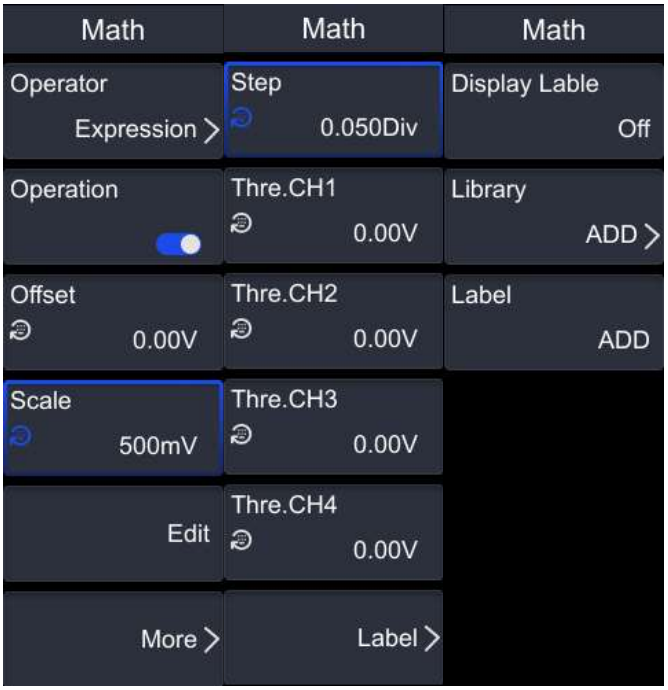


Figure 9.4 LowPass Operation

9.6 Expression

By selecting expressions from operators and editing working hours through the keyboard, multiple operators and sources can be freely combined to achieve more complex mathematical operations.



1. Operator:

Click on the **Operator** menu tab and select logical operation.

2. Offset:

Click on the **Offset** menu label and set the offset by selecting the multi-functional knob V0 or directly setting the vertical offset for the calculation result display window using the pop-up numeric keyboard.

3. Scale:

Click on the **Scale** menu label, set the gear by selecting the multi-functional knob V0 or directly set the vertical gear for the calculation result display window using the pop-up numeric keyboard.

4. Edit:

Click on the **Edit** menu tab, and the expression editing keyboard will pop up. Use the operation buttons to select the desired expression for calculation.

5. Step:

Click on the **Step** menu label, and the setting method can be set by adjusting the multifunctional knob or directly setting the single step value on the numeric keyboard.

6. CH1 threshold, CH2 threshold, CH3 threshold, CH4 threshold:

Set the threshold corresponding to the source channel, click on the **Thre.CH1(Thre.CH2\ Thre.CH3\Thre.CH4)** input box, and the setting method can be achieved by adjusting the multi-functional knob or directly setting the value on the numeric keyboard.

7. Label:

Used to display waveform labels in the calculation result display window. Display labels, label libraries, and labels. Please refer to the Label Setting section for the setting method.

10 Measure

The voltage time coordinate graph displayed by the oscilloscope can be used to measure the displayed waveform. There are various methods for measuring, including screen grid scale, cursor, or automatic measurement.
The menu can be accessed through the following methods:

- Click on the button area **[Meas]** to enter the measurement menu.
- Click on the navigation icon **[Hantek]** in the upper left corner of the screen, select **Measure**, and enter the measurement menu.

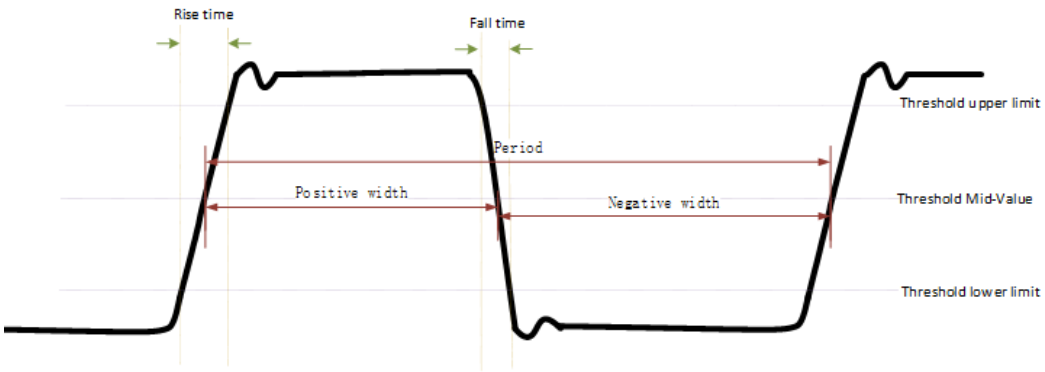
This chapter includes:

10.1 Square scale measurement

Grid scale: This method can be used to quickly and intuitively estimate the frequency and voltage amplitude of the waveform, and simple measurements can be made through the division of the grid chart and the scale coefficient.
For example, simple measurements can be made by calculating the relevant primary and secondary scale divisions and multiplying them by the proportional coefficient. If it is calculated that there are 6 main vertical scale divisions between the maximum and minimum values of the waveform, and the scaling coefficient is known to be 50mV/division, the peak to peak voltage can be calculated as follows:
6 divisions × 50mV/division=300mV

10.2 Measurement Parameter

10.2.1 Time Parameters



1. Period: defined as the time between the middle threshold points of two consecutive, like-polarity edges.
2. Frequency: defined as the reciprocal of period.
3. Rise Time: indicates the time for the signal amplitude to rise from the threshold lower limit to the threshold upper limit.
4. Fall Time: indicates the time for the signal amplitude to rise from the threshold upper limit to the threshold lower limit.
5. +Width: indicates the time between the threshold middle value of a rising edge to the threshold middle value of the next falling edge.
6. -Width: indicates the time between the threshold middle value of a falling edge to the threshold middle value of the next rising edge.
7. +Duty: indicates the ratio of the positive pulse width to the period.

$$+\text{Duty} = \frac{\text{positive pulse}}{\text{period}} \times 100\%$$

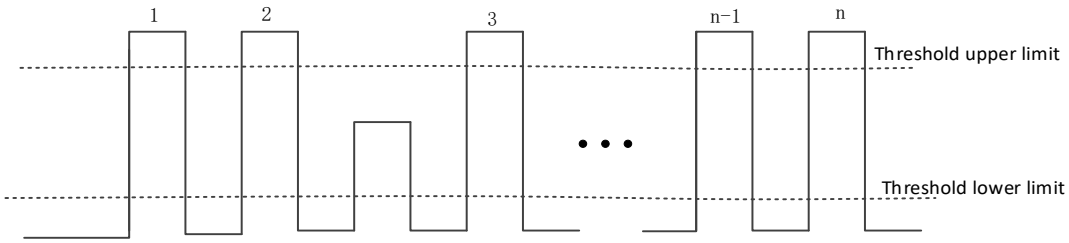
8. -Duty: indicates the ratio of the negative pulse width to the period.

$$-\text{Duty} = \frac{\text{negative pulse}}{\text{period}} \times 100\%$$

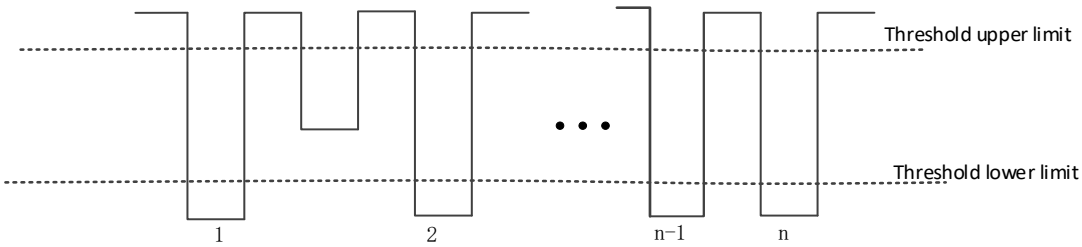
9. Bandwidth: The time from the first edge of the data source to the last edge of the data source.
10. MaxTime: indicates the time that corresponds to the maximum value of the waveform (Vmax).
11. MinTime: indicates the time that corresponds to the minimum value of the waveform (Vmin).

10.2.2 Count value parameters

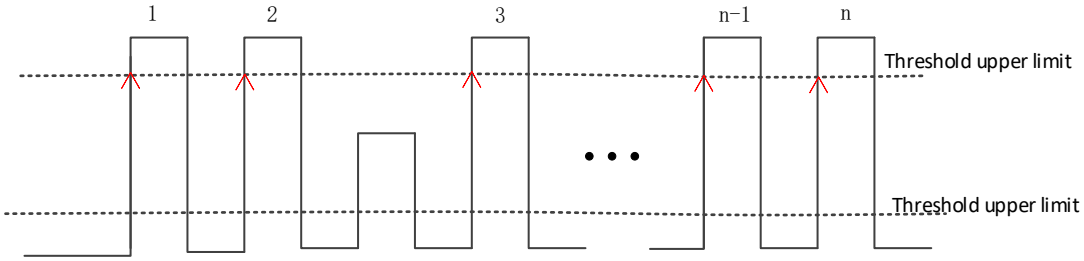
1. Positive Pulse Count [+ Pulse Count]: The number of positive pulses rising from be-low the lower threshold to above the upper threshold.



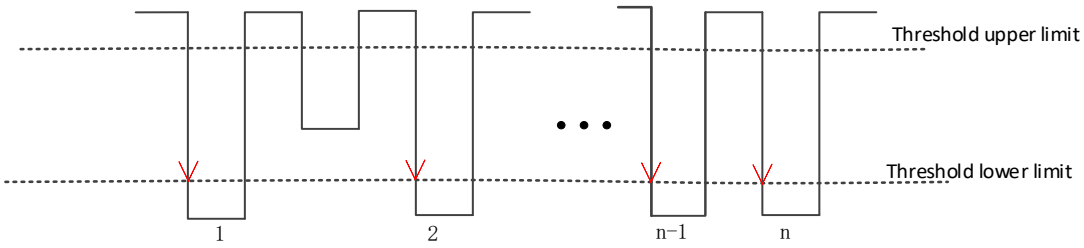
2. Negative pulse number [- Pulse Count]: the number of negative pulses from above the upper threshold to below the lower threshold.



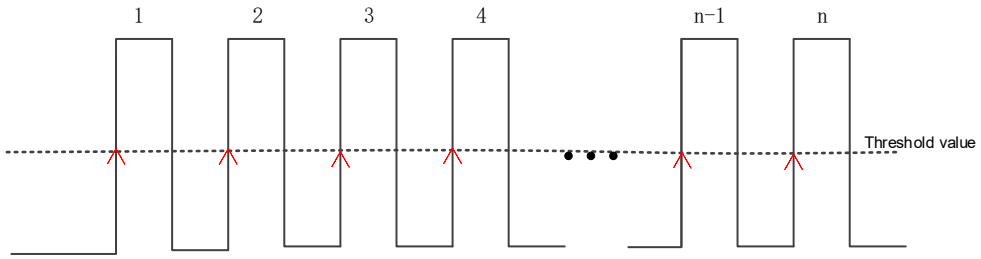
3. Rise Pulse Count: The number of rising edges rising from below the lower threshold to above the upper threshold.



4. Fall Pulse Count: The number of falling edges from the upper threshold to the lower threshold.

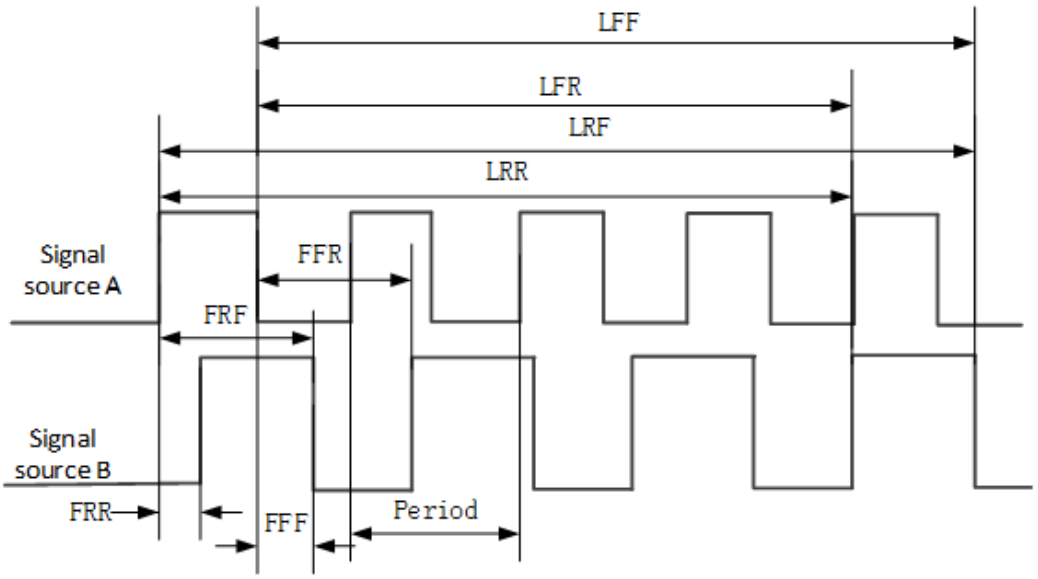


5. Trigger Count: The number of rising (or falling) edges from the threshold.



Note: The above measurement items are only applicable to analog channels. The default values for the upper and lower thresholds are 90% and 10%, respectively.

10.2.3 Delay and Phase



Data source A and data source B, namely data source A and data source B in the measurement settings menu, can be any channel in CH1-CH4.

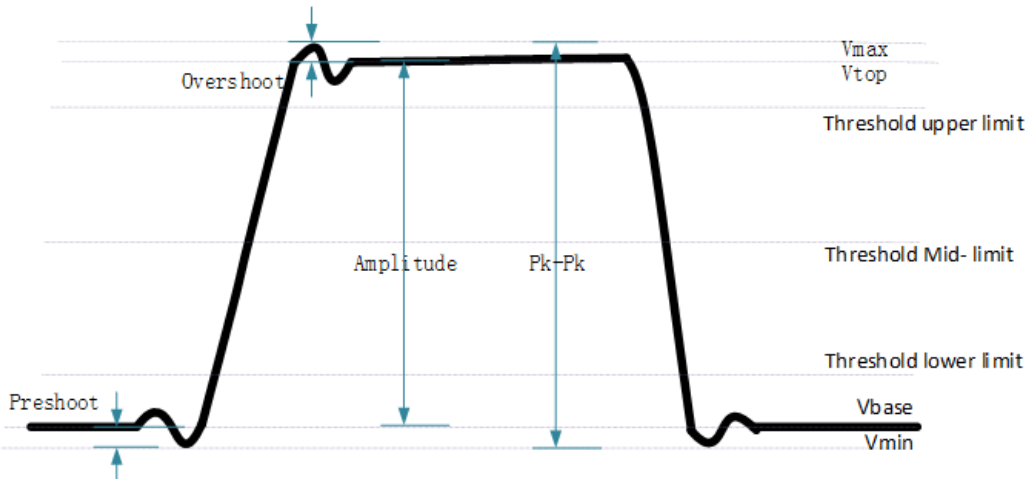
1. FRR [First Delay Rise Rise Rise]: ↑ A -> ↑ B, the time difference between the first rising edge of data source A and data source B.
2. FFF [First Delay Fall Fall]: ↓ A -> ↓ B, the time difference between the first falling edge of data source A and data source B.
3. FRF [First Delay Rise Fall]: ↑ A -> ↓ B, the time difference between the first rising edge of data source A and the first falling edge of data source B.
4. FFR [First Delay Fall Rise]: ↓ A -> ↑ B, the time difference between the first falling edge of data source A and the first rising falling edge of data source B.
5. LRR [Last Delay Rise Rise Rise]: ↑ A -> ↑ B, the time between the first rising edge of data source A and the last rising edge of data source B.
6. LRF [Last Delay Rise Fall]: ↑ A -> ↓ B, the time between the first rising edge of data source A and the last falling edge of data source B.
7. LFR [Last Delay Fall Rise]: ↓ A -> ↑ B, the time between the first falling edge of data source A and the last rising edge of data source B.
8. LFF [Last Delay Fall Fall]: ↓ A -> ↓ B, the time between the first falling edge of data source A and the last falling edge of data source B.
9. Positive phase difference [R-PhaseAB]: The phase difference on the rising edge between channels.
10. Negative phase difference [F-PhaseAB]: The phase difference at the falling edge between channels.
11. Phase [Phase Rise Time]: ↑ A ->B, the phase difference calculated based on the

period of "delay 1->2" and data source 1, expressed in degrees. The calculation formula for phase is:

$$\text{Phase} = \frac{Delay}{period1} \times 360^{\circ}$$

Among them, Phase represents "Phase ↑ A ->B" or "Phase ↓ A ->B".
Delay represents "Delayed FRR" or "Delayed FFF".
Period1 represents the period of digital data source 1.

10.2.4 Voltage Parameters



1. Vmax: The voltage value from the highest point of the waveform to GND (ground).
2. Minimum value [Vmin]: The voltage value from the lowest point of the waveform to GND (ground).
3. Pk Pk: The voltage value from the highest point to the lowest point of the waveform.

   Double peak = Vmax - Vmin
4. Vtop: The voltage value from the flat top of the waveform to GND (ground).
5. Vbase: The voltage value from the flat bottom of the waveform to GND (ground).
6. Vamp: The voltage value from the top to the bottom of the waveform.

   Amplitude = Vtop - Vbase
7. Vmid: The actual voltage value corresponding to the intermediate value of the measurement threshold.
8. Average: the arithmetic mean value of the whole waveform or gate area.
9. Cycle Average: arithmetic mean value of the first cycle of the waveform.

$$\text{Average} = \frac{\sum x_i}{n}$$

Among them, xi is the measurement result of the i-th point, and n is the number of measured points.

10. Vrms: The root mean square value on the entire waveform or gating region.
11. Per Vrms: The root mean square value within a period.

    Among them, xi is the measurement result of the i-th point, and n is the number of measured points.

12. Fall Pre shot: The ratio of the difference between the minimum and bottom values of the falling edge of a waveform and its amplitude.
13. Fall Overshoot: The ratio of the difference between the maximum value of the falling edge of the waveform and the top value to its amplitude.
14. Vrpr: The ratio of the difference between the minimum value of the falling edge of the waveform and the bottom value to the amplitude.
15. Vfov: The ratio of the difference between the maximum value of the falling edge and the top value of the waveform to the amplitude.
16. L@T: Voltage value at the trigger point.

10.2.5 Other parameters

Positive Slew Rate: On the rising edge, first calculate the difference between the high value and the low value, then use the difference to divide the corresponding time value to obtain the positive slew rate.
Negative Slew Rate: On the falling edge, first calculate the difference between the low value and the high value, then use the difference to divide the corresponding time value to obtain the negative slew rate.

10.3 Measurement Settings

10.3.1 Add

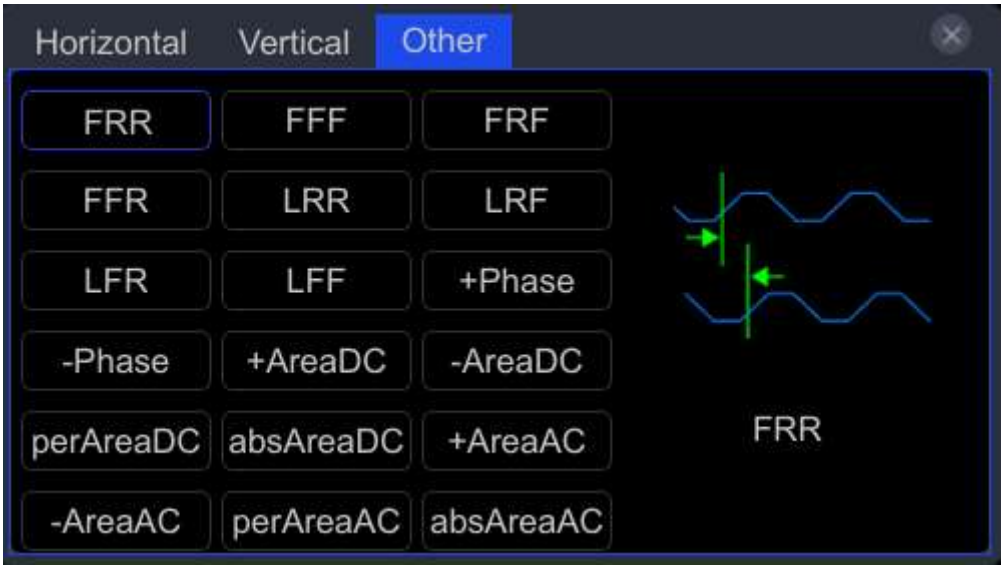
Click the **[Meas]** button to enter the measurement settings menu.

1. Measurement Category:

Click on **[Meas]** -> **Add** -> **Category** to select three measurement categories: Horizontal, Vertical, and Other. At the same time, the specific measurement parameters in the measurement categories are displayed on the screen, as shown in the following figure. You can also turn the multi-functional knob V0 in the measurement category area to select horizontal, vertical, or other measurement categories, and then press to switch to that measurement category.
Horizontal: Freq, Period, RiseT, FallT, PosPW, NegPW, PDuty, NDuty, BWidth, MaxTime, MinTime, +Edges, -Edges, +Pulses, -Pulses, TrigCnt, +slope, -slope.

Vertical: VMean, VMax, VMin, PkPk, VTop, VMid, VBase, VAmp, VRms, Vovr, Vper, PVRms, PVMeas, Vfov, Vrpr,L@T.

Other: FRR, FFF, FRF, FFR, LRR, LRF, LFR, LFF, +Phase, -Phase, +AreaDC, -AreaDC, perAreaDC, absAreaDC, +AreaAC, -AreaAC, perAreaAC,absAreaAC.



2. Source A, B:

- Horizontal and vertical parameter measurement

Press **[Meas]** -> **Add** -> **Source A**, rotate the multi-functional knob to select the channel to be measured, and press the knob to select it. You can also continuously press the source A button to switch to the current measurement source, or use the touch screen function to select using touch gestures. Optional channels include CH1-CH4, Math1-Math4, D1.0-D1.3, D2.0-D2.3, D3.0-D3.3, D4.0-D4.3, Ref.

- Measurement of other parameters

Press **[Meas]** -> **Add** -> **Source A** and **Source B** to set the two source channels for the current measurement type. Optional channels include CH1-CH4, Math1-Math4, D1.0-D1.3, D2.0-D2.3, D3.0-D3.3, D4.0-D4.3, Ref.

3. Remove:

Press **[Meas]** -> **Add** -> **Remove** to enter the delete detailed menu settings.

10.3.2 Remove

Press **[Meas]** -> **Remove** to enter the delete settings menu.

1. Remove

Click on the **Remove** menu tab to delete the last measurement item added. Note that pressing this soft key once will only delete one measurement item. When deleting a measurement item, the measurement result at the bottom of the screen will shift to the left and display one item.

2. Remove All

Click to **Remove All** menu labels and delete all displayed measurement items with one

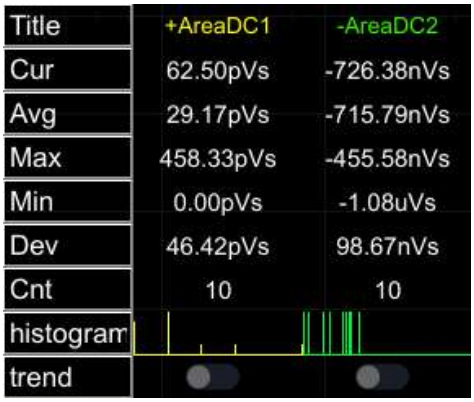
click.

3. Statistic

Click on the **Statistic** menu tab to open the statistics function. For a detailed introduction, please refer to the Statistics section.

10.3.3 Statistic

Calculate and display the current value (Cur), average value (Avg), maximum value (Max), minimum value (Min), standard deviation (Dev), count (Cnt), and histogram of the measurement results, as shown in the following figure.
After turning on the statistical histogram, the bar histogram at the bottom of the statistical area displays the number of measurement results counted in the form of columns, allowing users to visually view the distribution of the measured values of this parameter. The color of the histogram matches the color of the specified source.



1. Statistic

Click on the Statistic menu tab to open the statistics function.

2. Reset Stat.

Click on the **Reset Stat.** menu tab to clear historical data and perform statistics again.

3. Count

Click on the **Count** menu tab, rotate the multi-functional knob, or set the number of counts through the pop-up numeric keypad. The value range is 2 to 100000, with a default of 10.

10.3.4 All Measure

All measurements can display all the parameters of the current measurement source on the screen. Press **[Meas]** -> **All Measure**, select to turn off all measurement functions (OFF) or select the channels CH1-CH4 that need to be measured. If CH1 channel is

selected, all measurement parameter data of CH1 channel will be displayed on the screen.

Note:

The deletion operation will not clear the measurement results of all measurement functions.

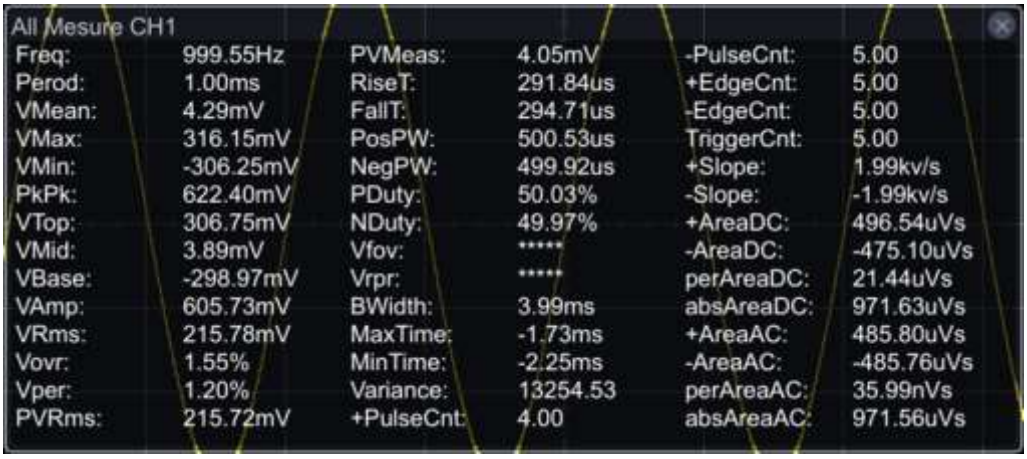


Figure 10.1 All Measure

10.4 Quick Measurement after AUTO

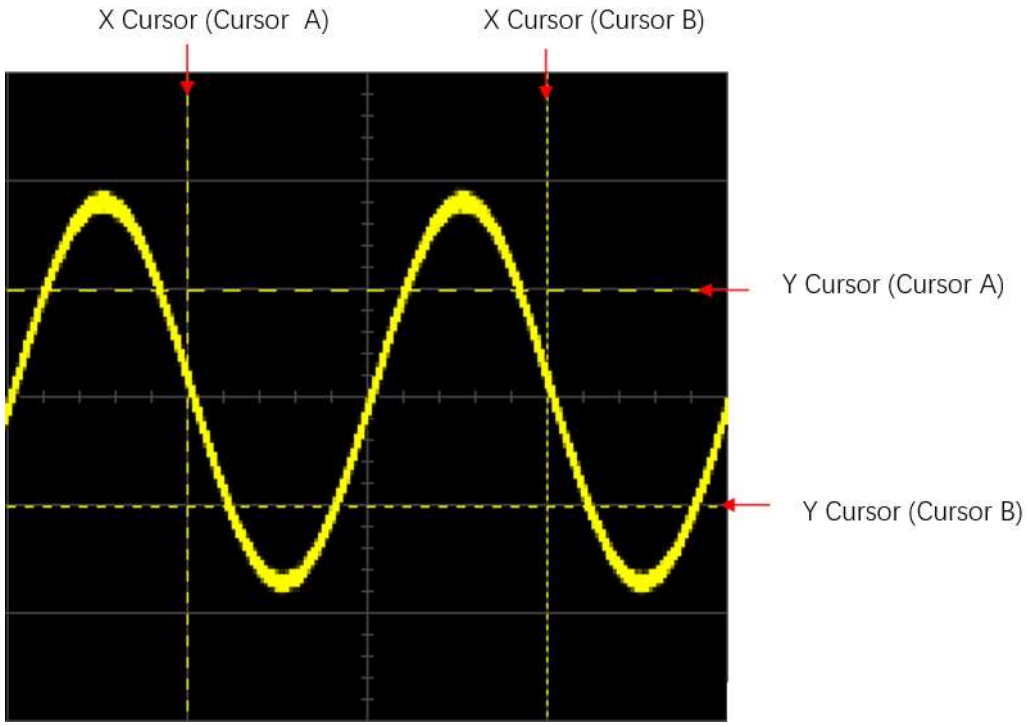
After connecting the oscilloscope correctly, input a valid signal, press the **[Auto Scale]** button to automatically set the waveform and open the following function menu:

- Single cycle: Set the screen to automatically display signals for a single cycle. Simultaneously measure the "time parameter" and "voltage parameter" of the current data source in a single cycle, and the measurement results are displayed at the bottom of the screen.
- Multi cycle: Set the screen to automatically display signals for multiple cycles. Simultaneously perform multi cycle "time parameter" and "voltage parameter" measurements on the current data source, and the measurement results are displayed at the bottom of the screen.
- Automatic range: Select the scale automatically set by the system: "Horizontal vertical gear", "Horizontal gear", or "Vertical gear".
- Data source: Set the channels to be automatically set: "Display only", "All". Display only: After the system performs automatic settings, only the waveform of the opened channel with signal input is displayed; All: After the system performs automatic settings, the waveforms of all channels with signal inputs are displayed.

10.5 Cursor Measurement

The cursor can be used to measure the X-axis value (such as time) and Y-axis value

(such as voltage) of the selected waveform. Before using the cursor for measurement, we need a stable waveform display. The cursor measurement function provides the following two types of cursors.



- X Cursor

The X cursor is a vertical dashed line used for horizontal adjustment, which can be used to measure time (s) and frequency (Hz).

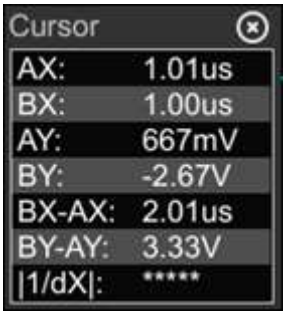
- Y Cursor

The Y cursor is a horizontal dashed line used for vertical adjustment, which can be used to measure amplitude (consistent with the amplitude unit of the source channel).
Click the **[Cursors]** in the button area to enter the cursor setting menu. The selected modes include: None, Manual, and Tracking. When selecting a manual cursor or tracking cursor, a cursor measurement window will appear on the screen, displaying the measurement results.

10.5.1 Manual Mode

In manual cursor mode, the value of the specified signal source waveform at the current cursor can be measured by manually adjusting the cursor. If the settings of parameters such as cursor type and measurement source are different, the results obtained by using the cursor will also be different.
Click the **[Cursors]** in the button area to enter the cursor setting menu. The cursor defaults to off mode.
Click on the **Mode** menu tab and select the cursor mode as manual. In the window

where the measurement results are displayed, the window can be moved. By changing the cursor position, the measurement results also change in real-time.



AX: The X value at cursor A.

BX: The X value at cursor B.

AY: The Y value at cursor A.

BY: The Y value at cursor B.

BX-AX: Horizontal spacing between cursors A and B.

BY-AY: The vertical spacing between cursors A and B.

|1/dX |: The reciprocal of the horizontal spacing between cursors A and B.

1. Select:

Click on the **Select** menu tab and select the cursor type X or Y.

- X: The X cursor is a vertical solid line (cursor A) and a vertical dashed line (cursor B), usually used to measure time parameters. The measurement results include AX, BX, ΔX, and 1/ΔX (only displayed when "s" or "Hz" is selected for "horizontal units").
- Y: The Y cursor is a horizontal solid line (cursor A) and a horizontal dashed line (cursor B), usually used to measure voltage parameters, and the measurement results include AY, BY, and ΔY.

2. Source:

Click on the **Source** menu tab, and the optional source channels are CH1-CH4, Math, or LA.

- Only the currently open channels are selectable.
- When the source selects "LA", the selection menu is grayed out and cannot be changed. By default, the "X" cursor type is selected.

3. AX:

Click on the **AX** menu label to select the horizontal position of cursor A. After selection, rotate the knob V0 to adjust the cursor position.

4. BX:

Click on the **BX** menu label to select the horizontal position of cursor B. After selection, rotate the knob V0 to adjust the cursor position.

5. AXBX:

Click on the **AXBX** menu tab to select both the horizontal positions of cursor A and cursor B. After selection, rotate knob V0 to simultaneously adjust the positions of cursor A and cursor B. The horizontal spacing between cursor A and cursor B remains unchanged.

6. AY:

Click on the **AY** menu label to select the vertical position of cursor A. After selection, rotate the knob V0 to adjust the cursor position.

7. BY:

Click on the **BY** menu label to select the vertical position of cursor B. After selection, rotate the knob V0 to adjust the cursor position.

8. AYBY:

Click on the **AYBY** menu label to select both the vertical positions of cursor A and cursor B. After selection, rotate knob V0 to simultaneously adjust the positions of cursor A and cursor B. The vertical spacing between cursor A and cursor B remains unchanged.

9. Test example:

Use the manual measurement function (cursor) and the automatic measurement function to measure the period of a sine wave, and the measurement results are both 1 ms.

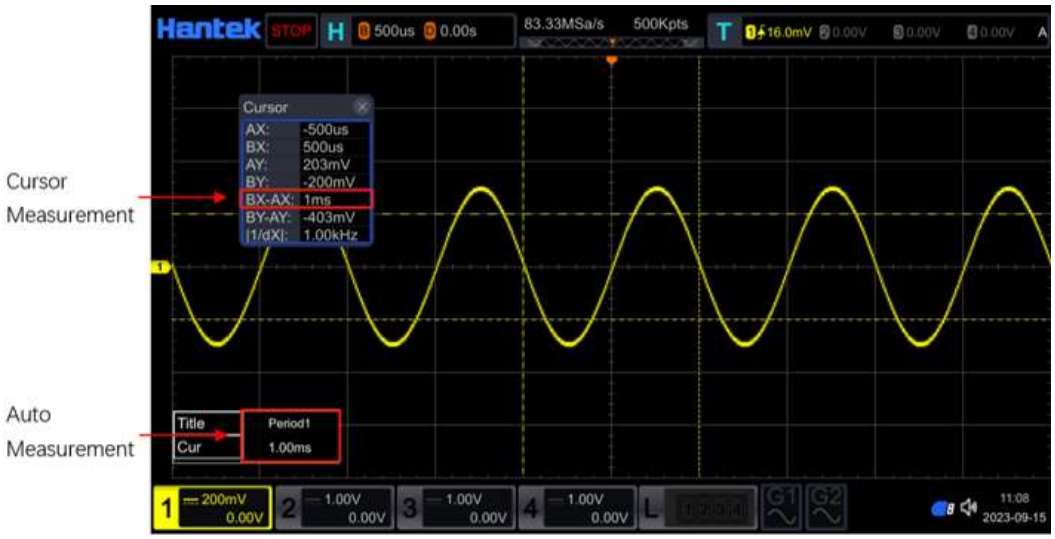


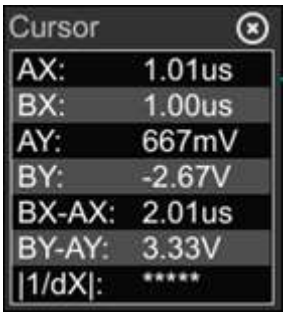
Figure 10.2 Manual Cursor Measurement Example

10.5.2 Track Mode

In tracking cursor mode, two cursors (cursor A and cursor B) can be adjusted to measure the X and Y values of two different sources, respectively. When moving the cursor horizontally/vertically, the marker will automatically position on the waveform. When expanding or compressing the waveform horizontally/vertically, the marker will track the point marked during the last adjustment of the cursor.

Click the **[Cursors]** button in the button area to enter the cursor setting menu. The cursor defaults to off mode.

Click on the **Mode** menu tab and select the cursor mode as tracking. In the window where the measurement results are displayed, the window can be moved. By changing the cursor position, the measurement results also change in real-time.



AX: The X value at cursor A.

BX: The X value at cursor B.

AY: The Y value at cursor A.

BY: The Y value at cursor B.

BX-AX: Horizontal spacing between cursors A and B.

BY-AY: The vertical spacing between cursors A and B.

|1/dX |: The reciprocal of the horizontal spacing between cursors A and B.

1. Source:

Click on the **Source** menu label, and the available information sources include CH1~CH4 and Math.

2. AX:

Click on the **AX** menu label to select the horizontal position of cursor A. After selection, rotate the knob V0 to adjust the cursor position.

3. BX:

Click on the **BX** menu label to select the horizontal position of cursor B. After selection, rotate the knob V0 to adjust the cursor position.

4. AXBX:

Click on the **AXBX** menu tab to select both the horizontal positions of cursor A and cursor B. After selection, rotate knob V0 to simultaneously adjust the positions of cursor A and cursor B. The horizontal spacing between cursor A and cursor B remains unchanged.

5. Measurement example:

Measure the waveform of CH1 using cursor A and cursor B, and then expand the waveform horizontally. It can be observed that the cursor will track the marked point, as shown in the following figure.

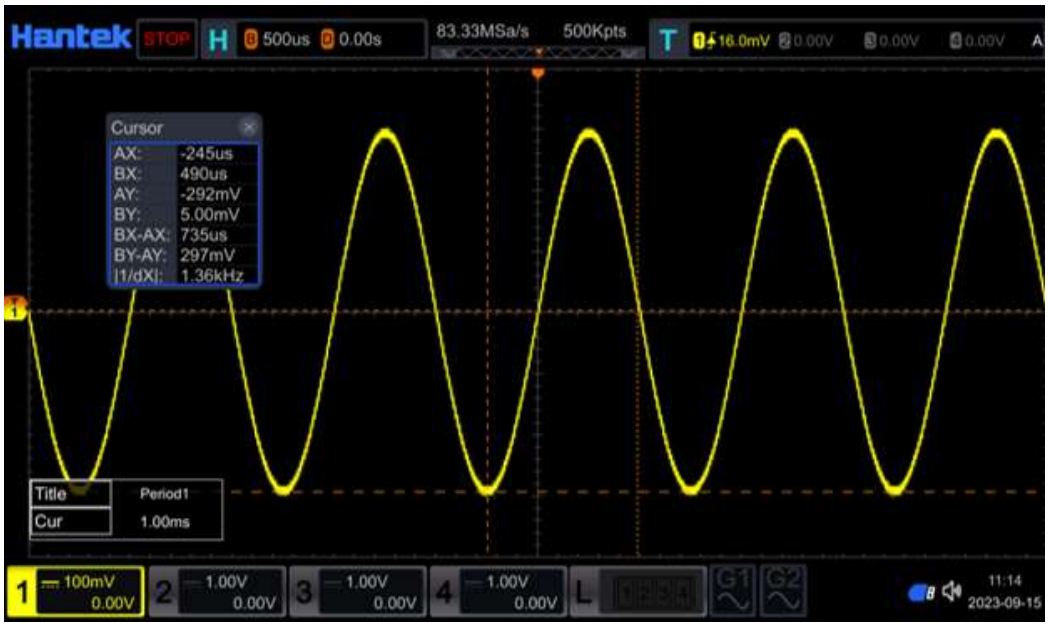


Figure 10.3 Track Measurement (before Horizontal Expansion)

Figure 10.4 Track Measurement (after Horizontal Expansion)

11 Reference Waveform

Click the **[REF]** button in the button area to enable the reference waveform function.

After enabling the REF function, you can select different colors for each reference waveform, set the signal source for each reference channel, adjust the vertical gear and offset of the reference waveform, save the reference waveform to internal or external storage, and recall it for use when needed.

This chapter includes:

11.1 Source

Click on the **Source** menu label and select the reference waveform source CH1~CH4 or D0~D15 from the pop-up menu.

11.2 Clear

Click on the **Clear** menu tab to clear the reference waveform.

11.3 To Adjust the Ref Waveform Display

Set the vertical gear and vertical offset of the reference waveform.

1. **Vertical Scale:**

Click on the **VScale** menu label and set the vertical gear of the reference waveform by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

2. **Vertical offset:**

Click on the **VOffset** menu tab and set the vertical offset of the reference waveform by rotating the multi-functional knob V0 or the pop-up numeric keyboard.

11.4 Detail

Click on the **Details** menu tab to open the Reference Details window.

11.5 To Reset the Reference Waveform

Reference waveform reset refers to: if a gear or offset operation is performed on the reference waveform, the reset operation can restore the waveform to the original saved position.

Click on the **Reset** menu tab to reset the reference waveform.

11.6 Color Setting

This series of oscilloscopes provides multiple colors to identify different reference waveforms, allowing users to better distinguish between reference waveforms.
Click on the **Color** menu tab and there are five colors to choose from.

11.7 Label Setting

Click on the **Label** menu label to enter the label settings menu.

1. **Display Label:**

Click on the **Display Label** menu label and choose to turn on or off the reference waveform label.

2. **Label Library:**

Click on the **Library** menu label, rotate the multi-functional knob to select the preset label, and press the knob to select it. You can also continuously press the tag library key or use the touch screen function to make selections. You can choose preset labels such as Default, ACK, ADO, or ADDR.

3. **Label:**

Click on the **Label** menu label and manually enter the label name at the pop-up keyboard.

11.8 Import

Users can import reference waveform files stored in the instrument's internal memory or external USB drive into the instrument for display on the screen.

Click on the **Import** menu tab to enter the reference waveform file loading interface. Please refer to the Store and Load for storage calls for specific operations.

Example of reference waveform:

Figure 11.1 Example of reference waveform

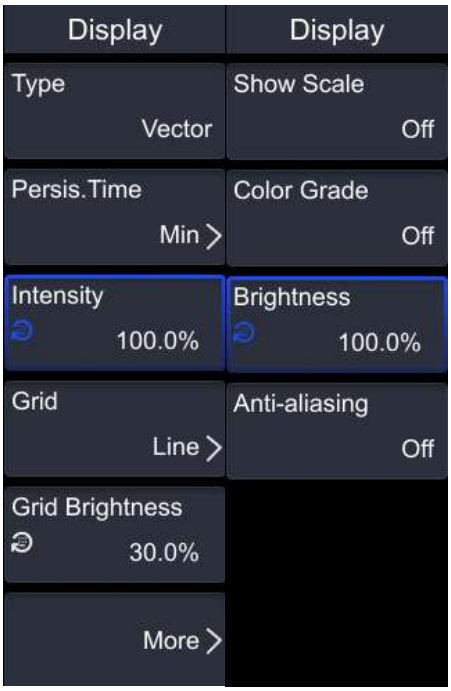
11.9 Export

Users can save the current reference waveform to the instrument's internal memory or external USB drive. The reference waveform file format is *. ref, *. bin, or *. csv.
Click on the **Export** menu tab to enter the reference waveform file save interface.
Please refer to the Store and Load for storage calls for specific operations.

12 Display

In the display settings menu, users can set the waveform display type, afterglow time, waveform brightness, grid type and brightness displayed on the screen, etc.

Click on the button area **[Display]** to enter the display settings menu.



This chapter includes:

12.1 To Select the Display Type

Click on the **Type** menu label, which can be selected as vector or point. Choose whether the waveform is displayed as a vector or as a point. The vector setting will fill in the blank space between adjacent sampling points in the display; Point settings only display sampling points.
The vector type sampling points are displayed through connecting lines. This mode provides the most realistic waveform in most cases, making it easy to view the steep edges of waveforms (such as square waves).

12.2 To Set the Persistence Time

In the displayed menu, click on the **Persis.Time** menu label and select afterglow time (Min, 100ms, 200ms, 500ms, 1s, 2s, 5s, 10s, Infinite).

- Minimum value: Observable waveform with high refresh rate changes.
- Specific value: burrs with slower changes or lower probability of occurrence can be observed. The afterglow time can be set to 100ms, 200ms, 500ms, 1s, 2s, 5s, and 10s.
- Infinite: When the oscilloscope displays the newly collected waveform, it will not clear the previously collected waveform. The collected waveforms will be displayed in colors with lower brightness, while the newly collected waveforms will be displayed in normal brightness and color. The use of infinite afterglow can measure noise and jitter, capturing occasional events.

12.3 To Set the Waveform Intensity

Click on the **Intensity** menu button, and set the brightness value directly through the multi-functional knob V0 or the numeric keyboard, with a range of 0%~100%.

12.4 To Set the Screen Grid

Click on the **Grid** menu tab to select the types of screen grids: Dot, Line, and Close.

- Dot: Set the background grid line as a point.
- Line: Set the background grid to solid lines.
- Close: Close the background grid.

12.5 To Set the Grid Brightness

Click on the **Grid Brightness** menu tab and set the grid brightness value directly through the multi-functional knob V0 or the numeric keypad, with a range of 0%~100%.

12.6 Scale

Click on the **Scale** menu tab and choose to turn the ruler function on or off.

12.7 Color Grade

Click on the **Color Grade** menu tab and choose to turn on or off the color temperature function. When the color temperature is displayed, different colors representing the number or probability of data collection will be displayed on the screen.

12.8 To Set the Screen Brightness

Click on the **Brightness** menu button, and set the brightness value directly through the multi-functional knob V0 or the numeric keypad, with a range of 0%~100%.

12.9 Anti-aliasing

Click on the **Anti-aliasing** menu tab and select whether to turn on or off the anti aliasing function.

13 System Utility Function Setting

Click on **[Utility]** in the button area. Enter system configuration.
Users can set system related functional parameters or interfaces.

This chapter includes:

13.1 Interface Configuration

Click on the **I/O** menu tab and select LAN interface or RS232 interface.

13.1.1 LAN Configuration

Click on **[Utility]** in the button area to enter the menu, click on the **I/O** menu tab, select LAN, and enter the LAN settings menu.

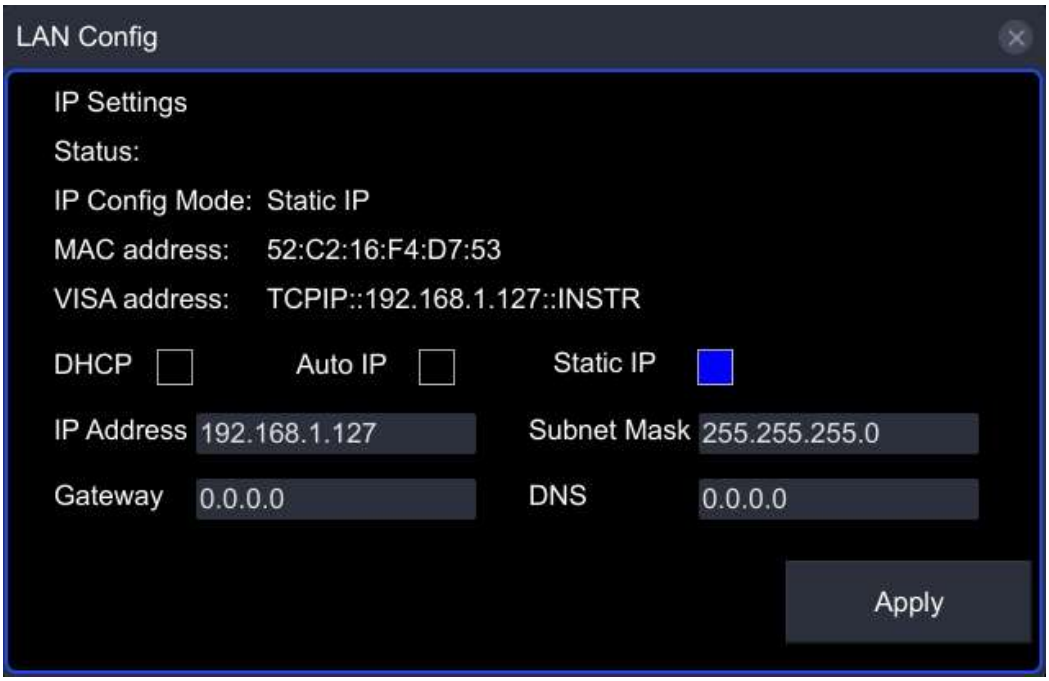


Figure 13.1 LAN Connection Setting Interface

1. Status:

The oscilloscope will provide different prompts based on the current network connection status:

- Network Config Succeeded!: (Network configuration successful!)
- Acquiring IP...: (Getting IP...)
- IP Conflict!: (IP Conflict!)
- Disconnected!: (No connection!)
- DHCP Configuration Failed: (DHCP configuration failed)
- Read Status Fail!: (Status reading failed!)
- Connected! : (Successfully connected!)
- Invalid IP: (Invalid IP)

- IP loss: (IP lost)
- Please wait...: (Please wait...)
- Auto IP Config Succeeded!: (Automatic IP allocation successful)
- Auto IP Config Failed!: (Automatic IP allocation failed)

2. IP Config Mode

The IP Config Mode include DHCP, Auto IP, and Static IP. Under different methods, the configuration methods for network parameters such as IP addresses vary.

- DHCP:

Click on the **IP Config Mode** menu tab and select Set as Dynamic IP. When the dynamic IP is valid, the dynamic IP server in the current network will assign IP address, subnet mask, gateway address, domain name server address and other network parameters to the oscilloscope.

- Auto IP:

Click on the **IP Config Mode** menu tab and select Set as Dynamic IP. In the automatic IP mode, the oscilloscope automatically acquires the IP address and subnet mask 255.255.0.0 from 169.254.0.1 to 169.254.255.254 according to the current network configuration. Automatic IP only takes effect when dynamic IP is not selected or connection fails.

- Static IP:

Click on the **IP Config Mode** menu tab and select Set as Static IP. When this mode is effective, manually turn off dynamic IP and automatic IP, and the menu of setting IP address, setting subnet mask, setting gateway address, and setting DNS address will be added. At this point, users can customize network parameters such as the IP address of the oscilloscope.

➢ Set IP Address

The format of the IP address is nnn. nnn. nnn. nnn. The first nnn can be set in the range of 0 to 255 (excluding 127), with a valid range of 0 to 223. The other three nnns all have a range of 0 to 255. It is recommended to consult your network administrator for an available IP address.
Press the **IP Address** key to pop up the numeric keypad and enter the desired IP address. This setting will be saved in non-volatile memory. If "Power on Call" is set to "Last Value", the next time the device is turned on, both dynamic IP and automatic IP will be turned off, and the oscilloscope will automatically load the set IP address.

➢ Set Subnet Mask

The format of the subnet mask is nnn.nnn.nnn.nnn, where nnn ranges from 0 to 255. It is recommended that you consult your network administrator for an available subnet mask.

Press the **Subnet Mask** key, a numeric keypad will pop up, and enter the required subnet mask. This setting is saved in the non-volatile memory. If the "power on call" is set to the "last value", the dynamic IP and automatic IP will be turned off when the next power on, and the oscilloscope will automatically load the set subnet mask.

- Set up Gateway

In Static IP mode, you can set up a gateway.
The format of the gateway is nnn. nnn. nnn. nnn, with the first nnn ranging from 0 to 223 (excluding 127) and the other three nnns ranging from 0 to 255. It is recommended to consult your network administrator for an available gateway address.
Press the **Gateway** key, pop up the numeric keypad, and enter the desired gateway address. This setting is saved in non-volatile memory. If the "power on call" is set to "last value", the next time the power is turned on, both dynamic IP and automatic IP are turned off, and the oscilloscope will automatically load the set gateway address.

- Set Domain Name Server Address

In static IP mode, you can set the domain name server address.
The address format of the domain name server is nnn.nnn.nnn.nnn, with the first nnn ranging from 0 to 223 (excluding 127) and the other three nnns ranging from 0 to 255. It is recommended to consult your network administrator for an available address.
Press the **DNS** key, pop up the numeric keypad, and enter the desired address. Generally speaking, users do not need to set the domain name server address in the network, so this parameter setting can be ignored.

3. MAC Address

For an oscilloscope, the MAC address is always unique. When assigning IP addresses to instruments, they are always identified by MAC addresses.

4. VISA Address

Display the VISA address currently used by the oscilloscope.

13.1.2 RS232 Configuration

Click on **[Utility]** in the button area to enter the menu, click on the **I/O** menu label, select RS232, and enter the RS232 settings menu label.
Click on the **Baud Rate** menu tab and select Baud Rate from the pop-up menu.

13.2 Beeper

[Utility] -> **Beeper**. The sound can be selected to be turned off or on.

Click on the **Beeper** menu tab to choose whether to turn sound on or off. The sound sign is displayed in the bottom right corner of the screen.

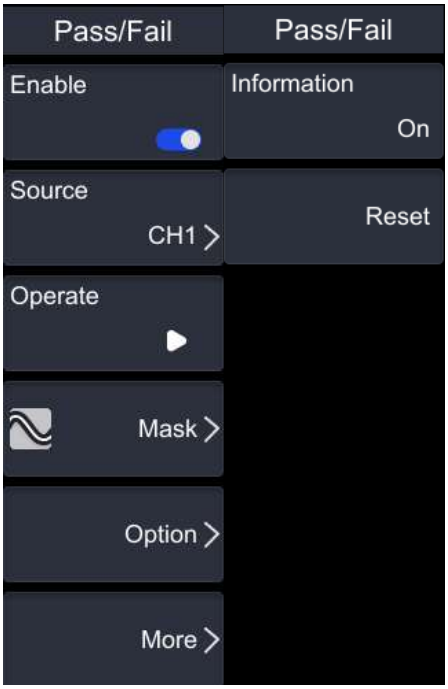
13.3 Language

Click on the **Language** menu tab to switch between Chinese or English.

13.4 Pass/Fail

By testing the function, it is possible to better monitor the changes in signals. You can enter the testing menu through the following methods:

- In **[Utility]** -> **Pass/Fail**, enter the Pass/Fail function menu.
- Click on the navigation label **[Hantek]** in the upper left corner of the screen, select **Pass/Fail**, and enter the Pass/Fail function menu.



13.4.1 To Enable or Disable the Pass/Fail Test Function

Click on the **Pass/Fail** menu tab to enter the Pass Test Settings menu.

Click on the **Enable** menu tab to select whether to enable or disable the pass testing function. The icon illuminates to enable the pass test function.

13.4.2 To Select the Source

Click on the **Source** Selection menu tab in the Test menu to select channels CH1 to CH4.

13.4.3 To Start or Stop the Pass/Fail Test Operation

Click on the **Operation** menu tab in the Test menu to start or stop the test operation. After starting the testing operation, the oscilloscope will test the waveform. Select the testing source and set testing rules as needed.

Only when the pass test function is enabled can the pass test operation be started or stopped.

The testing interface is shown in the following figure:

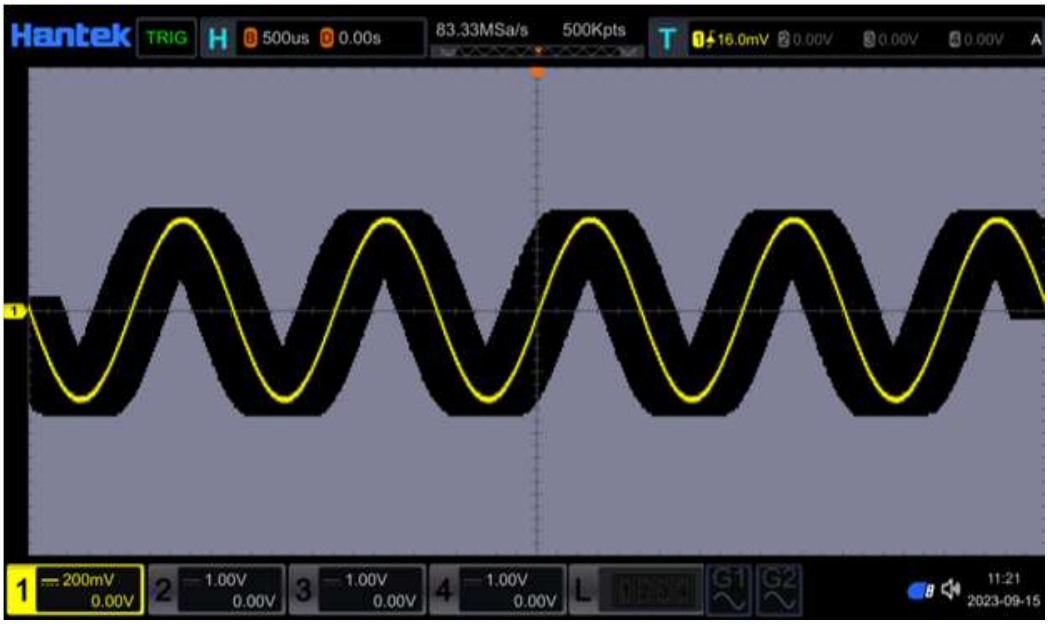


Figure 13.2 Pass/Fail examples

13.4.4 Mask

1. Create Rules

Click on the **Mask** menu tab in the Test menu to enter the Mask Settings menu, select **Create** Rule, and enter the Settings menu.
Click on the **Range** menu tab to select the Screen or Cursor.
Click on the **X Mask** menu tab and set the horizontal tolerance range on the pop-up numeric keyboard or V0 knob.
Click on the **Y Mask** menu tab and set the vertical tolerance range on the pop-up

numeric keyboard or V0 knob.
Click on the **Create** Rule menu tab to apply the currently created rule.

2. Load Rules

Click on the **Mask** menu tab in the Test menu to enter the Mask Settings menu, select **Load** Rules, and enter the Settings menu.
Click on the **File Type** menu tab, and the file types you can choose are: *. stp or *. ptr.
Click the **Load** menu tab to call out the stored files from the machine's internal memory or external memory (successfully connected to the USB flash disk) and apply them to the current passing test function.

3. Save Rule

In the pass test menu, users can enter the mask menu to customize the pass/fail test rules.
Enter the Test Settings menu, click on the **Mask** Menu tab to enter the Mask Settings menu, select **Save** Rules, and enter the File Save menu.
Click on the **File Type** menu tab and select the file save type, which can be *. stp or *. ptr.
Click on the **File Name** menu tab to modify the file name.
Click on the **Window** menu tab to choose whether to open or close the window.
Click on the **Save** menu tab to save the settings file in the machine's internal or external memory (external memory USB flash drive has been recognized on the machine).

13.4.5 Option

Click on the **Option** menu tab in the Pass/Fail menu to enter the Options Settings menu.

1. Aux Output:

Click on the **Aux Output** menu tab to choose whether to turn on or off the Aux output function. If set to enable Aux output function, then **[Utility]** -> **System** -> **AUXOut** will automatically be set to pass the test. If the Aux output function is turned off, then **[Utility]** -> **System** -> **AUXOut** is automatically set to trigger output.

2. Output Event:

Click on the **Output Event** menu tab to select either pass or fail.

3. Output polarity:

Click on the **Polarity** menu tab to select positive or negative pulses.

4. Output Pulse Width:

Click on the **Pulse** menu label, and the range of output pulse width can be selected from 100ns to 100ms. Set the pulse width through the pop-up numeric keypad.

5. Error Action:

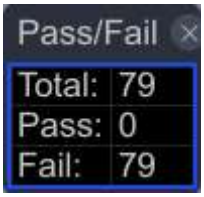
Click on the **Error Action** menu tab to select Attonity, Buzzer.
Buzzer: When a failure event is detected, the machine emits a sound (independent of the machine's sound on/off state).

6. Stop Output:

Click on the **Stop Output** menu tab to choose whether to turn it on or off.

13.4.6 Information

By clicking on the **Information** menu tab in the Test menu, you can choose to turn on or off the information. Select to open the information, and the test results table will appear on the screen. The test results include total number, pass, and fail.
When opening the measurement results tab, click on the **Reset** menu tab to reset the data in the measurement results table.



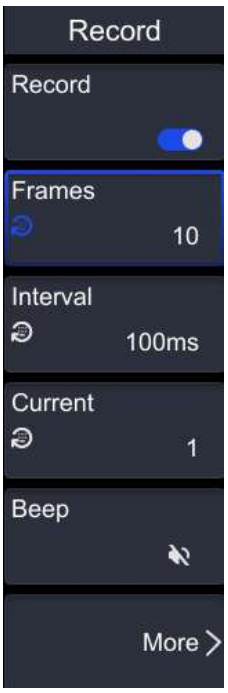
13.4.7 Statistical Reset

Click on the **Reset** menu tab in the Test menu to reset the data in the measurement results table.

13.5 Recording & Playing

The recording and playback functions of waveforms can play the recorded analog input channels (CH1-CH4) waveforms, making it convenient for users to analyze waveforms.
The menu can be accessed through the following methods:

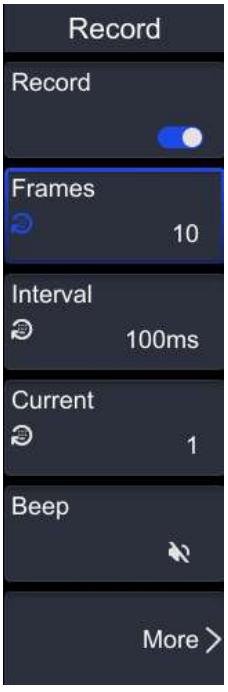
- In the **[Utility]** -> **Record**, enter the waveform recording function menu.
- Click on the navigation label **[Hantek]** in the upper left corner of the screen, select **Record**, and enter the waveform recording function menu.



13.5.1 Record Options

For waveform recording, it is necessary to set the recording time, recording frame number, etc. in advance. The oscilloscope can manually stop recording or the recorded frame number can reach the set recording frame number based on the currently open channel for recording.

Click **[Utility]** in the button area to enter the menu, Click on **Record**, select to turn on or off the waveform recording function, and turn it on to turn it on.



1. **Frames:**

Click on the **Frames** rate menu tab and set the recorded frame rate using the multi-functional knob V0 or numeric keyboard. The number of frames recorded represents the number of frames that can be recorded. After starting recording, if the number of recorded frames reaches the set recording frame number, the recording will automatically stop.

2. **Record Interval:**

Recording interval refers to the time interval between frames during the recording process.
Click on the **Record Interval** menu tab and set the recording time using the multi-functional knob V0 or the numeric keypad.

3. **Current Frames:**

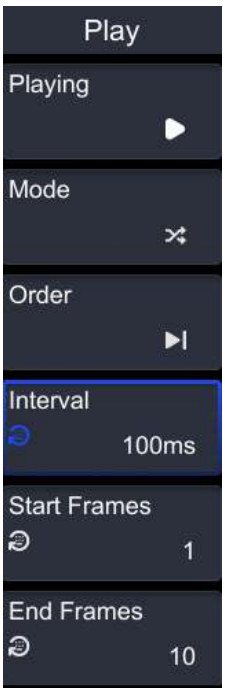
Press **Current**, then rotate the multifunction knob or use the pop-up numeric keypad to set the current frame. At this time, the waveform of the current frame is displayed on the screen..

4. **Beeper:**

Click on the **Beeper** menu tab to select whether the buzzer will sound when the recording ends.

13.5.2 Play Options

Play function:
Click on **[Utility]** in the button area to enter the menu, click on the **Record** -> **More** menu tab to enter the playback settings.



1. Play Mode:

Click on the **Mode** menu tab to choose between looping or single playback.

- Loop playback: Repeat playback from the start frame to the end frame until manually stopping the playback operation.
- Single playback: Automatically stops from the start frame to the end frame.

2. Play Order:

- Sequential playback: Plays from the start frame to the end frame.
- Reverse playback: Plays from the end frame to the start frame.

3. Play Interval:

Click on the **Play Interval** menu tab, and you can set the time interval between frames by rotating the knob V0 or directly setting the time interval between frames on the pop-up numeric keyboard.

4. Start Frame

Press **Start Frame**, then rotate the multifunction knob or use the pop-up numeric keypad to set the start frame of playback. By default, it is 1, and the maximum value is the number of recorded frames.

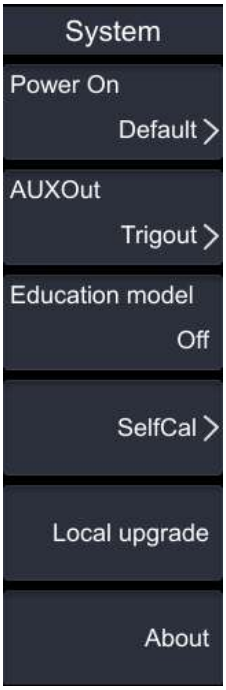
5. End Frame

Press **End Frame**, then rotate the multifunction knob or use the pop-up numeric keypad to set the end frame of playback. The default is the number of frames of the recorded

waveforms.

13.6 System

Click on the **[Utility]** button area to enter the menu, click on the **System** menu tab, and enter the system settings menu.



13.6.1 Power On

Click on the **Power On** menu tab and select the system configuration that will be called when the device is powered back on after a power down. You can choose to restore the Default or Last.

- Default: Restore the system to factory settings.
- Last: Reply to the settings from the last time the system was powered down.

13.6.2 Aux Output

[Utility] -> **More** -> **System** -> **AUXOut**, enter the Auxiliary Output menu tab, and there are two options: Trigout and Pass/Fail.

- Trigout: When the oscilloscope generates a hardware trigger, it will output a signal from the auxiliary output connector that reflects the current capture rate of the oscilloscope. Input the signal into the port, display the waveform on the screen, and measure the frequency of the waveform. It is found that the frequency is the same as the current capture rate.
- Pass/Fail: In the pass test, when the oscilloscope detects a pass or fail, a positive

or negative pulse will be output from the auxiliary output connector. Set **[Utility]** -> **Pass/Fail** -> **AUXOut** to On, then the **[Utility]** -> **More** -> **System** -> **AUXOut** menu will automatically be set to "Pass/Fail". If the auxiliary output menu here is set to "trigout", then **[Utility]** -> **Pass/Fail** -> **AUXOut** function will automatically close.

13.6.3 Education model

Click on the **Education Model** menu tab and select whether to turn Education Mode on or off.

13.6.4 Self-calibration

The self-calibration program can optimize the oscilloscope signal path with maximum measurement accuracy. You can run this program at any time, but if the ambient temperature changes by more than 5℃, you should run this program.

For more accurate calibration, please turn on the oscilloscope power and preheat for 30 minutes before performing self calibration. Follow the prompts on the screen to operate. To compensate for the signal path, disconnect any probes or cables connected to the input connector.

Click on the **SelfCal** menu tab and select **Offset Start**.

13.6.5 Local upgrade

[Utility] -> **More** -> **System** -> **Local Upgrade**,

Click **Local Upgrade** and select whether to upgrade the firmware. If you click **OK**, select the firmware file to be upgraded; Click **Cancel** to exit the upgrade interface.

13.6.6 About this oscilloscope

[Utility] -> **More** -> **System** -> **About**.

Click on the menu label of **About**, and the system information of the oscilloscope will pop up on the screen, including the model, serial number, software version, hardware version, etc. of the oscilloscope.

The menu also includes the functions included in the oscilloscope: 16 digital channels, any signal generator, digital voltmeter, power analysis, Bode diagram, histogram, RS232/422/485/UART, IIC/SPI, CAN LIN FlexRay, MIL-STD-1533 ARINC429, storage depth option.

13.7 Clock Source

Click on the **[Utility]** -> **More** -> **Clock Source** menu tab, and select whether the clock source is Internal or External.

13.8 Key Locker

Click on **[Utility]** -> **More** -> **Key Locker** menu tab. Users can set whether to lock the keyboard. After the keyboard is locked, users will no longer be able to use the keyboard for configuration. At the same time, the **[Touch Lock]** button in the keyboard area lights up.

13.9 Quick Setting

You can access the shortcut menu through the following methods:

- Click on **[Utility]** -> **More** -> **Quick setting** to enter the shortcut menu.
- Click on the navigation icon **[Hantek]** in the upper left corner of the screen, select shortcut operations, and enter the settings menu.

Quick setting include: Save Image, Save Wave, Save Setup, All Measure, Stat Reset, Record, Save Group.

13.9.1 Save Image

Click on the **Operation Type**, select the **Save Image** menu tab, and make further settings.

- In the **Format** item, you can click to select image types including "*. png", "*. bmp", "*. jpg", or "*. tif".
- In the **Invert** option, click to open the Invert function.
- In the **Color** item, click to select the color stored in the image.

After setting up, click the **[Quick Action]** button in the button area to capture the current screen and save the image in the set type. Do not insert a USB drive, it is saved internally by default. Insert a USB drive and save it in the USB drive.

13.9.2 Save Wave

Click on the **Operation Type**, select the **Save Wave** menu tab, and make further settings.

- In the **Data Source** item, you can click to select the stored waveform from the

screen or memory. After selecting the memory as the data source, you can choose the corresponding source: CH1-CH4, D0-D15.

- In the **Format** item, optional format types include "*. bin" or "*. csv".

After setting, click the **[Quick Action]** button to save the waveform according to the settings. Do not insert a USB drive, it is saved internally by default. Insert a USB drive and save it in the USB drive.

13.9.3 Save Setup

Click on the **Operation Type**, select the **Save Setup** menu tab, and make further settings.

After setting, click the **[Quick Action]** button to save the waveform settings according to the settings. Do not insert a USB drive, it is saved internally by default. Insert a USB drive and save it in the USB drive.

13.9.4 All Measure

Click on the **Operation Type**, select **All Measure** menu labels, and make further settings.

In **All Measure** items, options include OFF or measurement channels CH1 to CH4. After setting up, click the **[Quick Action]** button to measure the specified channel.

13.9.5 Stat Reset

Click on the **Operation Type**, select the **Stat Reset** menu tab, and make further settings.

In the **Stat Reset** option, you can select the function "Measure" or "Pass/Fail". After setting, click the **[Quick Action]** button to reset the statistical results of the specified function and restart the measurement.

13.9.6 Record

Click on the **Operation Type**, select the **Record** menu tab, and make further settings. After setting, click the **[Quick Action]** button to record the waveform.

13.9.7 Save Group

Click on the **Operation Type**, select the **Save Group** menu tab, and make further

settings.

In the **Save Group** option, the available storage options are "Save Image", "Save Wave", or "Save Setup".

After setting up, click the **[Quick Action]** button to store the selected content according to the configuration. Do not insert a USB drive, it is saved internally by default. Insert a USB drive and save it in the USB drive.

13.10 Screen Saver

Click **[Utility]** -> **More** -> **Screen Saver** menu tab, options: off, Picture, Text.

Select Picture: Click to **Select Picture** and load it locally on the machine or on a USB drive. The image format is *. png, *. bmp, *. jpg, *. tif; Click on the **Time to Start** menu tab and set the waiting time (5min, 10min, 30min, 1h, 2h, 4h); Click on the **Preview** menu tab to see the effect of the settings in advance; Click on the **Default** Menu tab to restore the settings to their default state.

Select text: Click on the **Text** menu label and enter custom text on the keyboard; Click on the **Time to Start** menu tab and set the waiting time (5min, 10min, 30min, 1h, 2h, 4h); Click on the **Preview** menu tab to see the effect of the settings in advance; Click on the **Default** Menu tab to restore the settings to their default state.

13.11 Self Check

Enter the self-test menu through the following methods:

- Click on the **[Utility]** -> **More** -> **More** -> **Self Check** menu tab to enter the Self Check menu.
- Click on the navigation icon **[Hantek]** in the upper left corner of the screen, select **Self Check**, and enter the Self Check menu.

The self-test options include Key Test, Screen Test, and Touch Test.
Key Test: Click on the **Key Test** menu tab to enter the keyboard testing interface. The icon corresponding to the button appears on the screen. By pressing the button on the front panel of the instrument or rotating the knob, the corresponding icon can be illuminated to determine whether the button or knob function is normal. Press' Run/Stop 'three consecutive times to exit the testing function.

Screen Test: Click on the **Screen Test** menu tab to enter the screen testing interface. Red and XXX appear in sequence. Follow the prompts and press "Single" to enter the next screen. Press' Run/Stop 'three consecutive times to exit the testing function.
Touch Test: Click on the **Touch Test** menu tab to enter the touch function testing interface. Use your finger to click on the screen. If there are corresponding lines displayed in the blank area and the boxes that pass through turn into green background boxes, it indicates that the touch screen function is normal in that area.

13.12 Time

Click on **[Utility]** -> **More** -> **More** -> **Time** to enter the time setting menu.

Click on the **Show Time** menu tab and set whether the time is displayed in the bottom left corner of the screen.

Click on the **Set Time** menu tab to enter the Set Time menu, where you can set the Year, Month, Day, Hour, and Minute.

13.13 Clock Set

Click on **[Utility]** -> **More** -> **More** -> **Clock Set** to enter the clock settings menu. You can choose between 2GSa/s and 2.5GSa/s.

14 Decode

Protocol decoding is the foundation of protocol analysis. Only correctly decoded protocol analysis can be accepted, and only correct decoding can provide more error information. This oscilloscope provides commonly used protocol decoding, including RS232, I2C, SPI, LIN, and CAN. You can enter the decoding menu through the following methods:

Click on the button area **[Decode]** to enter the decoding settings menu.

This chapter includes:

14.1 UART Decoding

The RS232 serial bus consists of a transmit data line (TX) and a receive data line (RX).

Figure 14.1 Schematic Diagram of UART Serial Bus

The industry standard for RS232 uses "negative logic", which means that high levels represent logic "0" and low levels represent logic "1".

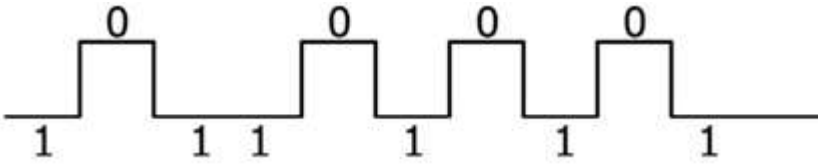
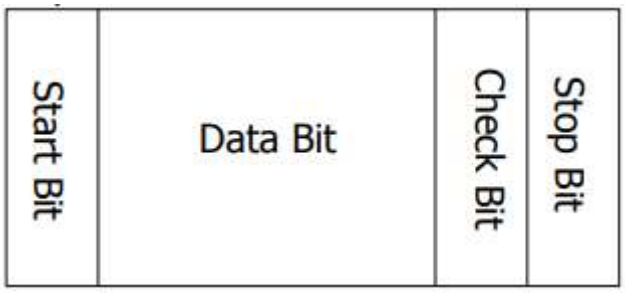


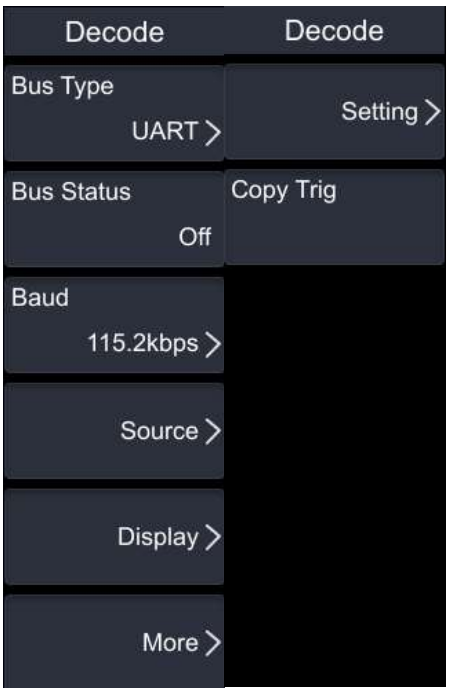
Figure 14.2 Schematic Diagram of Negative Logic

The baud rate is used in RS232 to represent the transmission rate of data (i.e. bits per second). RS232 requires setting the start bit, data bit, check bit (optional), and stop bit for each frame of data.



- Start Bit: indicates when the data starts to be output.
- Data Bit: represents the actual number of data bits contained in each frame of data.
- Check Bit: used to verify the correctness of data transmission.
- Stop Bit: indicates when the data stops outputting.

Click **[Decode]** in the button area to enter the decoding function menu.



1. Bus Type:

Click on the **Bus Type** menu tab and select UART from the pop-up menu to configure UART decoding.

2. Bus Status:

Click on the **Bus Status** menu tab and choose to turn on (ON) or off (OFF) the bus switch function.

3. Baud:

Click on the **Baud** menu tab and select Baud Rate from the pop-up menu.

4. Source:

Click on the **Source** menu tab to enter the Source Settings menu.

- Set the Tx source and Threshold

Click on the **Tx** menu tab and select the source channel from the pop-up menu.
Click on the **Threshold** menu label and set the threshold through the multi-functional knob V0 or the pop-up numeric keyboard.

- Set the Rx source and threshold

Click on the **Rx** menu tab and select the source channel from the pop-up menu.
Click on the **Threshold** menu label and set the threshold through the multi-functional knob V0 or the pop-up numeric keyboard.

- Polarity:

Click on the **Polarity** menu tab (select data decoding as positive or negative polarity).

  - Positive polarity: Use negative logic, meaning that high levels represent logic

"0" and low levels represent logic "1".

- Negative polarity: Use positive logic, where high level represents logic "1" and low level represents logic "0".

5. Display:

Click on the **Display** Menu tab to enter the Display Settings menu.

- Format

Click on the **Format** menu tab, and the available formats include hexadecimal, decimal, binary, and ASCII.

- Position

Click on the **Position** menu tab to set the location.

- Label

Click on the **Label** to display the menu label, and choose to turn it on or off.

- Event Table

Click on the **Event Table** menu tab to enter the event table settings menu. The event table displays detailed decoding information in chronological order in the form of a table, making it easy to observe longer decoded data. Decoded information includes the decoded data (TX and/or RX, corresponding line numbers, time, and error information).

- Event Table
  The event table can be selected to open or close.
- Format
  Click on the **Format** menu tab to set the format of the event table. The options include hexadecimal, decimal, binary, and ASC.
- View
  Click on the **View** menu tab to select Package, Details, and Data. Select 'Package', and the decoded data, time, and error information will be displayed in the event table accordingly; Select 'Detailed' to display detailed data for the specified row in the event table. If '...' appears in the row data, it indicates that not all decoded data is displayed. At this time, you can view detailed information in the 'Package' view; Select 'Data' to display all data for the specified column in the event table. If '...' appears in the column data, it indicates that not all decoded data is displayed. At this time, you can view detailed information in the 'Package' view.
- Export
  Click on the **Export** menu label, and when the view selects "package", information such as time and corresponding decoding data can be exported.
- Jump
  Click on the **Jump** menu tab to display the specified data waveform in the center view of the screen.

6. Setting:

Click on the **Setting** menu tab to enter the Settings menu.

- Data:

Click on the **Data** menu tab, and the available data bits are: 5bit, 6bit, 7bit, 8bit, and 9bit.

- Parity:

Click on the **Parity** menu tab, and the available verification methods include: None, Odd, and Event.

  - None: There is no checksum during transmission.
  - Odd: The total number of "1" in the data bit and parity bit is odd. For example, if you send 0x55 (01010101), you need to check that the bit stuffing is filled with 1.
  - Event: The total number of "1" in the data bit and parity bit is even. For example, if 0x55 (01010101) is sent, 0 needs to be filled in the calibration position.

- Stop Bit:

Click on the **Stop Bit** menu tab, and the available stop bits are: 1 bit, 1.5 bit, and 2 bit.

- Endian:

Click on the **Endian** menu tab, and the available bit orders are: LSB, MSB.

  - LSB: Least Significant Bit, which means that the low bit of data is transmitted first.
  - MSB: Most Significant Bit (most significant bit), that is, the high data bits are transmitted first.

- Package:

Click on the **Package** menu tab to select whether to open or close the package.

- Separator:

Click on the **Separator** menu label, and the available package separators are 0A (LF), 0D (CR), 20 (SP), and 0 (NULL).

7. Copy Trig:

Click on the **Copy Trig** menu tab to select the Copy Trigger function. The copy trigger function can quickly apply trigger settings to corresponding types of decoding functions.

8. Error manifestations during decoding:

P represents checksum error, S represents stop bit error, and does not display when there are no errors.

9. Decoding example:

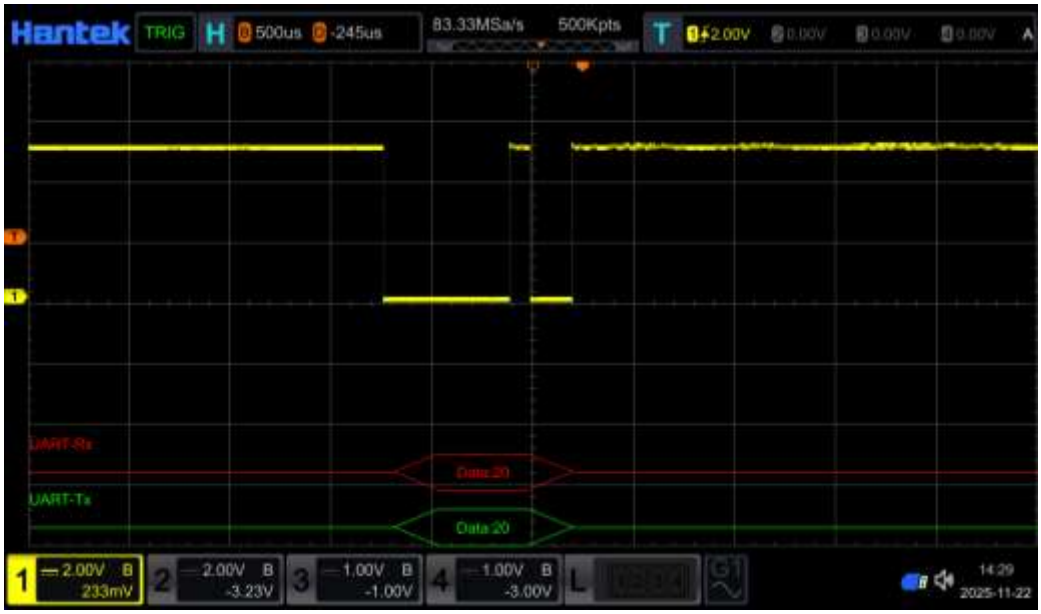


Figure 14.3 UART decoding example

14.2 I2C Decoding

The I2C serial bus consists of a clock line (SCL) and a data line (SDA).

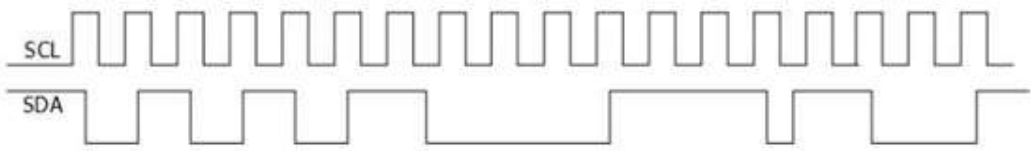
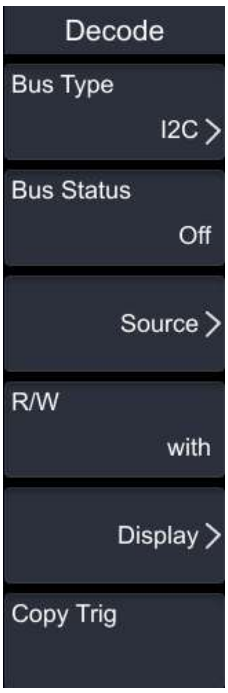


Figure 14.4 Schematic diagram of I2C serial bus

- SCL: Sampling SDA on the rising or falling edge of the clock.
- SDA: Represents a data channel.

Click **[Decode]** in the button area to enter the decoding function menu.



1. Bus Type:

Click on the **Bus Type** menu tab and select I2C from the pop-up menu to configure I2C decoding.

2. Bus Status:

Click on the **Bus Status** menu tab and choose to turn on (ON) or off (OFF) the bus switch function.

3. Source:

Click on the **Source** menu tab to enter the Source Settings menu.

- Set clock channel source and threshold
  - Click on the **SCL** menu tab and select the source of the clock channel.
  - Click on the **SCL Thre** menu tab and adjust the threshold of the clock channel through XX settings. When changing the threshold of the clock channel, a dashed line appears on the screen displaying the current threshold level. When the change is stopped, the dashed line of the threshold level disappears after about 2 seconds.
- Set data channel source and threshold
  - Click on the **SDA** menu tab and select the source of the data channel.
  - Click on the **SDA Thre** menu tab and adjust the threshold of the data channel through XX settings.
- Exchange

Click on the **Exchange** menu tab and select Clock/Data or Data/Clock. Can exchange the current clock channel and data channel sources.

4. R/W:

Click on the **R/W** menu tab to select whether the address information includes read/write bits or does not. When selecting include, the read/write bit will be included in the address information and will be in the lowest position; When not included, the address information does not include read/write bits.

5. Display:

Click on the **Display** Menu tab to enter the Display Settings menu.

- Format

Click on the **Format** menu tab to set the display format, which can be selected from: hexadecimal, decimal, binary, or ASCII.

- Position

Click on the **Position** menu tab to set the location.

- Label

Click on the **Label** to display the menu label, and choose to turn on or off the decoded label display. (Label display position)

- Event Table

Click on the **Event Table** menu tab to enter the event table settings menu.

  - Event Table
    The event table can be selected to open or close.
  - Format
    Click on the **Format** menu tab to set the format of the event table. The options include hexadecimal, decimal, binary, and ASC.
  - View
    Click on the **View** menu tab to select Package, Details, and Data.
  - Export
    Click on the **Export** menu label, and when the view selects "package", information such as time and corresponding decoding data can be exported.
  - Jump
    Click on the **Jump** menu tab to display the specified data waveform in the center of the screen.

6. Copy Trig:

Click on the **Copy Trig** menu tab to select the Copy Trigger function. The copy trigger function can quickly apply trigger settings to corresponding types of decoding functions.

7. Decoding example:

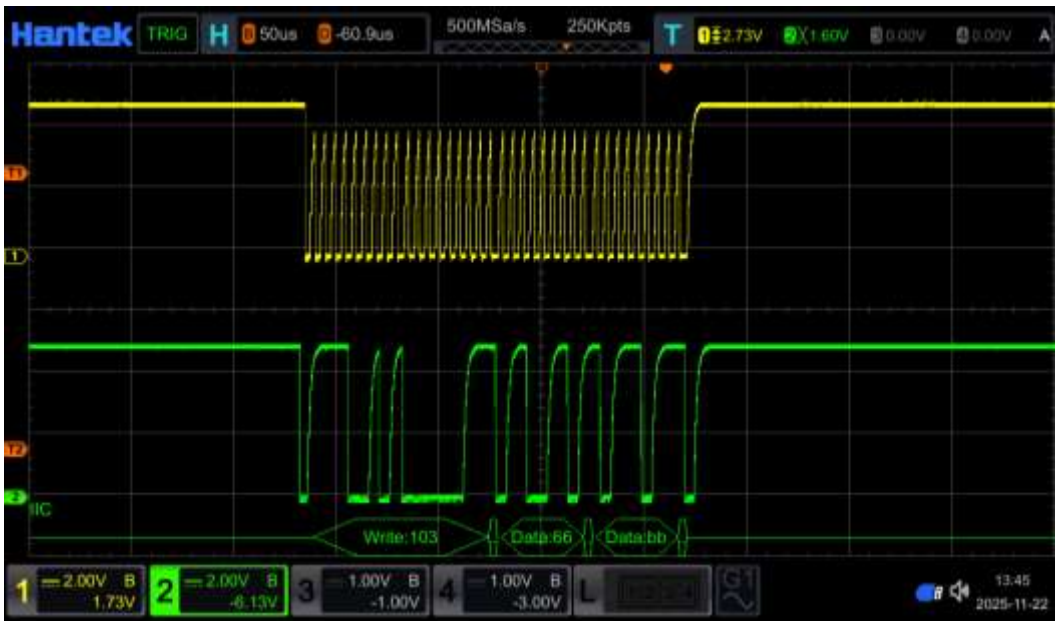


Figure 14.5 I2C Decoding example

14.3 SPI Decoding

SPI bus communication is based on master-slave configuration and generally consists of chip selection lines (CS), clock lines (CLK), and data lines (SDA). The data lines include MISO (master input slave output) and MOSI (master output slave input). The oscilloscope samples channel data on the rising or falling edge of the clock signal (if it is an analog channel, it will also determine whether each data point is a logical "1" or a logical "0" based on the set threshold level).

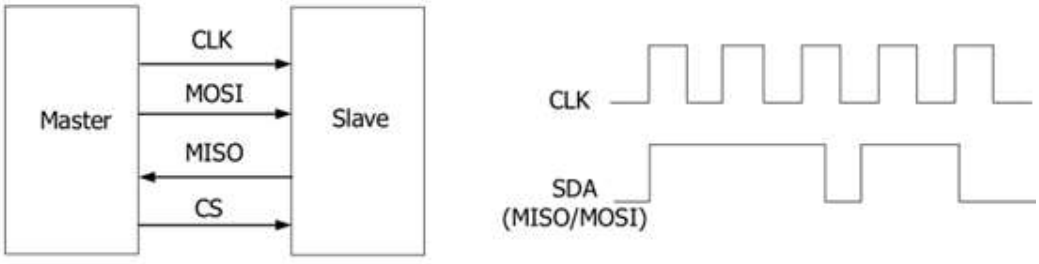
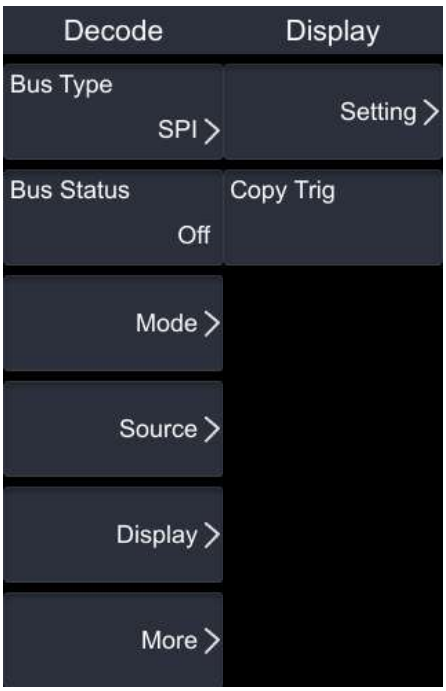


Figure 14.6 SPI Bus Schematic

Click **[Decode]** in the button area to enter the decoding function menu.



1. Bus Type:

Click on the **Bus Type** menu tab and select SPI from the pop-up menu to configure SPI decoding.

2. Bus Status:

Click on the **Bus Status** menu tab and choose to turn on (ON) or off (OFF) the bus switch function.

3. Mode:

Click on the **Mode** menu tab to select the SPI decoding mode as Timeout or CS.

- Timeout mode: frame synchronization is performed based on the timeout time, which must be greater than half a clock cycle.
  - Time: Click on the **Time** menu tab and set the timeout value through the multi-functional knob V0 or numeric keyboard.
- CS mode: Contains film selection line CS, and performs frame synchronization based on CS.
  - Film selection: Click on the **CS** menu tab and select the desired film selection menu (CH1-CH4, D0-D3, OFF).
  - CS Polarity: Click on the **CS Polarity** menu tab to select the polarity (positive or negative).
  - Threshold: Click on the **Threshold** menu tab and set the threshold using the multi-functional knob V0 or the numeric keypad.

4. Source:

Click on the **Source** menu tab to enter the Source Settings menu.

- Set clock signal:
  - Click on the **CLK** menu tab and select the clock source channel (CH1-CH4, D0-D3, OFF).
  - Click on the **Threshold** menu tab and set the threshold using the multi-functional knob V0 or the numeric keypad.
  - Click on the **Slope** menu label and select the edge type positive or negative polarity.
- Set MISO and MOSI:
  - Click on the **MISO** menu tab and select the MISO source channel (CH1-CH4, D0-D3, OFF).
  - Click on the **Threshold** menu tab and set the threshold using the multi-functional knob V0 or the numeric keypad.
  - Click on the **MOSI** menu tab and select the MOSI source channel (CH1-CH4, D0-D3, OFF).
  - (Reminder: The MISO and MOSI source channels cannot be set to OFF at the same time.)

5. Display:

Click on the **Display** Menu tab to enter the Display Settings menu.

- Format

Click on the **Format** menu tab, and the available formats include hexadecimal, decimal, binary, and ASCII.

- Position

Click on the **Position** menu tab to set the location.

- Label

Click on the **Label** to display the menu label, and choose to turn it on or off.

- Event Table

Click on the **Event Table** menu tab to enter the event table settings menu.

- Event Table
  The event table can be selected to open or close.
- Format
  Click on the **Format** menu tab to set the format of the event table. The options include hexadecimal, decimal, binary, and ASC.
- View
  Click on the **View** menu tab to select Package, Details, and Data.
- Export
  Click on the **Export** menu label, and when the view selects "package", information such as time and corresponding decoding data can be exported.
- Jump
  Click on the **Jump** menu tab to display the specified data waveform in the

center view of the screen.

6. Setting:

Click on the **Setting** menu tab to enter the Settings menu.

- Endian

Click on the **Endian** menu tab to select either MSB or LSB.

- LSB: Least Significant Bit, which means that the low bit of data is transmitted first.
- MSB: Most Significant Bit (most significant bit), that is, the high data bits are transmitted first.

- Polarity

Click on the **Polarity** menu tab to select the decoded data as either positive or negative polarity.

- Width

Click on the **Width** menu tab and select the bus width for decoding data through the pop-up menu.

7. Copy Trig:

Click on the **Copy Trig** menu tab to select the Copy Trigger function. The copy trigger function can quickly apply trigger settings to corresponding types of decoding functions.

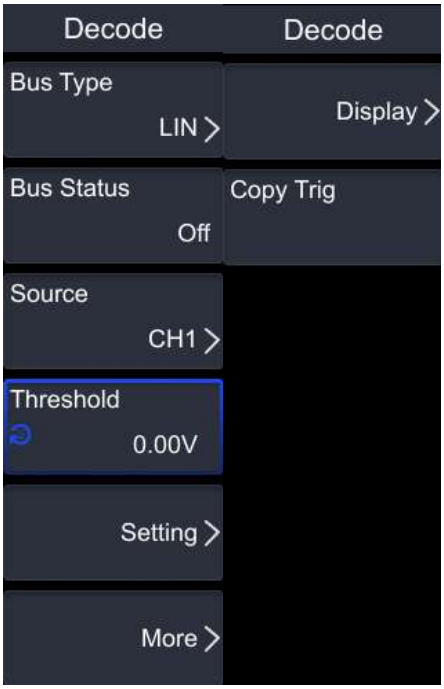
8. Decoding example:

Figure 14.7 SPI Decoding example

14.4 LIN Decoding

The oscilloscope samples the LIN signal and determines whether each data point is a logic "1" or a logic "0" based on the set threshold level. LIN decoding requires specifying the LIN signal protocol version.

Click **[Decode]** in the button area to enter the decoding function menu.



1. Bus Type:

Click on the **Bus Type** menu tab and select LIN from the pop-up menu to configure LIN decoding.

2. Bus Status:

Click on the **Bus Stauts** menu tab and choose to turn on (ON) or off (OFF) the bus switch function.

3. Source:

Click on the **Source** menu tab and select the source channel.

4. Threshold:

Click on the **Threshold** menu tab and set the threshold using the multi-functional knob V0 or the numeric keypad.
When changing the threshold of the clock channel, a dashed line appears on the screen displaying the current threshold level. When the change is stopped, the dashed line of the threshold level disappears after about 2 seconds.

5. Setting:

Click on the **Setting** menu tab to enter the Settings menu.

- Baud Rate:

Click on the **Baud Rate** menu tab and select Baud Rate from the pop-up menu.

- Parity Bit:

Click on the **Parity Bit** menu tab and select whether to include inspection positions or not.

- Version:

Click on the **Version** menu tab and select versions 1. x, 2. x, and Both.

6. Display:

Click on the **Display** Menu tab to enter the Display Settings menu.

- Format

Click on the **Format** menu tab, and the available formats include hexadecimal, decimal, binary, and ASCII.

- Position

Click on the **Position** menu tab to set the location.

- Label

Click on the **Label** to display the menu label, and choose to turn it on or off.

- Event Table

Click on the **Event Table** menu tab to enter the event table settings menu.

  - Event Table
    The event table can be selected to open or close.
  - Format
    Click on the **Format** menu tab to set the format of the event table. The options include hexadecimal, decimal, binary, and ASC.
  - View
    Click on the **View** menu tab to select Package, Details, and Data.
  - Export
    Click on the **Export** menu label, and when the view selects "package", information such as time and corresponding decoding data can be exported.
  - Jump
    Click on the **Jump** menu tab to display the specified data waveform in the center view of the screen.

7. Copy Trig:

Click on the **Copy Trig** menu tab to select the Copy Trigger function. The copy trigger function can quickly apply trigger settings to corresponding types of decoding functions.

8. Decoding example:

- Break (Synchronous Break): Can be displayed in hexadecimal, decimal, binary, or ASCII format, represented by a black block.
- SYNC (Synchronous): Can be displayed in hexadecimal, decimal, binary, or ASCII format, represented by a black block; when an error occurs, it is represented by a red block.
- ID (Frame ID): Can be displayed in hexadecimal, decimal, binary, or ASCII format, represented by a black block; when an error occurs, it is represented by a red block.
- Data (Data): Can be displayed in hexadecimal, decimal, binary, or ASCII format, represented by a black block.
- CRC (Cyclic Redundancy Check): Can be displayed in hexadecimal, decimal, binary, or ASCII format, represented by a black block; when an error occurs, it is represented by a red block.

Figure 14.8 LIN Decoding example

9. Error Representation During Decoding:

LIN decoding may encounter parity errors, checksum errors, or synchronization errors.

- Synchronization error: If a synchronization error is detected, SYNC (Synchronization) will be displayed in red, as shown in the figure below.

Figure 14.9 LIN Decoding synchronization error

- Parity Error: If a parity error is detected, the Frame ID and parity bit will be displayed in red, as shown in the figure below.

Figure 14.10 LIN Decode parity error

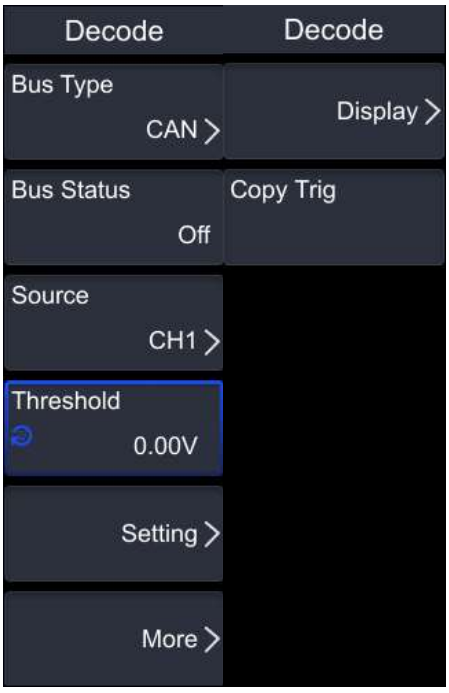
- Checksum Error: If a checksum error is detected, CRC (Cyclic Redundancy Check) will be displayed in red, as shown in the figure below.

Figure 14.11 LIN Decoding Calibration and Errors

14.5 CAN Decoding

The oscilloscope samples CAN or CAN-FD signals at the designated sampling position, and will also determine whether each data point is a logic "1" or a logic "0" based on the set threshold level. CAN decoding requires specifying the CAN or CAN-FD signal type and sampling location.

Click **[Decode]** in the button area to enter the decoding function menu.



1. Bus Type:

Click on the **Bus Type** menu label and select CAN from the pop-up menu to configure CAN decoding.

2. Bus Status:

Click on the **Bus Status** menu tab and choose to turn on (ON) or off (OFF) the bus switch function.

3. Source:

Click on the **Source** menu tab and select the source channel.

4. Threshold:

Click on the **Threshold** menu tab and set the threshold using the multi-functional knob V0 or the numeric keypad.
When changing the threshold of the clock channel, a dashed line appears on the screen displaying the current threshold level. When the change is stopped, the dashed line of the threshold level disappears after about 2 seconds.

5. Setting:

Click on the **Setting** menu tab to enter the Settings menu.

- Signal:

Click on the **Signal** menu label, and the signal types that can be selected are: CAN_ H, CAN_ L, Rx, Tx, Diff.

  - CAN_ H: Actual CAN_ H-bus signal.
  - CAN_ L: Actual CAN_ L-bus signal.
  - Rx: Received signal from CAN signal line.
  - Tx: Sending signal from CAN signal line.
  - Diff: Use a differential probe to connect to the CAN differential bus signal of the analog channel. Positive connection CAN of differential probe_ H-bus signal, negative connection CAN of differential probe_ L-bus signal.

- Baud:

Click on the **Baud** menu tab and select Baud Rate from the pop-up menu.

- Sample Position:

Click on the **Sample Position** menu label and select versions 1. x, 2. x, and Both.

6. Display:

Click on the **Display** Menu tab to enter the Display Settings menu.

- Format

Click on the **Format** menu tab, and the available formats include hexadecimal, decimal, binary, and ASCII.

- Position

Click on the **Position** menu tab to set the location.

- Label

Click on the **Label** to display the menu label, and choose to turn it on or off.

- Event Table

Click on the **Event Table** menu tab to enter the event table settings menu.

- Event Table
  The event table can be selected to open or close.
- Format
  Click on the **Format** menu tab to set the format of the event table. The options include hexadecimal, decimal, binary, and ASC.
- View
  Click on the **View** menu tab to select Package, Details, and Data.
- Export
  Click on the **Export** menu label, and when the view selects "package", information such as time and corresponding decoding data can be exported.
- Jump
  Click on the **Jump** menu tab to display the specified data waveform in the center view of the screen.

7. Copy Trig:

Click on the **Copy Trig** menu tab to select the Copy Trigger function. The copy trigger function can quickly apply trigger settings to corresponding types of decoding functions.

8. Decoding example:

Figure 14.12 CAN Decoding example

15 Protocol

This series of machines support a variety of protocol generator functions, click the navigation icon **[Hantek]** in the upper left corner of the screen, select **Protocol**, enter the setting menu.

The contents of this chapter include:

15.1 UART Protocol

Click the navigation icon **[Hantek]** to open the protocol generator menu.

1. Type:

Click the **Type** menu tab and select the "UART" type for specific Settings.

After the protocol generation type is selected, the menu changes to the UART Setting menu.

2. Method:

Click the **Method** menu tab to select the following modes: SigFixedValue and SigRandomValue.

3. SendData:

Click on the **SendData** menu tab to set the data value by rotating the multifunction knob V0 or directly set the send data value by popping up the numeric keypad.

4. **DataBits:**

Click the **DataBits** menu tab and select data bits from the drop-down menu: 4bits, 5bits, 6bits, 7bits, 8bits.

5. **Bandrate:**

Click the **Baudrate** menu tab and select the desired Baud rate from the drop-down menu or select Custom Baud rate.

6. **Idle Level:**

Click the **Idle Level** menu tab to select the idle level as High or Low.

7. **Parity Check:**

Click the **Parity Check** tab to select the check mode as Null, Odd check, or Even check.

8. **Trigger Mode:**

Click the **Trigger Mode** menu tab and select Auto or Manual.

9. **Send Interval:**

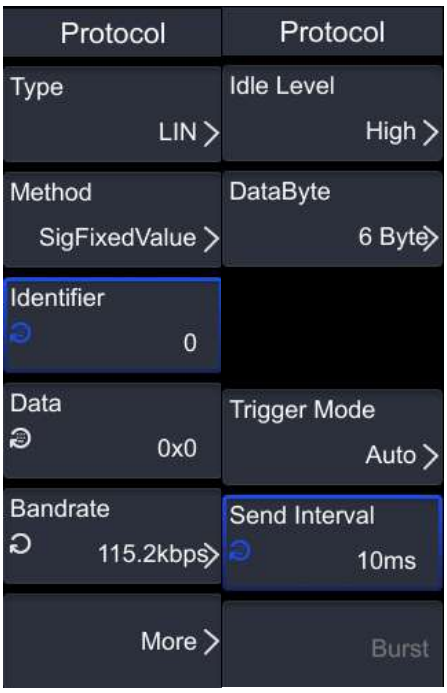
Click on the **Send Interval** menu tab to set the interval value by turning the multifunction knob V0 or directly set the interval value by popping up the numeric keypad.

10. **Burst:**

When the trigger mode is set to manual trigger, click the **Burst** menu tab to manually trigger once.

15.2 LIN Protocol

Click the navigation icon **[Hantek]** to open the protocol generator menu.



1. Type:

Click on the **Type** menu tab and select the "LIN" type for specific Settings.
After selecting the protocol generation type, the menu changes to the LIN Settings menu.

2. Method:

Click the **Method** menu tab to select the following modes: SigFixedValue and SigRandomValue.

3. Identifier:

Click on the **Identifier** menu tab and set the identifier value by rotating the multifunction knob V0 or set the send identifier value directly on the popup numeric keypad.

4. Data:

Click on the **Data** menu tab to set the data values by turning the multifunction knob V0 or by the pop-up numeric keypad directly.

5. Baudrate:

Click the **Baudrate** menu tab and select the desired Baud rate from the drop-down menu or select Custom Baud rate.

6. Idle Level:

Click the **Idle Level** menu tab to select the idle level as High or Low.

7. DataByte:

Click on the **DataByte** menu tab and select from the drop-down menu: 1 Byte, 2 Byte, 3 Byte, 4 Byte, 5 Byte, 6 Byte, 7 Byte, 8 Byte.

8. Trigger Mode:

Click the **Trigger Mode** menu tab and select Auto or Manual.

9. Send Interval:

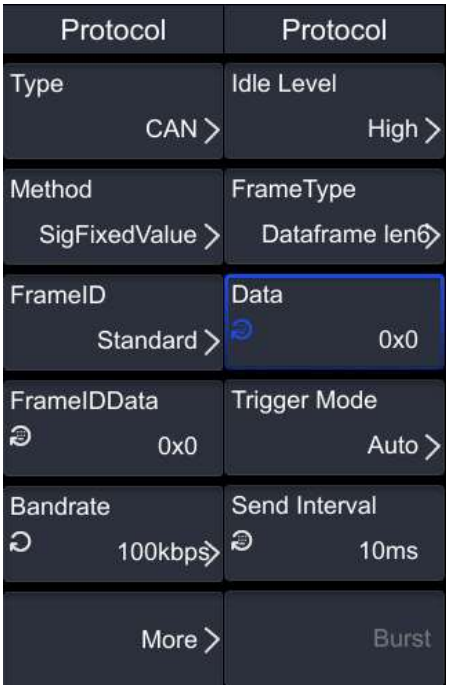
Click on the **Send Interval** menu tab to set the interval value by turning the multifunction knob V0 or directly set the interval value by popping up the numeric keypad.

10. Burst:

When the trigger mode is set to manual trigger, click the **Burst** menu tab to manually trigger once.

15.3 CAN Protocol

Click the navigation icon **[Hantek]** to open the protocol generator menu.



1. Type:

Click the **Type** menu tab and select the "CAN" type for specific Settings.
After the protocol generation type is selected, the menu changes to the CAN Settings menu.

2. Method:

Click the **Method** menu tab to select the following modes: SigFixedValue and SigRandomValue.

3. **FrameID:**

Click the **FrameID** menu tab, you can select the label ID as: standard, extended 29 bits.

4. **Frame ID data:**

Click on the **Frame ID data** menu tab to set the data value by rotating the multifunction knob V0 or directly set the frame ID data value by popping up the numeric keypad.

5. **Bandrate:**

Click the **Baudrate** menu tab and select the desired Baud rate from the drop-down menu or select Custom Baud rate.

6. **Idle Level:**

Click the **Idle Level** menu tab to select the idle level as High or Low.

7. **FrameType:**

Click the **Frame Type** menu tab and select from the drop-down menu: Data frame length 1-8, Remote frame, Error frame, Overload frame.

8. **Data:**

Click on the **Data** menu tab to set the data values by turning the multifunction knob V0 or by the pop-up numeric keypad directly.

9. **Trigger Mode:**

Click the **Trigger Mode** menu tab and select Auto or Manual.

10. **Send Interval:**

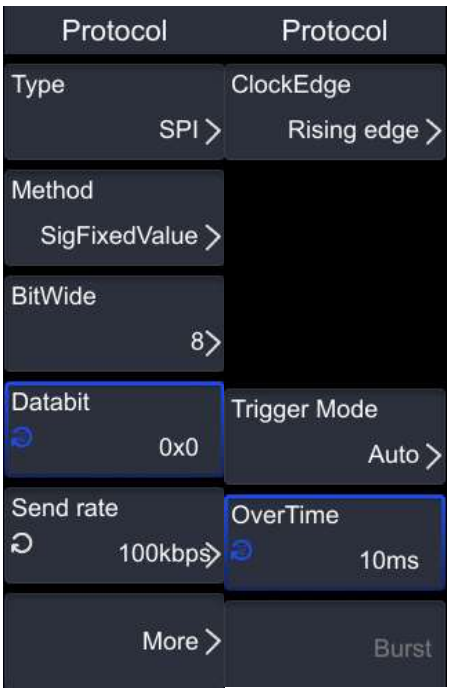
Click on the **Send Interval** menu tab to set the interval value by turning the multifunction knob V0 or directly set the interval value by popping up the numeric keypad.

11. **Burst:**

When the trigger mode is set to manual trigger, click the **Burst** menu tab to manually trigger once.

15.4 SPI Protocol

Click the navigation icon **[Hantek]** to open the protocol generator menu.



1. Type:

Click the **Type** menu tab and select the "SPI" type for specific Settings.
After the protocol generation type is selected, the menu changes to the SPI Settings menu.

2. Method:

Click the **Method** menu tab to select the following modes: SigFixedValue and SigRandomValue.

3. BitWide:

Click the **BitWide** menu tab, and the data bit width can be selected from the drop-down menu: 4, 8, 16, 24, 32.

4. Databit:

Click on the **Databit** menu tab to set the data values by turning the multifunction knob V0 or by the pop-up numeric keypad directly.

5. Send rate:

Click the **Send rate** menu tab and select the desired send rate from the drop-down menu or select Custom send rate.

6. ClockEdge:

Click the **ClockEdge** menu tab to select the clock edge as a rising or falling edge.

7. Trigger Mode:

Click the **Trigger Mode** menu tab and select Auto or Manual.

8. OverTime:

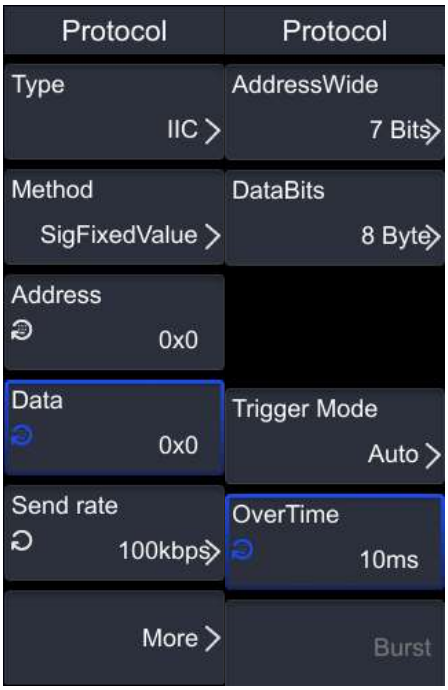
Click on the **OverTime** menu tab to set the interval value by turning the multifunction knob V0 or directly set the interval value by popping up the numeric keypad.

9. Burst:

When the trigger mode is set to manual trigger, click the **Burst** menu tab to manually trigger once.

15.5 IIC Protocol

Click the navigation icon [Hantek] to open the protocol generator menu.



1. Type:

Click the **Type** menu tab and select the "IIC" type for specific Settings.
After the protocol generation type is selected, the menu changes to the IIC Settings menu.

2. Method:

Click the **Method** menu tab to select the following modes: SigFixedValue and SigRandomValue.

3. Address:

Click on the **Address** menu tab and set the address by turning the multifunction knob V0 or the pop-up numeric keypad directly.

4. **Data:**

Click on the **Data** menu tab to set the data values by turning the multifunction knob V0 or by the pop-up numeric keypad directly.

5. **Send rate:**

Click the **Send rate** menu tab and select the desired send rate from the drop-down menu or select Custom send rate.

6. **AddressWide:**

Click the **AddressWide** menu tab, and select the address bit width: 7bits, 10bits from the drop-down menu.

7. **DataBits:**

Click the **DataBits** menu tab, and select the data bits as follows: 1 Byte, 2 Byte, 3 Byte, 4 Byte, 5 Byte, 6 Byte, 7 Byte, 8 Byte from the drop-down menu.

8. **Trigger Mode:**

Click the **Trigger Mode** menu tab and select Auto or Manual.

9. **OverTime:**

Click on the **OverTime** menu tab to set the interval value by turning the multifunction knob V0 or directly set the interval value by popping up the numeric keypad.

10. **Burst:**

When the trigger mode is set to manual trigger, click the **Burst** menu tab to manually trigger once.

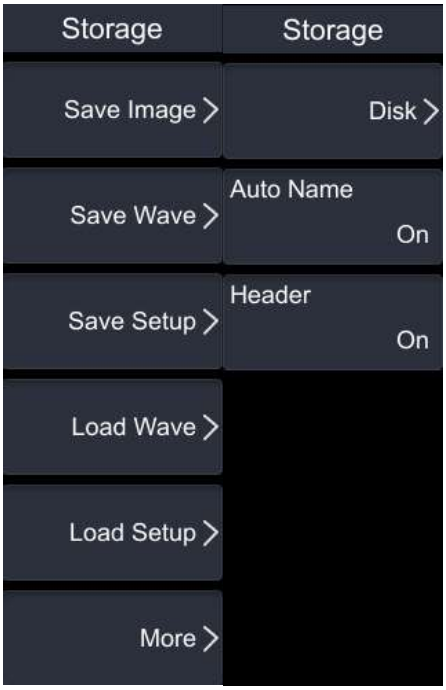
16 Store and Load

Users can save the current oscilloscope settings, waveforms, screen images, and parameters in various formats to internal memory
Or an external USB storage device (such as a USB flash drive), and can reload saved settings or waveforms when needed.
In addition, users can perform replication on specified types of files in internal storage or external USB drives through the disk management menu
Operations such as creating, deleting, and renaming.

This chapter includes:

16.1 Storage Type

Click the **[Save/Recall]** button in the button area to open the save/recall menu.



11. Image storage:

Click on the **Save Image** menu tab to enter the Image Storage Settings menu.

- Format:

Click on the **Format** menu tab, and the types you can choose include: *. png, *. bmp, *.

ipg, and *. tif. Image storage will be stored in the selected type to the internal or external memory of the machine.

- File Name:

Click on the **File Name** menu tab and set the file name on the pop-up keyboard. Please refer to the NewFolder for the file name input method. Press the **More** -> **Auto Name** key to automatically generate a file name. If a file name already exists and is saved with the same file name again, a prompt box will pop up to prompt the user that the file will be overwritten. Do you want to continue.

- Invert:

Click on the **Invert** menu tab to select the ON or OFF image anti color function.

- Color:

Click on the **Color** menu tab to choose whether to save the image in color or gray.

- NewFolder:

Click on the **NewFolder** menu tab and set the directory name on the pop-up keyboard.

- Window:

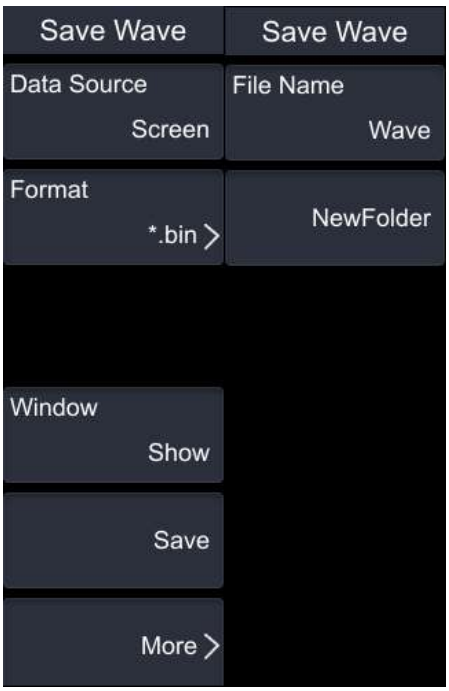
Click on the **Window** menu tab and choose to open or close the window.

- Save:

Click on the **Save** menu tab to save the set image in internal or external memory. Press the **More** -> **Header**. If the header display is turned on, when saving the image file, the instrument model and image construction date will be displayed in the header of the image.

12. Save Wave:

Click on the **Save Wave** menu tab to enter the waveform storage settings menu. The waveform storage can save the oscilloscope's setting information (channel switch settings, vertical gear, horizontal time base, etc.) and open the waveform data of the channel.



- Data Source:

Click on the **Data Source** menu tab and select Screen or Memory.

- Format:

Click on the **Format** menu tab, and the types you can choose are: *. bin, *. csv.

- Source:

Click on the **Source** menu tab to select types such as CH1, CH2, CH3, CH4, D0-D3, D4-D7, D8-D11, and D12-D15. The resource menu can only be set when selecting the data source as memory.

- Window:

Click on the **Window** menu tab and choose to open or close the window.

- File Name:

Click on the **File Name** menu tab and set the file name on the pop-up keyboard. Please refer to the NewFolder for the file name input method. Press the **More** -> **Auto Name** key to automatically generate a file name. If a file name already exists and is saved with the same file name again, a prompt box will pop up to prompt the user that the file will be overwritten. Do you want to continue.

- NewFolder:

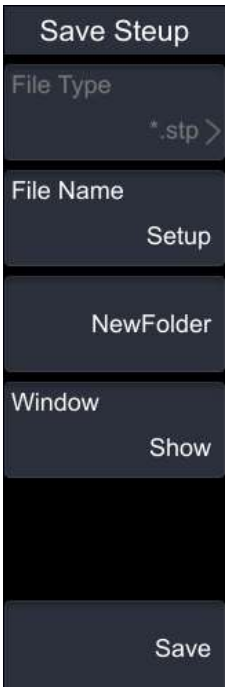
Click on the **NewFolder** menu tab and set the directory name on the pop-up keyboard.

- Save:

Click the **Save** menu tab to save the set waveform in internal or external memory.

13. Save Setup:

Click on the **Save Setup** Menu tab to enter the Set Storage Settings menu. Set Storage will save the oscilloscope settings to internal or external memory.



- File Type:

Click on the **File Type** menu tab to select types such as *. stp and *. ptr.

- File Name:

Click on the **File Name** menu tab and set the file name on the pop-up keyboard. Please refer to the NewFolder for the file name input method. Press the **More** -> **Auto Name** key to automatically generate a file name. If a file name already exists and is saved with the same file name again, a prompt box will pop up to prompt the user that the file will be overwritten. Do you want to continue.

- NewFolder:

Click on the **NewFolder** menu tab and set the directory name on the pop-up keyboard.

- Window:

Click on the **Window** menu tab and choose to open or close the window.

- Save:

Click on the **Save** menu tab to save the settings in internal or external memory.

16.2 Load Type

Load files stored inside or outside the machine into the machine. Select waveform loading or set loading, and the file path will pop up on the screen.

1. **Load Wave:**

Click on the **Load Wave** menu tab to enter the waveform loading settings menu.

- File Type:

Click on the **File Type** menu label, with the type: *. wfm.

- Load:

Click on the **Load** menu tab to load the selected file.

2. **Load Setup:**

Click on the **Load Setup** Menu tab to enter the Settings Load Settings menu.

- File Type:

Click on the **File Type** menu tab to select types such as *. stp and *. ptr.

- Load:

Click on the **Load** menu tab to load the selected file.

16.3 External Storage and Load

The first step for external storage of an oscilloscope is to ensure that the USB flash drive is correctly connected. The external storage function supports image, waveform, setting storage, and loading of waveforms and settings.

1. **For example, storing the waveform of a specified type of file in an external USB drive:**

- Oscilloscope obtains signal;
- Open the waveform storage function and enter the storage settings menu;
- Open the menu interface for external storage through multi-functional knobs or touch screen functions, and select the path to store. In addition, a new storage path can be created in external storage through the function of creating a new directory;
- In the storage menu, select the storage type as waveform storage. For specific information on waveform storage, please refer to the relevant chapters;

- Click **Save**, and the waveform file will be stored in the selected directory in the set save format.

2. **For example, to load the waveform of a specified type of file from an external USB drive:**

- Open the waveform storage function and enter the waveform loading interface;
- Open the menu interface of external storage through multi-functional knobs or touch screen functions, and select the loading path;
- After selecting the file to load, click **Load** to load the selected file.

16.4 Internal Storage and Load

The internal storage function supports the storage and loading of images, waveforms, settings, reference waveforms, pass/fail test rule files.

1. **For example, saving the specified type of file settings file in internal storage:**

- Oscilloscope obtains signal;
- Open the storage function menu, select Set Storage, and the disk management interface will pop up. By default, "XX" is selected;
- Open the internal storage interface through multifunctional knobs or touch screen functions;
- Open the menu interface for external storage through multi-functional knobs or touch screen functions, and select the path to store. In addition, a new storage path can be created in external storage through the function of creating a new directory;
- Click on the **File Name** to name the stored settings file. Click **More** -> **Auto Name**, and the file name will be automatically generated. If the file name is duplicate and already exists, a prompt box will pop up.

2. **For example, loading the settings file for internal storage:**

- Open the storage function menu, select Set Storage, and the disk management interface will pop up. By default, "XX" is selected
- Open the internal storage interface through multifunctional knobs or touch screen functions;
- Open the menu interface for external storage through multi-functional knobs or

touch screen functions, and select the path to store;

- Select the file to load through the multi-functional knob or touch screen function, and click **Load** to load the selected file.

16.5 Disk Management

Click the **[Save/Recall]** button in the button area to open the save/recall menu. Click on **Disk** menu to open the disk management interface. Select the disk by rotating the multi-functional knob V0 or touching the function.

Through the disk management menu, you can perform the following operations:

- Choose a file type
- Copy and paste files or directories
- Rename files or directories
- Delete files or directories
- Create a new directory
- Safely clear internal memory

1. File Type

Click on **[Save/Recall]** -> **Disk** -> **File Type**, and rotate the multi-functional knob V0 to select the desired file type. The default is' *. * '. Only files with a file name suffix that matches the selected file type will be displayed in the current path.

2. Copy

Press **[Save/Recall]** -> **Disk**, rotate the multi-functional knob V0 to select internal or external storage, and press this knob to open it. Rotate the multi-functional knob V0 to select the file or directory to be copied. Press the **Copy** key to copy the selected file or directory.

3. Paste

Press **[Save/Recall]** -> **Disk**, rotate the multi-functional knob V0 to select internal or external storage, and press this knob to open it. Rotate the multi-functional knob V0 to select the file or directory to be pasted. Press the **Paste** button to paste the selected file or directory.

4. Rename

Press **[Save/Recall]** -> **Disk**, rotate the multi-functional knob V0 to select internal or external storage, and press this knob to open it. Rotate the multi-functional knob V0 to select the file or directory that needs to be renamed. Press the **Rename** button to open the file name input interface. Please refer to the NewFolder for the file name input method.

5. Delete

- Delete files or directories from internal storage
  - Press **[Save/Recall]** -> **Disk**, rotate the multifunction knob to select the internal storage Local Disk (C), and press the knob to open it.
  - Rotate the multi-functional knob V0 to select the file or directory to be deleted.
  - Press the **Delete** button and in the pop-up delete confirmation box, click **OK** to delete the file or directory.
- Delete files or directories from external storage
  - Press **[Save/Recall]** -> **Disk**, rotate the multifunction knob to select external memory, and press the knob to open it.
  - Rotate the multi-functional knob V0 to select the file or directory to be deleted.
  - Press the **Delete** button and in the pop-up delete confirmation box, click **OK** to delete the file or directory.

6. NewFolder

Before using external storage, please ensure that the USB flash drive (FAT32 format, Flash type) is properly connected. In the disk management interface, select the storage path through the multi-functional knob V0 or use the touch screen function, and select Internal Storage Local Disk (C) by default. Then press the **NewFolder** button to open the interface shown in the following figure.

Figure 16.1 To Create a Folder

This oscilloscope supports both Chinese and English input methods. Below are instructions on how to input the name of a directory using both Chinese and English input methods.

- Create English Name:

➢ Choose English input method

If the current input method is En/中, please skip to the next step; If the current input method is 中/En, click this button to switch the input method to En/中, with "En" arranged first.

➢ Clear the name input area

If the current "Name Input Area" does not contain characters, please skip to

the next step; If the current "Name Input Area" contains characters, click the delete button to delete all characters in the name input area in sequence.

➢ Input the first character 'N'

✧ Currently in uppercase, please skip to the next small step; If lowercase is currently displayed, select the case switch button to switch the characters to uppercase.

✧ If "F" is selected and the knob is pressed, the character will appear in the input area.

➢ Input other characters

Refer to the previous step and use the same method to input the remaining characters 'ame' in sequence, paying attention to capitalization during the input process.

➢ Modify or delete inputted characters

During the name input process, you can modify or delete the characters that have already been entered. To delete a character that has already been entered, select the delete key and press to delete the character. If you modify a character that has already been entered, delete the character and re-enter the desired character. If the modified character is in the middle, it needs to be deleted from the last character forward (left) to the character that needs to be modified, and then re-enter the character.

➢ After completing the input, click the completion button, and the oscilloscope will create a specified type of file or directory under the current path with that file name.

- Create a Chinese name File name:

➢ Select Chinese input method

If the current input method is Medium/En, please skip to the next step; If the current input method is En/Medium, click this button to switch between input methods, so that the input method is Medium/En and the middle is arranged first.

- Clear the Pinyin input area and name input area

  If the current Pinyin input area and name input area do not contain characters, please skip to the next step; If the current pinyin input area and name input area contain characters, click the delete button to delete all characters in the name input area in sequence.

- Input the first Chinese character 'text'

  - Input pinyin wen in the pinyin input area
    Click on the first letter "w" in Pinyin wen to enter "w" in the Pinyin input area. Use the same method to input the remaining letters "en" in Pinyin wen in sequence. After the input is completed, optional Chinese characters will be displayed in the Chinese character selection area.
  - Select the desired Chinese character in the Chinese character selection area
    Click on "text" in the Chinese character selection area to input text in the name input area. If the desired Chinese character is not on the current page, please press the switch to next page key to switch to the next page selection.

- Input other characters

  Refer to the previous step and use the same method to input the remaining Chinese characters' part names' in sequence.

- Modify or delete inputted characters

  During the input process, you can modify or delete the inputted characters in the name input area and delete the inputted pinyin characters in the pinyin input area. To delete the entered characters, select the delete key on the virtual keyboard and press to delete the characters. If you modify a character that has already been entered, delete the character and re-enter the desired character. If the modified character is in the middle, it needs to be deleted from the last character forward (left) to the character that needs to be modified, and then re-enter the character.

- After completing the input, click the completion button, and the oscilloscope will create a specified type of file or directory under the current path with that file name.

7. SecurityClear

Press **[Save/Recall]** -> **Disk**, rotate the multifunction knob to select Internal Memory

Local Disk (C).

Press the **SecurityClear** button to pop up a security clear confirmation dialog box.

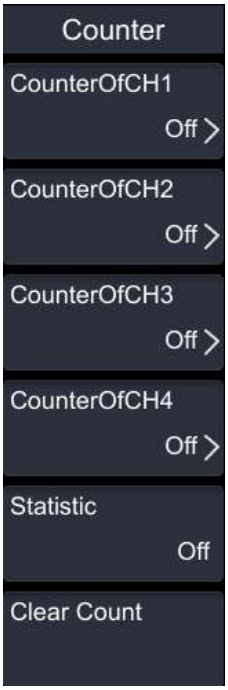
Press the **OK** key to clear all stored files on the internal memory.

17 Analyze

This chapter includes:

17.1 Frequency Counter

The frequency meter analysis function can provide counting and measurement of frequency, period, or edge events on any analog channel. You can enter the frequency meter settings menu by using the following methods:



- Click on the button area **[Analyze]** to select the frequency **counter** function.
- Click on the navigation icon **[Hantek]** in the upper left corner of the screen, select the frequency counter, and enter the settings menu.

After enabling the frequency meter function, enter the frequency meter settings menu and select the signal source and measurement item.

1. Select measurement source:

Select the desired measurement signal source and click on the corresponding frequency meter menu label: CH1 frequency meter, CH2 frequency meter, CH3 frequency meter, and CH4 frequency meter.

2. Select measurement item:

The measurement items supported by a frequency meter include Frequency, Period, and Totalize.
Totalize: Counting of signal edge events.

3. Statistics:

Click on the **Statistic** menu tab and select on or off. After selecting to open statistics, a window with statistical results will appear on the left side of the screen

4. Clear Count:

When the measurement item is selected to accumulate and measure the count of signal edge events, clicking the **Clear Count** button can reset the measurement result to zero.

17.2 Digital Voltmeter (DVM)

The digital voltmeter (DVM) of this series of digital oscilloscopes can measure the voltage of three significant digits on any analog channel. The collection system of DVM measurement and oscilloscope is asynchronous and always performs collection.
The voltmeter setting menu can be accessed through the following methods:

- Click the key area **[Analyze]** -> **DVM** to enter the DVM setting menu.
- Click the navigation icon **[Hantek]** in the upper left corner of the screen, select the digital voltmeter, and enter the setting menu.

1. DVM:

Select the desired measurement signal source and click on the corresponding voltmeter menu label: CH1 voltmeter, CH2 voltmeter, CH3 voltmeter, CH4 voltmeter.

Attention: The unit displayed on the voltmeter measurement should be consistent with the channel unit.

2. **Mode:**

The selectable modes of the voltmeter include AC RMS, DC, and AC+DC RMS.

- AC RMS: Display the root mean square value of the DC component in the collected mode.
- DC: Display the average value of the collected data.
- AC+DC RMS: displays the root mean square value of the collected data.

3. **Limits:**

Click on the **Limits** menu tab to enter the limit setting menu.

- Source:

Click on the **Source** menu tab, select the desired source, and choose the analog channels CH1-CH4.

- Beeper:

Click on the **Beeper** menu tab and choose to turn the buzzer on or off.

- Limit conditions:

Click on the menu label of **When** to select whether the In Limits or Out Limits.

  - In Limits: Enable or disable the buzzer when the voltage value is within the set limit range.
  - Out Limits: Enable or disable the buzzer when the voltage value is outside the set limit range.

- Upper and Lower values:

Set the upper or lower limit of the voltage limit value through the multi-functional knob V0 or the pop-up numeric keyboard.

4. **Buzzer:**

Click on the **Buzzer** menu tab and choose to turn the buzzer on or off.

17.3 Power Analysis (Option)

The DPO8000 series oscilloscope is equipped with power analysis function, which can help users quickly and easily analyze the efficiency and reliability of switching power supplies. Using the power analysis function, you can analyze the power quality of the input power supply and output ripple noise. You can enter power analysis through the following methods:

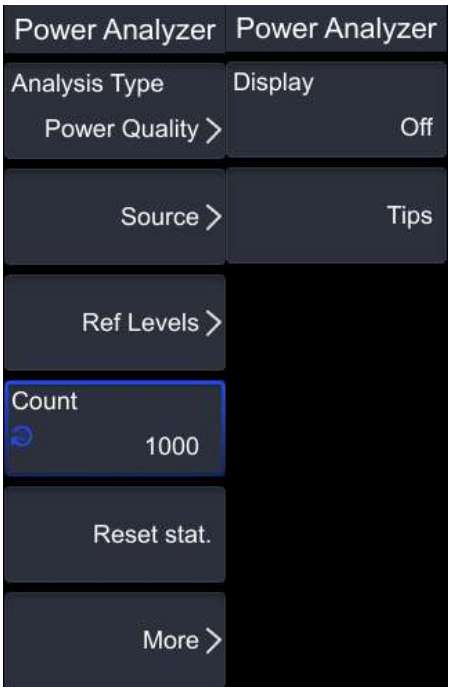
- Click on the button area **[Analyze]** -> **Power Analyzer** to enter the Power Analysis Settings menu.
- Click on the navigation icon **[Hantek]** in the upper left corner of the screen, select

Power Analyzer, and enter the settings menu.

17.3.1 Power Quality

Analyzing power quality can test the quality of AC input lines. The specific measurement parameters for power quality analysis include the measurement of electrical parameters such as voltage effective value, current effective value, active power, apparent power, reactive power, power factor, reference frequency, phase angle, impedance, voltage peak factor, and current peak factor at the input end of the power supply

In the Power Analysis menu, click the **Analysis Type** menu tab and select the Power Quality option.



1. Source:

Click on the **Source** menu tab to set the power quality source. You can choose voltage channel, current channel, and frequency reference.

- Voltage channel:

Click on the **Voltage** menu label and select the channels CH1~CH4 for collecting voltage.

- Current channel:

Click on the **Current** menu tab and select the channels CH1~CH4 for collecting current.

- Frequency reference:

Click on the **Fre.Ref** menu tab and set the frequency to be referenced by the voltage channel or current channel.

2. Display:

Click on the **Display** menu tab to choose whether to turn on or off the display results. When opened, the results are displayed on the screen, as shown in the following figure.

Power Quality

	Curr	Avg	Min	Max	Dev	Cnt
V_RMS(1)	****	****	****	****	****	0
I_RMS(2)	****	****	****	****	****	0
Real Power(1)	****	****	****	****	****	0
Apparent Power(1-2)	****	****	****	****	****	0
Reactive Power(1-2)	****	****	****	****	****	0
Power Factor(1-2)	****	****	****	****	****	0
Frequency(2)	****	****	****	****	****	0
Phase Angle(1-2)	****	****	****	****	****	0
Impedance(1-2)	****	****	****	****	****	0
V Crest Factor(1)	****	****	****	****	****	0
I Crest Factor(2)	****	****	****	****	****	0

3. Wiring diagram:

Click on the **Tips** menu label, and a wiring diagram for power quality analysis will pop up on the screen.

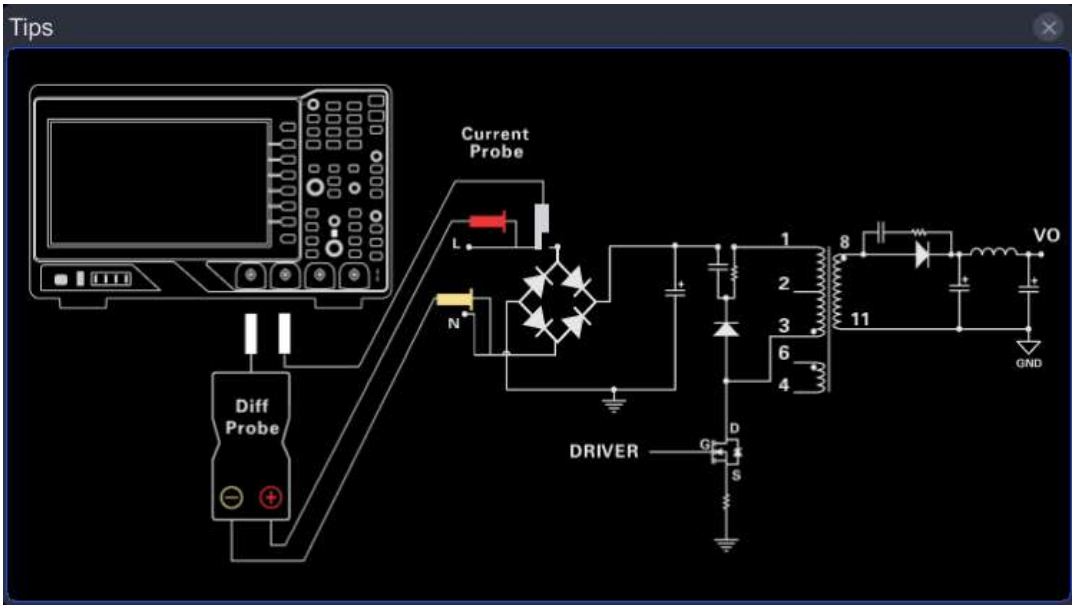


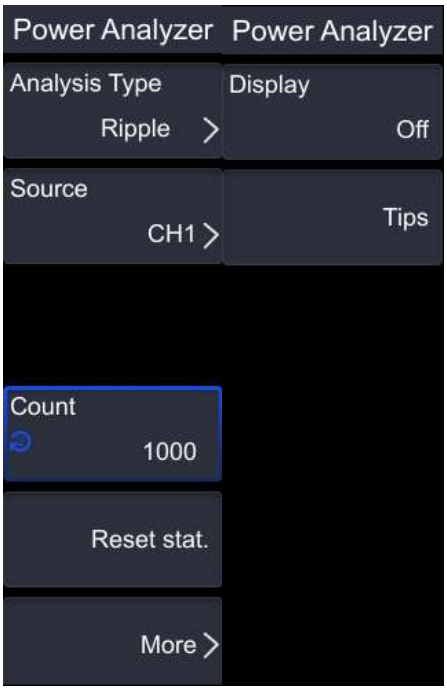
Figure 17.1 Power Analysis Wiring Diagram

17.3.2 Ripple

Power ripple is an important parameter for evaluating DC power supplies, representing the fluctuation of output DC voltage. Ripple analysis can measure the current value, average value, minimum value, maximum value, standard deviation, and count value of the ripple at the power output end.

In the Power Analysis menu, click the **Analysis Type** menu tab and select the Ripple

option.



1. Source:

Click on the **Source** menu tab and select the ripple analysis sources CH1~CH4.

2. Count:

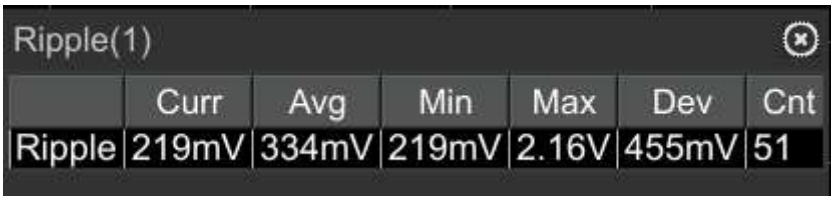
Click on the **Count** menu tab, and you can set the frequency meter by rotating the knob V0 or directly set the counting on the pop-up numeric keyboard.

3. Reset:

Click on the **Reset stat.** menu tab to clear the current data and recalculate the measurement results.

4. Display:

Click on the **Display** menu tab to choose whether to turn on or off the display results. When opened, the results are displayed on the screen, as shown in the following figure.



5. Wiring diagram:

Click on the **Tips** menu label, and a wiring diagram for power quality analysis will pop up on the screen.

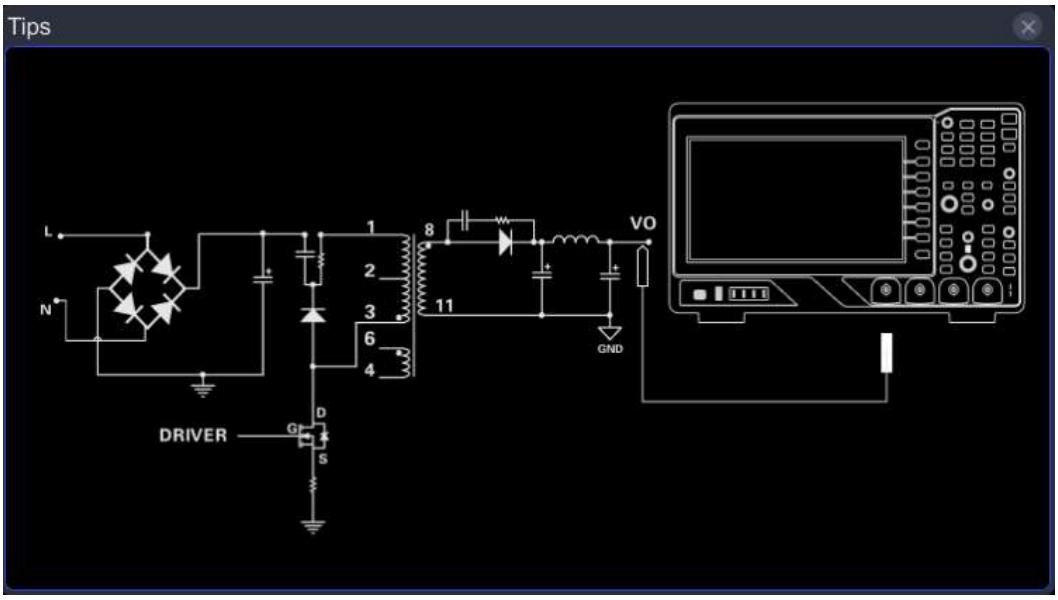
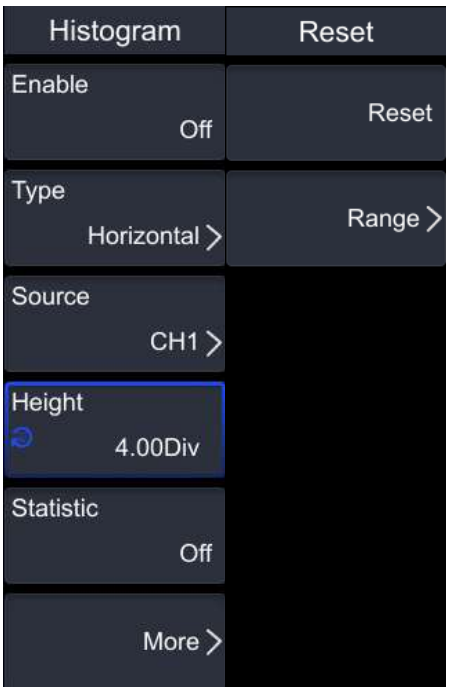


Figure 17.2 Ripple Wiring Diagram

17.4 Histogram Analysis

The analysis histogram function of this series of oscilloscopes can provide statistical views for waveforms or measurement results.
You can enter the histogram through the following methods:

- Click on the button area **[Analyze]** -> **Histogram** to enter the Histogram Settings menu.
- Click on the navigation icon **[Hantek]** in the upper left corner of the screen, select the histogram, and enter the settings menu.



17.4.1 Enable

Click on the **Enable** menu tab to turn on or off the histogram analysis function.

17.4.2 Type

Click on the **Type** menu tab to select the histogram types: Horizontal and Vertical.

- Horizontal histogram: A bar histogram at the bottom of the screen displays the number of times statistical data is displayed in columns.
- Vertical histogram: displays the number of times statistical data is displayed in rows in a bar histogram on the left side of the screen.

17.4.3 Source

Click on the **Source** menu tab, and you can select CH1~CH4 information sources. Only when the histogram type is horizontal or vertical histogram, it is necessary to select the source (CH1~CH4).

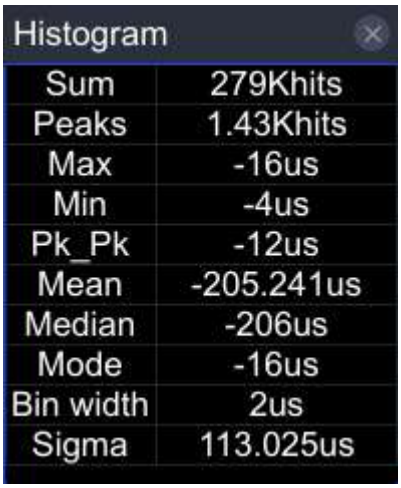
17.4.4 Histogram height

Click on the **Histogram** height menu tab and set the histogram height in the pop-up numeric keypad or rotate the knob V0 to set the histogram height, with a range of 1-4. The height of the histogram is the number of grids occupied by the histogram on the

screen.

17.4.5 Statistics

Click on the **Statistics** menu tab to select whether to turn on or off the statistics function. Turn on the statistics function, and the screen will pop up with the statistical results shown in the following figure.



- Sum: The number of times all data is counted.
- Peaks: The number of times the data is counted the most.
- Max: The maximum value of the histogram range.
- Min: The minimum value within the histogram range.
- Pk_Pk: The increment of the value corresponding to the highest number of data statistics and the value corresponding to the lowest number of data statistics.
- Mean: The average value corresponding to the histogram.
- Median: The median value corresponding to the histogram.
- Mode: The crowdsourcing value corresponding to the histogram.
- Bin width: The width corresponding to the histogram.
- Sigma: The standard deviation corresponding to the histogram.

17.4.6 Reset Statistics

Click the **Reset** menu tab to reset the statistical data to zero and perform statistics again.

17.4.7 Range settings

Click on the **Range** menu tab to enter the range setting menu. You can adjust the size and position of the histogram window by setting the left, right, upper, and lower boundaries. Set the boundary value by rotating the knob V0 or the pop-up numeric keyboard.
Window size and position can only be set for horizontal or vertical histograms.

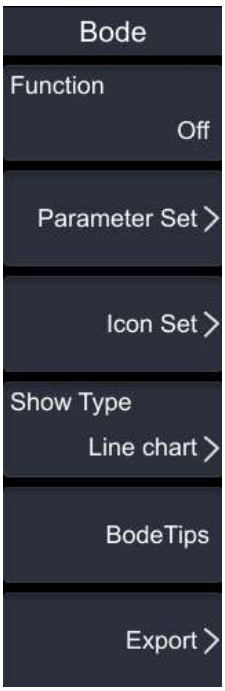
Note:

The adjustment of horizontal time base and vertical gear does not affect the time base size of the histogram range, but only shows synchronous changes with gear.

17.5 Bode Plot

Porter plot is a graphical method of describing the frequency response of a system. The baud chart is the curve that loop analysis can measure the gain and phase of a system as a function of frequency in a switching power supply and operational amplifier feedback network. The gain margin and phase margin of the system can be analyzed through the Bode diagram, and the stability of the system has been determined.
The DPO8000 series digital oscilloscope generates a sweep signal within a specified frequency range through a built-in signal generation module, and outputs it to the injection point of the tested switching power supply circuit. The oscilloscope tests the phase difference change curve and gain change curve of the injection and output ends at different frequencies to draw a Bode plot.
You can access the Bode plot by:

- Click on the button area **[Analyze]** -> **Bode Plot** to enter the Porter Chart Settings menu.
- Click on the navigation icon [Hantek] in the upper left corner of the screen, select the Porter chart, and enter the settings menu.



17.5.1 Enable

Click on the **Function** menu tab and choose to open or close the Porter chart.

17.5.2 Parameter settings

Click on the **Parameter Set** menu tab to enter the parameter setting menu.

1. **Input Source:**

Click on the **Input Source** menu tab to select CH1-CH4 as the input source. The input source is the channel of the input signal, which serves as the reference for the current frequency. Before selecting an input source, it is necessary to connect the measured signal to the analog channel input of the oscilloscope.

2. **Output Source:**

Click on the **Output Source** menu tab to select CH1-CH4 as the output source. The output source is the channel that connects the feedback output signal. Before selecting the output source, it is necessary to connect the measured signal to the analog channel output terminal of the oscilloscope.

3. **Minimum frequency:**

Click on the **Min Freq** menu tab, and the frequencies you can select are: 10Hz, 100Hz, 1KHz, 10KHz, 100KHz, 1MHz, and 10MHz.

4. **Maximum frequency:**

Click on the **Max Freq** menu tab, and the frequencies you can select are: 100Hz, 1KHz, 10KHz, 100KHz, 1MHz, 10MHz, and 25MHz.

17.5.3 Chart Settings

Click on the **Icon Set** menu tab to enter the Chart Settings menu.

1. **Gain gear:**

Click on the **Gain Scale** menu tab and adjust the gain gear through the multi-functional knob V0.

2. **Gain offset:**

Click on the **Gain Off** menu tab and adjust the gain offset through the multi-functional knob V0.

3. **Phase Scale:**

Click on the **Phase Scale** menu tab and adjust the phase shift through the multi-functional knob V0.

4. **Phase offset:**

Click on the **Phase Off** menu tab and adjust the phase shift through the multi-functional knob V0.

17.5.4 Show Type

Click the **Show Type** menu tab to select Line Chart or Form.

17.5.5 Wiring Diagram

Click on the **Bode Tips** menu label, and a wiring diagram of the Bode diagram will pop up on the screen.

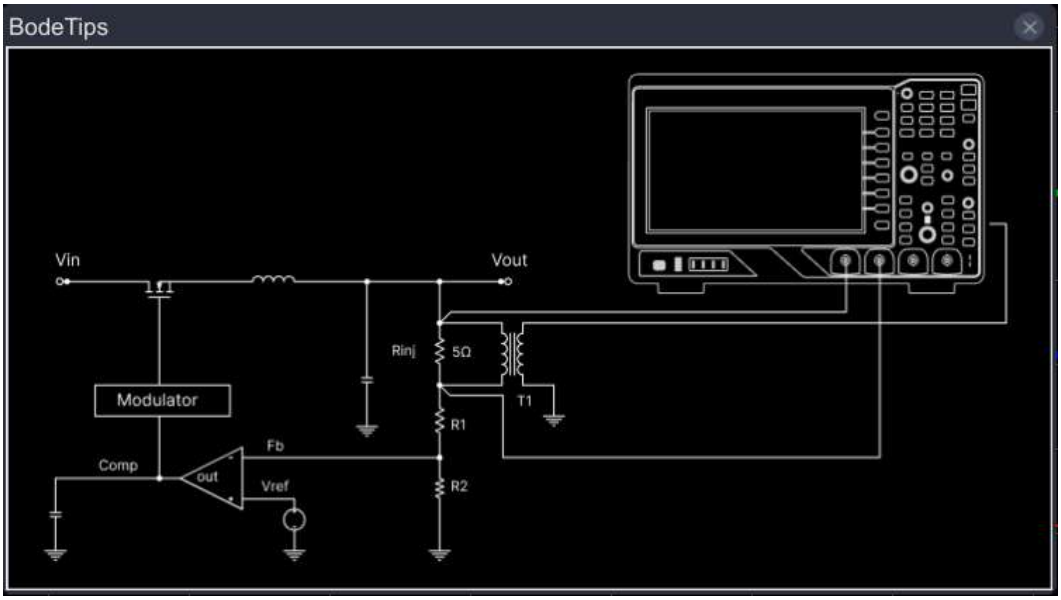


Figure 17.3 Bode diagram wiring diagram

18 Digital Channel

This series of machines are equipped with Logic analyzer function as standard, including 16 digital channels. The default channel label is D0-D15. The oscilloscope compares the voltage obtained from each sampling with the preset logical threshold. If the voltage is greater than the threshold, it is stored as a logical "1", and vice versa, it is stored as a logical "0". The oscilloscope displays the logical level values (1 and 0) of waveform points in a graphical manner, making it easy for users to detect and analyze errors in circuit design.

- Click the **[LA]** button in the front panel button area to enter the digital channel.
- By touching the digital channel symbol on the screen, you can enter the LA logic analysis setting menu.

This chapter includes:

18.1 Enable

Click the **Enable** menu tag and select to turn on or off the Logic analyzer function.

18.2 Source of Information

Click on the selection menu tab to open the channel menu, rotate the multifunction knob or touch screen function to select the channel.

- Select any channel from D1.0-D1.3, D2.0-D2.3, D3.0-D3.3, D4.0-D4.3, and the corresponding channel label and waveform will be displayed in red.
- Select a custom channel group, and the labels and waveforms of the channels in the selected channel group will be displayed in red.

Only open digital channels and grouped custom channel groups can be selected.

18.3 Type

Click on the **Type** menu tab and select the type of digital channel.

18.4 Threshold

Click on the **Threshold** menu tab to enter the threshold setting menu. If the voltage is greater than the threshold, it is stored as logical "1", and vice versa, it is stored as logical "0".

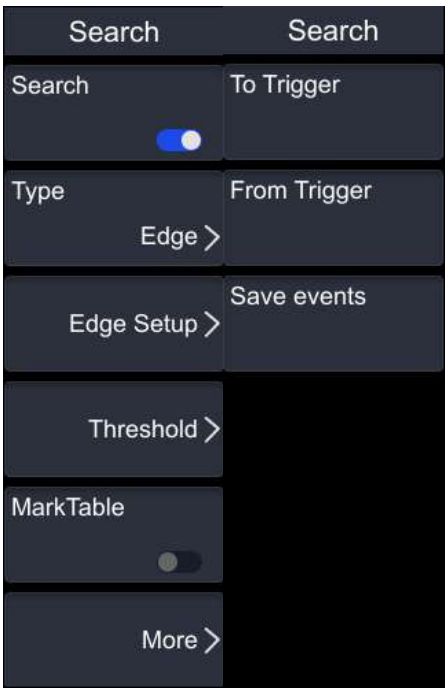
19 Search and Navigation Function

The search function allows users to quickly search for related events by setting search criteria. Navigation function includes recording, playing, and guiding Navigation, time navigation, and event navigation.

This chapter includes:

19.1 Search Function

The search function allows users to quickly search for related events by setting search criteria.
The search function can search for channel edges, pulse width triggering, under amplitude pulse triggering, slope triggering, RS232 triggering, I2C triggering, or SPI triggering.
Press the **[Search]** button on the front panel to open the search menu.



1. Search:

Click on the **Search** menu tab to turn on or off the search function.

2. Type:

Select the desired type, which can include Edge, Slope, Runt, Pulse, RS232, I2C, or SPI.

- Edge:

Select the search type as Edge, and refer to the detailed introduction of Edge Trigger for edge settings.
Threshold setting: Click on the **Threshold** setting menu label, rotate the multi-functional knob or the pop-up numeric keyboard to set the threshold.

- Slope:

Select the search type as Slope, and refer to the detailed introduction of Slope Trigger for slope settings.

Threshold setting: Click on the **Threshold** setting menu label, rotate the multi-functional knob or the pop-up numeric keyboard to set the threshold A and threshold B.

- Runt:

Select the search type as Runt, and refer to the detailed introduction of Runt Trigger for under amplitude settings.
Threshold setting: Click on the **Threshold** setting menu label, rotate the multi-functional knob or the pop-up numeric keyboard to set the threshold.

- Pulse:

Select the search type as Pulse, and refer to the detailed introduction of Pulse Trigger for pulse width settings.
Threshold setting: Click on the **Threshold** setting menu label, rotate the multi-functional knob or the pop-up numeric keyboard to set the threshold.

- RS232:

Select the search type as RS232, and refer to the detailed introduction of RS232 Trigger for RS232 settings.
Threshold setting: Click on the **Threshold** setting menu label, rotate the multi-functional knob or the pop-up numeric keyboard to set the threshold.

- I2C:

Select the search type as I2C trigger, and refer to the detailed introduction of I2C Trigger for I2C settings.
Threshold setting: Click on the **Threshold** setting menu label, rotate the multi-functional knob or the pop-up numeric keyboard to set the threshold.

- SPI:

Select the search type as edge trigger, and refer to the detailed introduction of SPI Trigger for edge settings.
Threshold setting: Click on the **Threshold** setting menu label, rotate the multi-functional knob or the pop-up numeric keyboard to set the threshold

3. Mark Table:

Click on the **Mark Table** menu tab, open or close the tag table, select Open Tag Table, and the tag table will appear on the screen.

4. To Trigger:

Click on the **To Trigger** Menu tab to copy the settings for the selected search type to the corresponding trigger type. For example, if the current search type is Edge, press Copy to Trigger to copy the edge search Settings to the Edge Trigger Settings.

5. From Trigger:

Click on the **From Trigger** Menu tab to copy the settings for the selected search type

into the search settings. To operate the copy self triggering function, you need to first set the search type. For example, if the current trigger type is Edge Trigger, press the Copy Self-trigger key to copy the edge trigger Settings to the Edge search Settings. Note: If you select the "Copy Trigger" function, you need to set the search type before you can copy the corresponding trigger type Settings from the trigger menu.

6. Save events:

Users can Save events marker data in internal or external storage. For specific operations, please refer to the relevant introduction in the Store and Load.

19.2 Navigation Function

Navigation functions include Recoding & Playing Navigation, Time navigation, and Event navigation.

1. Recoding & Playing Navigation:

After opening the waveform recording function and completing waveform recording, you can press the navigation combination key to play and navigate the recorded waveform. Pressing the  key plays the recorded waveform in reverse order, pressing the  key plays the recorded waveform in sequence, and pressing the  key stops playback.

2. Time Navigation:

After data collection stops, you can use the navigation key combination to quickly and continuously play the captured data waveform. Press the  key to play the waveform to the left, press the  key to play the waveform to the right, and press the  key to stop playing. Pressing the  key or the  key multiple times can accelerate the playback speed of the waveform and quickly locate the desired waveform.

Note:

The waveform of the time navigation function is in YT time base mode and the operating state is in STOP state.

3. Event Navigation:

After opening the search function and completing the event search, you can use the navigation combination keys to quickly navigate to specific events in the event marker table. Press the  key to navigate to the previous event (decreasing the number in the tag table), and press the  key to navigate to the next event (increasing the number in the tag table). Pressing the key does not have any effect.

20 Quik key

Automatic setting: Automatically set the oscilloscope controls to generate useful displays of input signals. Please refer to the table below for details.
Click the **[Auto Scale]** button to automatically set the oscilloscope.
Default Settings: Automatically call up the default settings.
Click the **[Default Setup]** button, and the machine will enter the default setting operation.

This chapter includes:

20.1 Auto Scale

Automatic setting is one of the advantages of digital oscilloscopes. When you click on automatic settings, the oscilloscope will recognize the waveform type and adjust the control method to accurately display the waveform of the input signal.

Table 20.1 Auto Scale

Parameter	Settings
Acquire	Normal
Cursor	Close
Timebase Mode	YT
Display	Vector
Horizontal	Adjusted
SEC/DIV	Adjusted
Holdoff	8ns
Trigger Level	50%
Trigger Mode	Auto
Trigger Source	Adjusted
Trigger Slope	Adjusted
Trigger Type	Edge
Sync	Adjusted
Standard	Adjusted
Bandwidth	Close
Coupling	DC
VOLTS/DIV	Adjusted

Click on **[Auto Scale]** to enable the automatic setting function and open the automatic

setting function menu.

1.  :Press the menu label, and the screen will automatically display the complete single cycle of the signal, while measuring the cycle and frequency of the currently displayed complete single cycle waveform. The measurement results are displayed at the bottom of the screen.
2.  :Press the menu label and the screen will automatically display multiple cycles of the signal, while measuring the cycle and frequency of the currently displayed multi cycle waveform. The measurement results are displayed at the bottom of the screen.
3.  :Press the menu label and the screen will automatically display a rising edge of the signal, while measuring the rise time of the current rising edge. The measurement results are displayed at the bottom of the screen.
4.  : Press the menu label and the screen will automatically display a falling edge of the signal, while measuring the falling time of the current falling edge. The measurement results are displayed at the bottom of the screen.
5.  :Press the menu label to return to the menu displayed when the user last set it.

20.2 Default Setup

Press the **[Default Setup]** button, and the oscilloscope will display waveforms for four channels. The following table provides settings for changing options, buttons, and controls under default settings.

Table 20.2 Default Setup

Horizontal	
Horizontal Time Base	1us
Horizontal Position	0s
Delayed Sweep	Off
Timebase Mode	YT
Fine	Off
Horizontal Expansion	Center
Vertical	
Vertical Scale	1V
Vertical Offset	CH1: 3V, CH2: 1V, CH3: -1V, CH4: -3V
CH1/CH2/CH3/CH4	CH1-CH4 On

Channel Coupling	DC
BW Limit	20M
Attenuation	1X
Input Impedance	1MΩ
Invert	Off
Channel Unit	V
Fine	Off
Ch-Ch Skew	0s
Offset Cal	0V
Acquire	
Acquisition Mode	Normal
Memory Depth	Auto
Trigger	
Trigger Type	Edge Trigger
Source Selection	CH1
Edge Type	Rising
Trigger Holdoff	8ns
Noise Rejection	Off
Display	
Display Type	Vector
Persistence Time	Min
Intensity	100%
Grid	Line
Gird Brightness	30%
Show Scale	Off
Color Grade	Off
Brightness	100%
AWG	
AWG	Off

Wave	Sine
Frequency	1KHz
Amplitude	500mV
Offset	0V
Start Phase	0°
Setting Type	Off
Impedance	HighZ
Cursor	
Mode	Off
Manual	
Select	X
Source	CH1
AX	-1us
BX	1us
Track	
Source	CH1
AX	-1us
BX	1us
Storage	
Save Image	
Format	*.png
Invert	Off
Color	Color
Window	Show
Save Wave	
Data Source	Screen
Format	*.bin
Window	Show
Save Setup	
File Type	*.stp

Window	Show
Load Wave	
File Type	*.wfm
Load Setup	
File Type	*.stp
Disk	
File Type	*.*
Utility	
Sound	On
Clock Source	Import
Touch Lock	Off
Screen Saver	Off
Pass/Fail	
Enable	Off
Source	CH1
Aux Output	Off
Output Event	Fail
Polarity	Positive
Pulse	100ns
Err Action	None
Stop Output	Off
Record	
Record	Off
Interval	100ms
Frames	100
Beeper	None
Play	
Mode	single
Sequence	Sequential playback

Interval	100ms
Quick Settings	
Operation	Save Image
Format	*.png
Invert	Off
Color	Color
Math	
Operator	A+B
Operation	Off
Source A	CH1
Source B	CH1
Offset	0V
Scale	500mV
LA	
state	Off
Decode	
Bus Type	RS232
Bus Stat u s	Off
Baud	115.2kbps
Tx	CH1
Threshold	0V
Rx	Off
Polarity	Positive
Format	Hex
Position	100%
Label	Off
Data	8bit
Parity	None
Stop Bit	1bit
Package	Off

Separator	0(NULL)
Ref	
Source	CH1
VScale	1V
VOffset	0V
Color	Blue
Label	REF1

21 Arbitrary Waveform Generator (Option)

This series of oscilloscopes has a built-in function arbitrary waveform generator, which combines the oscilloscope and signal generator to make it more convenient for users to use. This chapter introduces the functions of the built-in signal generator in the oscilloscope.

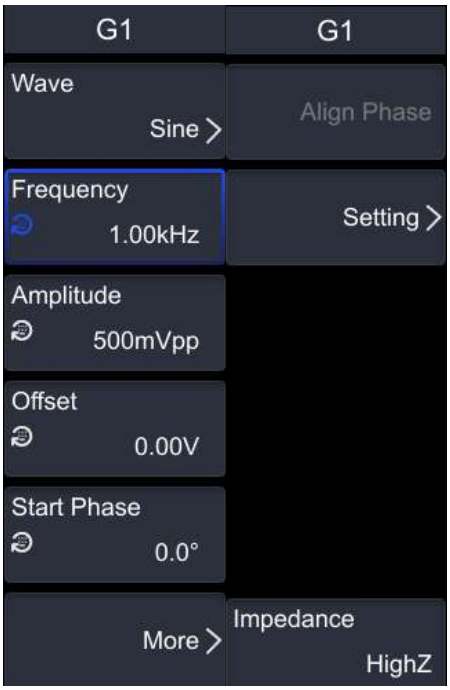
This chapter includes:

21.1 To Output Basic Waveforms

Click **[G1]** in the button area to enter the signal generator setting interface and select the output of the basic wave.

1. Output sine wave:

Click on the **Wave** menu tab and select sine wave from the pop-up menu.



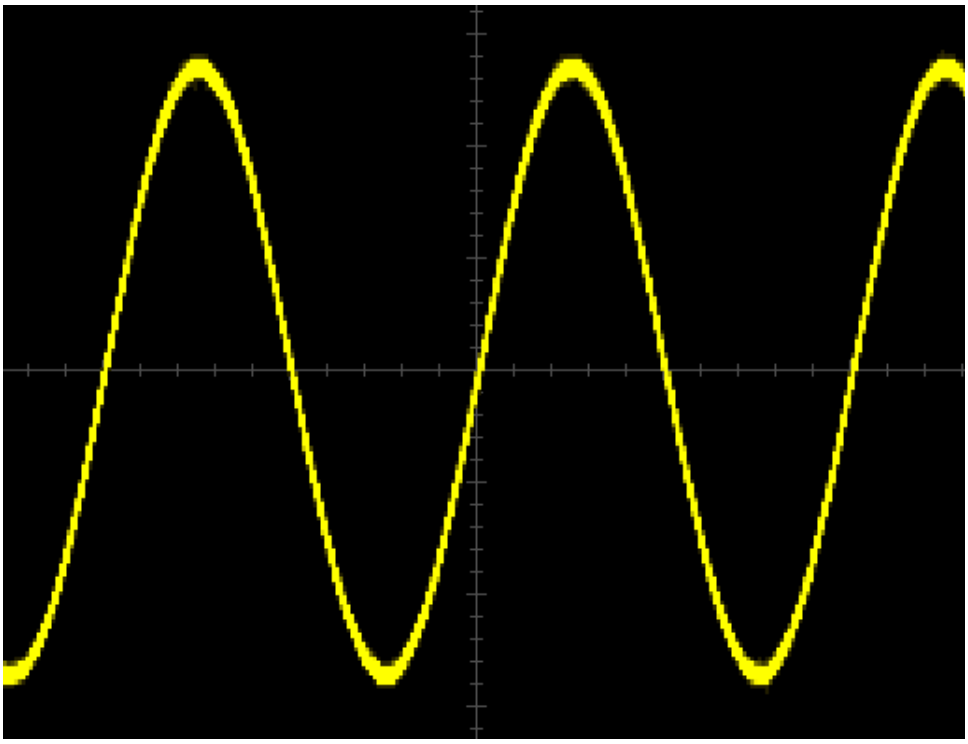


Figure 21.1 sine wave

- Frequency:

Click on the **Frequency** menu tab, and you can set the frequency value directly by rotating the knob V0 or on the pop-up numeric keypad.
DC and noise: no frequency parameters.

- Amplitude:

Click on the **Amplitude** menu label, and you can set the amplitude value directly by rotating the knob V0 or on the pop-up numeric keyboard. When the impedance is high, the setting range is 10mVpp~5Vpp; When the impedance is 50 Ω, the setting range is 5mVpp~2.5Vpp.

- Offset:

Click on the **Offset** menu label, and you can set the offset value directly by rotating the knob V0 or on the pop-up numeric keyboard.

- Start Phase:

Click on the **Start Phase** menu tab, and you can set the phase value directly by rotating the knob V0 or on the pop-up numeric keypad. Set the range to 0 °~360 °.

- Align Phase:

Click on the **Align Phase** menu tab and reconfigure the two channels to output according to the set frequency and phase. For two signals with the same frequency or a multiple frequency relationship, pressing the same phase can align the two phases.

Use an oscilloscope to collect waveforms from two channels, and after stable display, switch the channel switch state. The phase difference between the two waveforms will change. Press the same phase button, and the two waveforms will automatically recover the phase difference of the current phase.

- Setting:

Click on the **Setting** menu tab to select the types as None, Modulation, or Burst.

- Impedance:

Click on the **Impedance** menu tab to select the output impedance of the signal generator as high impedance or 50 Ω.

2. Output square wave:

Click on the **Wave** menu tab and select Square Wave from the pop-up menu.

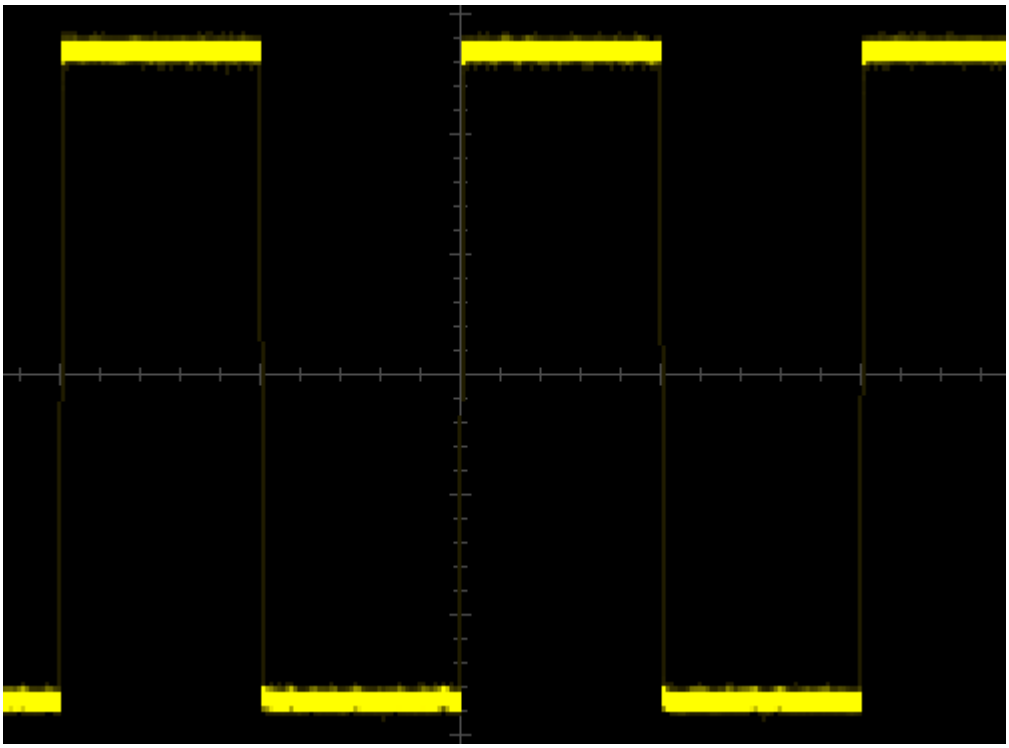


Figure 21.2 square wave

Please refer to the "Output Sine Wave" section for setting parameters. The fixed duty cycle of the square wave is 50%.

3. Output Ramp wave:

Click the **Wave** menu label, and select Ramp wave in the pop-up menu.

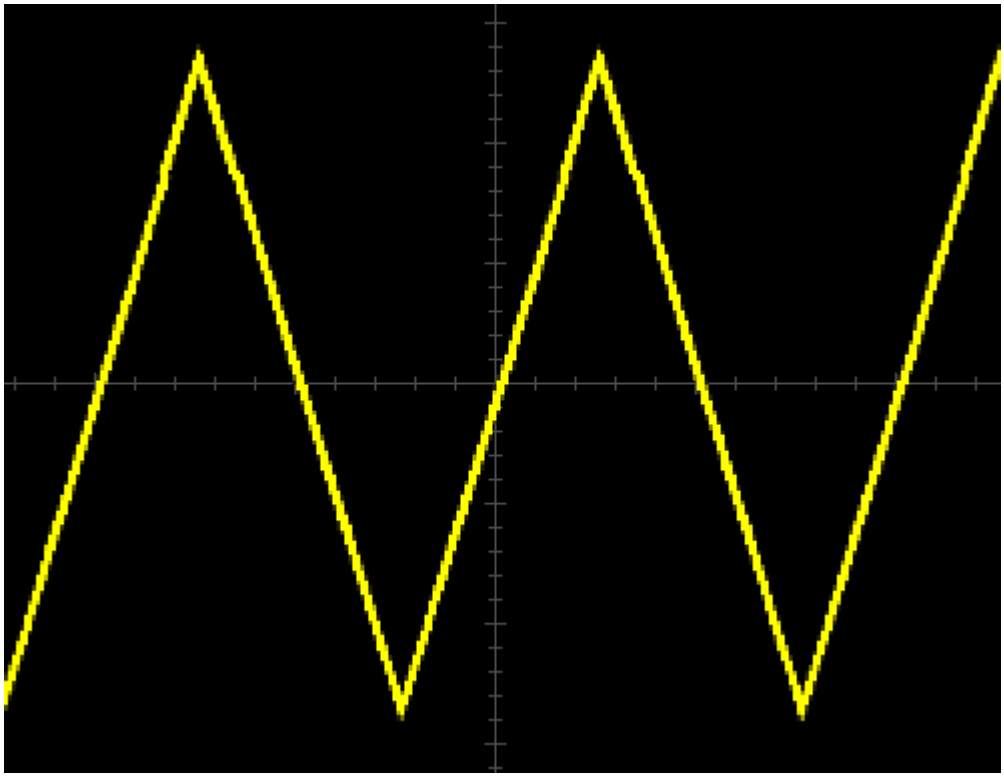


Figure 21.3 Ramp wave

Please refer to the "Output Sine Wave" section for setting parameters.
Symmetry refers to the percentage of the period during which the sawtooth wave waveform is rising. Click on the **Symmetry** menu tab, and you can set the symmetry value directly by rotating the knob V0 or on the pop-up numeric keypad.

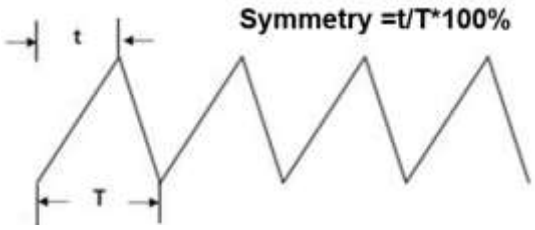


Figure 21.4 Definition of symmetry

4. Pulse:

Click on the **Wave** menu tab and select Pulse from the pop-up menu.

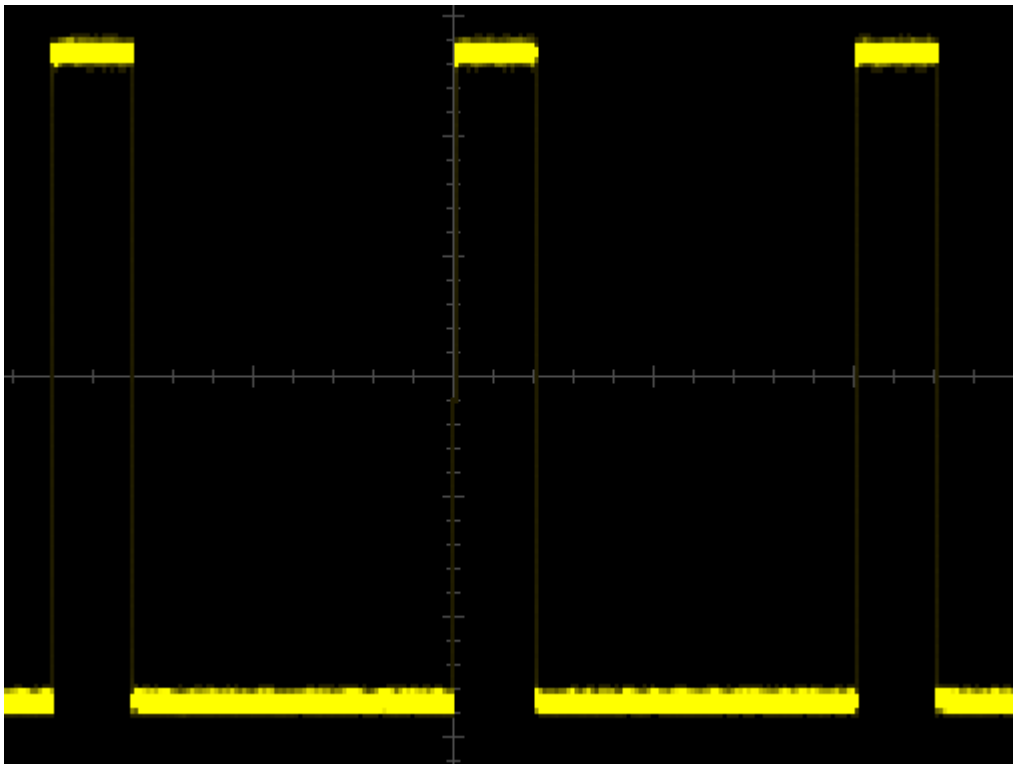


Figure 21.5 pulse

Please refer to the "Output Sine Wave" section for setting parameters.
Click on the **Pulse Width** menu tab, and you can set the pulse width directly by rotating the knob V0 or on the pop-up numeric keyboard.
Click on the **Rise Time** menu tab, and you can set the rising time by rotating the knob V0 or directly setting it on the pop-up numeric keyboard.
Click on the **Fall Time** menu tab, and you can set the descent time by rotating the knob V0 or directly setting it on the pop-up numeric keypad.

5. DC:

Click on the **Wave** menu tab and select DC from the pop-up menu.

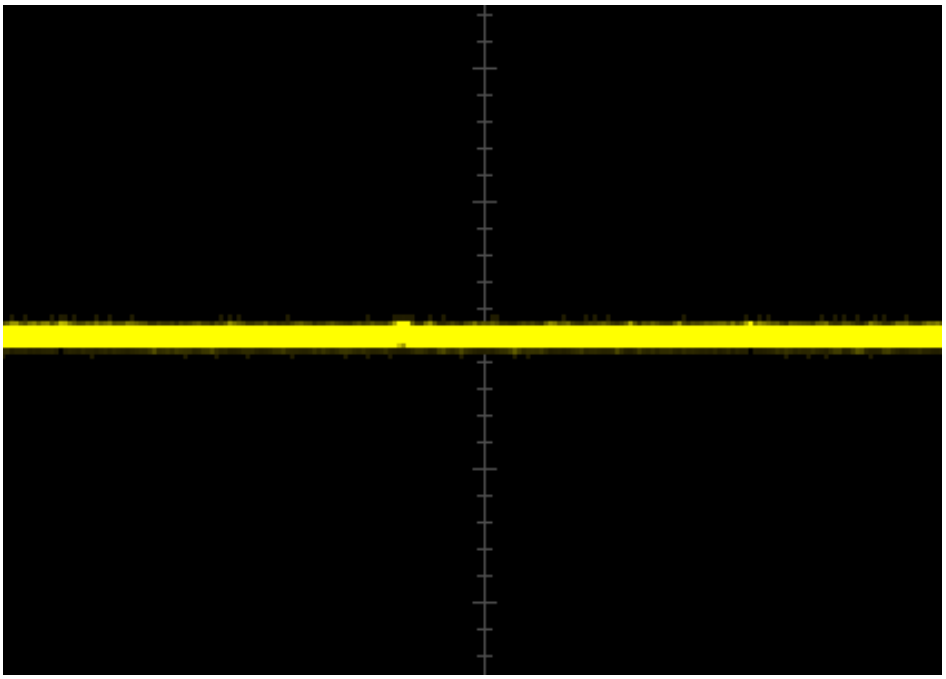


Figure 21.6 DC

- Offset:

Click on the **Offset** menu label, and you can set the offset value directly by rotating the knob V0 or on the pop-up numeric keyboard.

- Impedance:

Click on the **Impedance** menu tab to select the output impedance of the signal generator as high impedance or 50 Ω.

6. Noise:

Click on the **Wave** menu tab and select Noise from the pop-up menu.

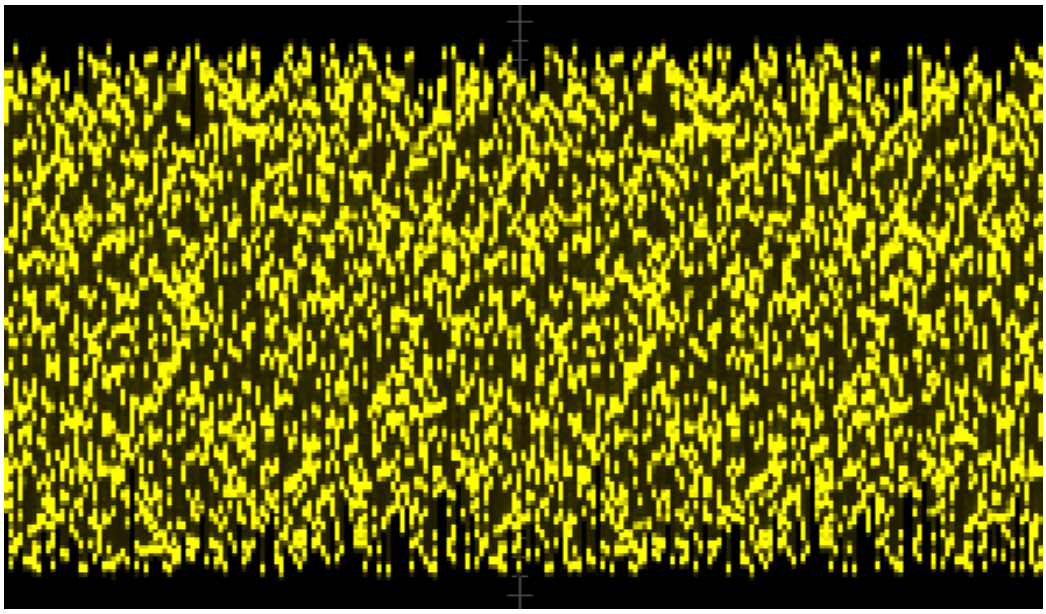


Figure 21.7 Noise

- Amplitude:

Click on the **Amplitude** menu label, and you can set the amplitude value directly by rotating the knob V0 or on the pop-up numeric keyboard.

- Set offset:

Click on the **Offset** menu label, and you can set the offset value directly by rotating the knob V0 or on the pop-up numeric keyboard.

- Impedance:

Click on the **Impedance** menu tab to select the output impedance of the signal generator as high impedance or 50 Ω.

7. Sinc:

Click on the **Wave** menu tab and select Sinc from the pop-up menu.

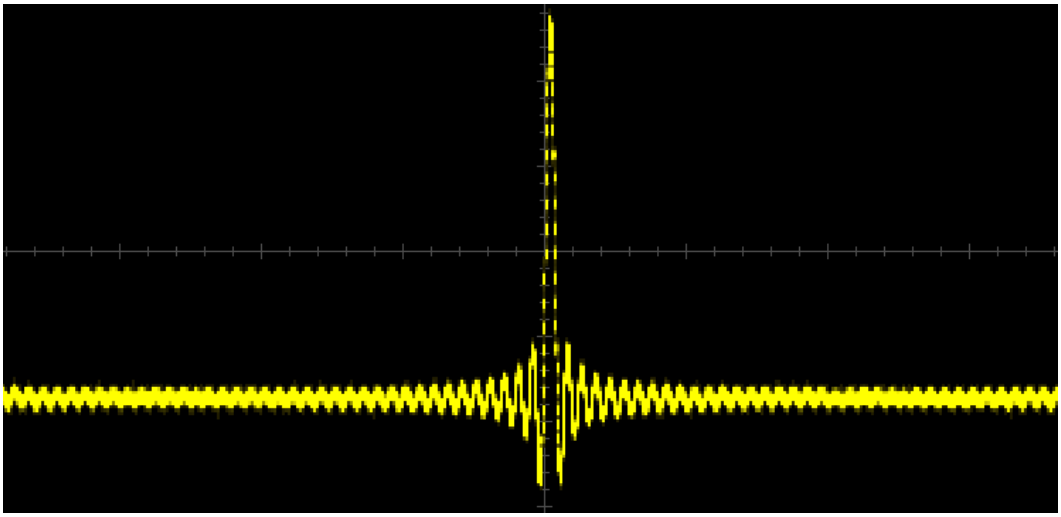


Figure 21.8 Sinc

Please refer to the "Output Sine Wave" section for setting parameters.

8. Exp.Rise:

Click on the **Wave** menu tab and select Exponential Rise from the pop-up menu.

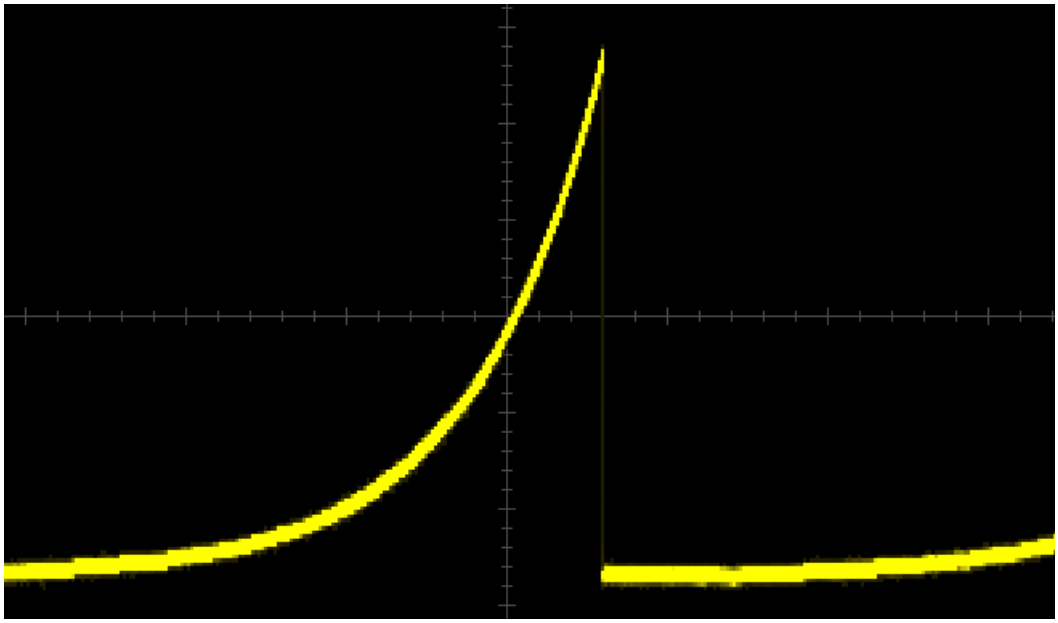


Figure 21.9 Exp.Rise

Please refer to the "Output Sine Wave" section for setting parameters.

9. Exp.Fall:

Click on the Wave menu tab and select Exponential Descent from the pop-up menu.

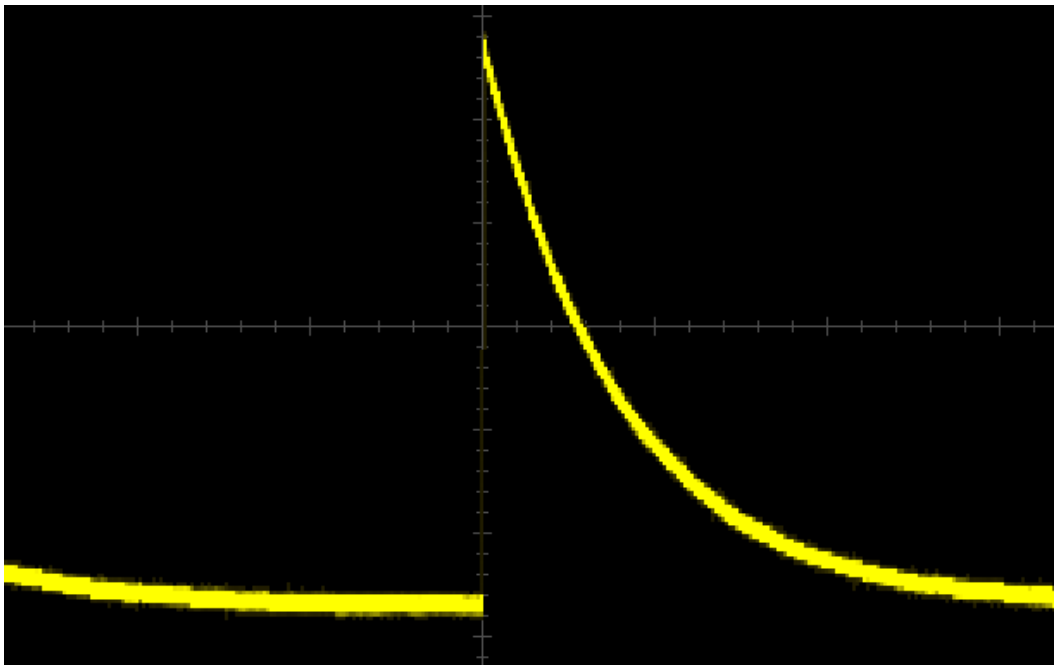


Figure 21.10 Exp.Fall

Please refer to the "Output Sine Wave" section for setting parameters.

10. ECG:

Click on the **Wave** menu tab and select electrocardiogram from the pop-up menu.

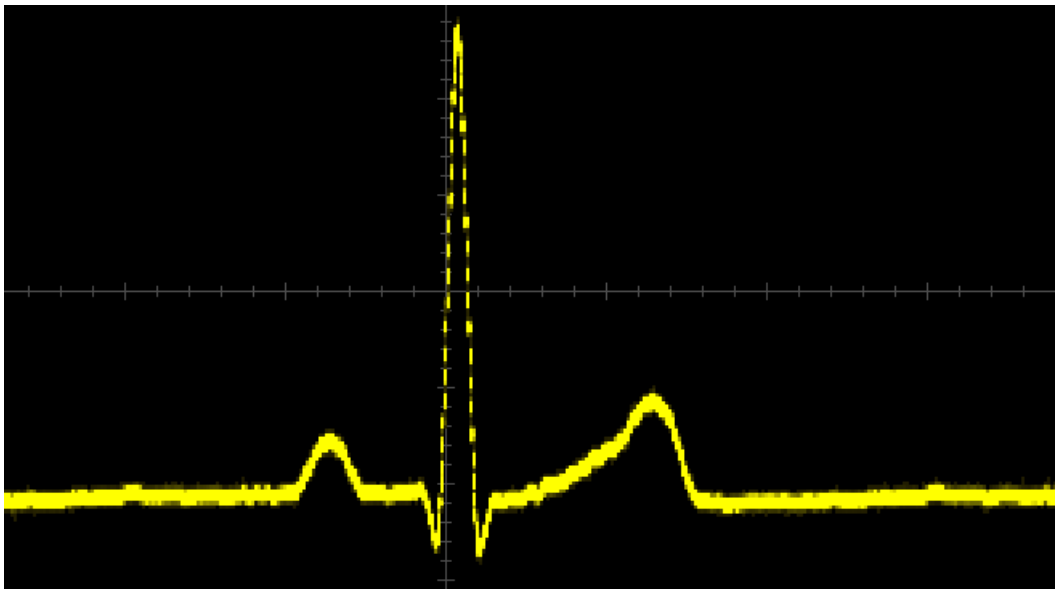


Figure 21.11 ECG

Please refer to the "Output Sine Wave" section for setting parameters.

11. Gauss:

Click on the **Wave** menu tab and select Gaussian from the pop-up menu.

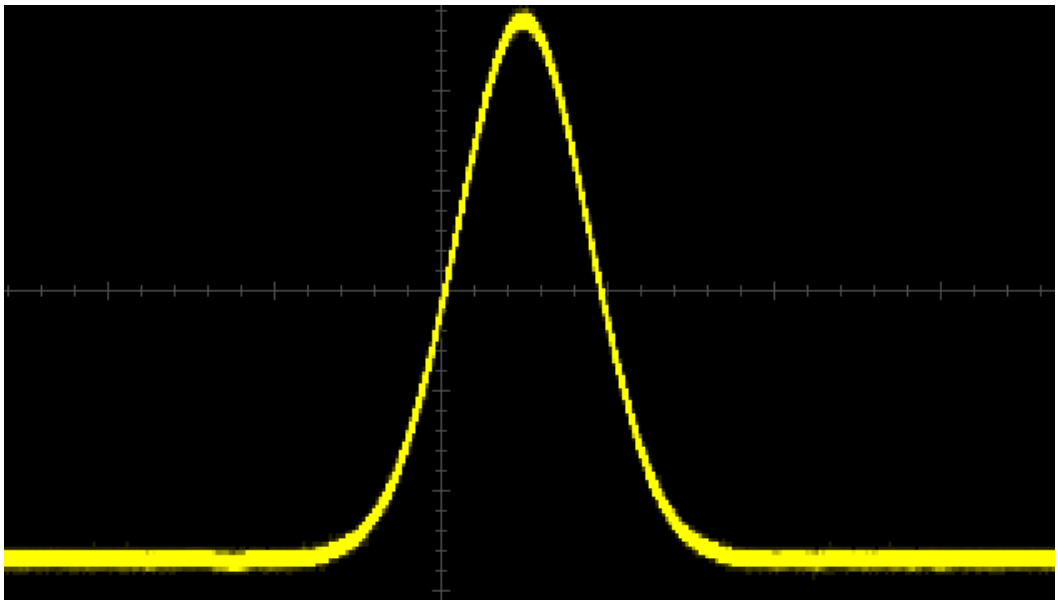


Figure 21.12 Gauss

Please refer to the "Output Sine Wave" section for setting parameters.

12. Lorentz:

Click on the **Wave** menu tab and select Lorentz from the pop-up menu.

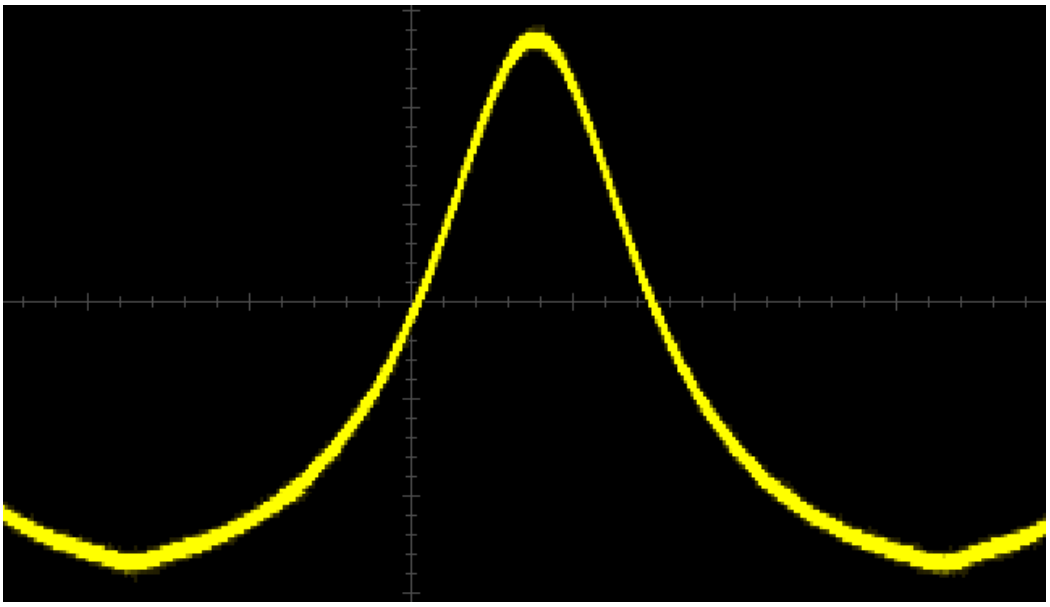


Figure 21.13 Lorentz

Please refer to the "Output Sine Wave" section for setting parameters.

13. Haversine:

Click on the **Wave** menu tab and select half vector from the pop-up menu.

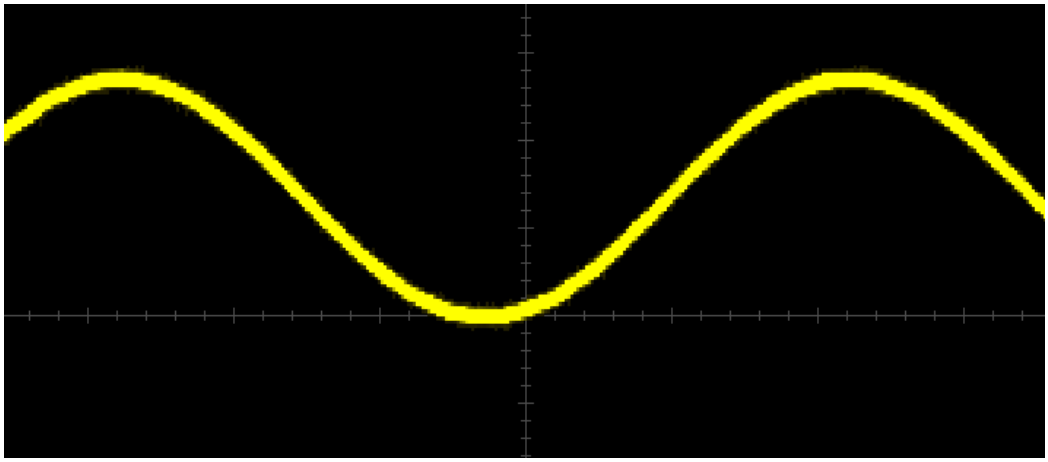


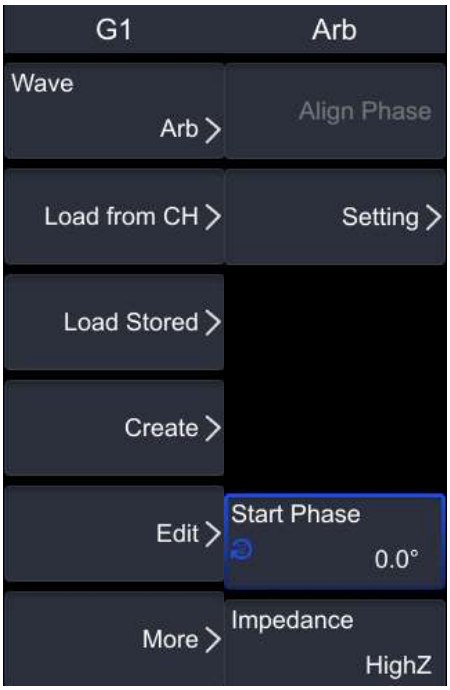
Figure 21.14 Haversine

Please refer to the "Output Sine Wave" section for setting parameters.

21.2 To Output the Arbitrary Waveform

This series of oscilloscopes supports customizing any waveform and saving it in internal or external memory.

Click **[G1]** in the button area to enter the signal generator setting interface, click the **Wave** menu label, and select any wave from the pop-up menu.



1. Load channel:

Click on the **Load from CH** menu tab to enter the Settings menu.

- Channel selection:

Click on the **Channel** menu tab to select channels CH1~CH4.

- Region:

Click on the **Region** menu tab to select the screen or cursor.
If the waveform range is selected as the cursor, click on **Cursor A** and **Cursor B**, and rotate knob V0 to adjust the positions of the two light markings respectively.
Alternatively, click on the **Cursor AB** and rotate the knob V0 to adjust the positions of both cursors simultaneously.

- Load:

Click on the **Load** menu tab to load the set channel signal.

2. Load waveform:

Click on the **Load Stored** menu tab to enter the Settings menu.
Select the waveform from the internal or external memory in the disk management interface that pops up on the screen, in the format of *. arb.

- Load:

Click on the **Load** menu tab to load the selected channel signal.

3. Creat:

Click on the **Create** menu tab to enter the Settings menu, where users can create any wave according to their needs.

- Frequency:

Click on the **Frequency** menu tab, and you can set the frequency value directly by rotating the knob V0 or on the pop-up numeric keypad.

- Amplitude:

Click on the **Amplitude** menu label, and you can set the amplitude value directly by rotating the knob V0 or on the pop-up numeric keyboard.

- Offset:

Click on the **Offset** menu label, and you can set the offset value directly by rotating the knob V0 or on the pop-up numeric keyboard.

- Init Points:

Click on the **Init Points** menu tab, and you can set the point by rotating the knob V0 or directly setting the point on the pop-up numeric keyboard.

- Linear Interp:

Click the **Linear Interp** menu tab, and you can choose to turn linear interpolation on or off.

  - Open: The waveform editor connects two defined points with a straight line.

- Close: The waveform editor will maintain a constant voltage level between two points and establish a stepped waveform.

- Edit Points:

Click on the **Edit Points** menu tab to enter the point editing settings menu, and define the waveform by assigning a voltage value to it.

- PointX:
  Click on the **PointX** menu label, and you can set the current point directly by rotating the knob V0 or on the pop-up numeric keyboard.
- Coltage:
  Click on the **Coltage** menu tab, and you can set the voltage value directly by rotating the knob V0 or on the pop-up numeric keypad.
- Insert:
  Click on the **Insert** menu tab to insert a new editable point at the midpoint of the current and next editing points, and the initialization points will automatically increase by 1. If you continue to press **Insert**, you can add editable points one by one.
- Delete:
  Click the **Delete** menu tab to delete the current point from the waveform, and the remaining points are connected using interpolation.
- Zoom:
  Click on the **Zoom** in menu tab to select whether to turn on or off the horizontal time base zoom in function.
- Apply:
  Click on the **Apply** menu tab to output any currently edited waveform.

- Save any wave:

Click on the **Save** menu tab to enter the Save Settings menu.
Refer to the instructions in the "Store and Load" section.

4. Edit:

Click the **Edit** menu tab to enter the Settings menu, where users can edit the current waveform.

- Frequency:

Click on the **Frequency** menu tab, and you can set the frequency value directly by rotating the knob V0 or on the pop-up numeric keypad.

- Amplitude:

Click on the **Amplitude** menu label, and you can set the amplitude value directly by rotating the knob V0 or on the pop-up numeric keyboard.

- Offset:

Click on the **Offset** menu label, and you can set the offset value directly by rotating the knob V0 or on the pop-up numeric keyboard.

- Linear Interp:

Click the **Linear Interp** menu tab, and you can choose to turn linear interpolation on or off.

- Open: The waveform editor connects two defined points with a straight line.
- Close: The waveform editor will maintain a constant voltage level between two points and establish a stepped waveform.

- Edit Points:

Click on the **Edit Points** menu tab to enter the point editing settings menu, and define the waveform by assigning a voltage value to it.

- PonitX:
  Click on the **PointX** menu label, and you can set the current point directly by rotating the knob V0 or on the pop-up numeric keyboard.
- Coltage:
  Click on the **Coltage** menu tab, and you can set the voltage value directly by rotating the knob V0 or on the pop-up numeric keypad.
- Insert:
  Click on the **Insert** menu tab to insert a new editable point at the midpoint of the current and next editing points, and the initialization points will automatically increase by 1. If you continue to press **Insert**, you can add editable points one by one.
- Delete:
  Click the **Delete** menu tab to delete the current point from the waveform, and the remaining points are connected using interpolation.
- Zoom:
  Click on the **Zoom** in menu tab to select whether to turn on or off the horizontal time base zoom in function.
- Apply:
  Click on the **Apply** menu tab to output any currently edited waveform.

- Save any wave:

Click on the **Save** menu tab to enter the Save Settings menu.
Refer to the instructions in the "Store and Load" section.

5. Setting:

Click on the **Setting** menu tab and select None, Modulation, or Burst.

6. Start Phase:

Click on the **Start Phase** menu tab, and you can set the phase value directly by rotating the knob V0 or on the pop-up numeric keypad.

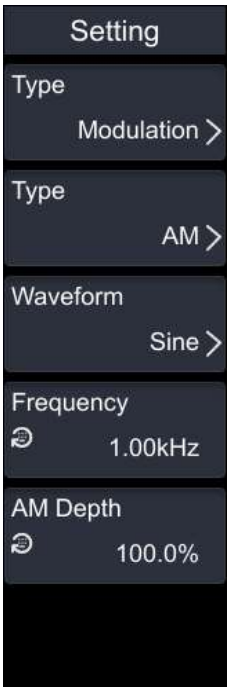
7. Impedance:

Click on the **Impedance** menu tab and select high resistance or 50 ohms.

21.3 Modulation

The signal generator of this series of oscilloscopes has three modulation methods: amplitude modulation, frequency modulation, and phase modulation. The modulated waveform consists of a carrier wave and a modulated wave. The carrier signal is a waveform signal output by the signal generator, and the modulation signal can be selected from the sine wave, square wave, triangular wave, and noise built-in in the signal generator.

Click on the button area **[G1]**, go to **More** -> **Setting** -> **Type** -> **Modulation** to enter the modulation settings menu.



Click on the **Type** menu tab to select amplitude modulation (AM), frequency modulation (FM), and phase modulation (PM).

1. AM:

Amplitude Modulation (AM) refers to the variation of the amplitude of the carrier wave with the modulation wave.

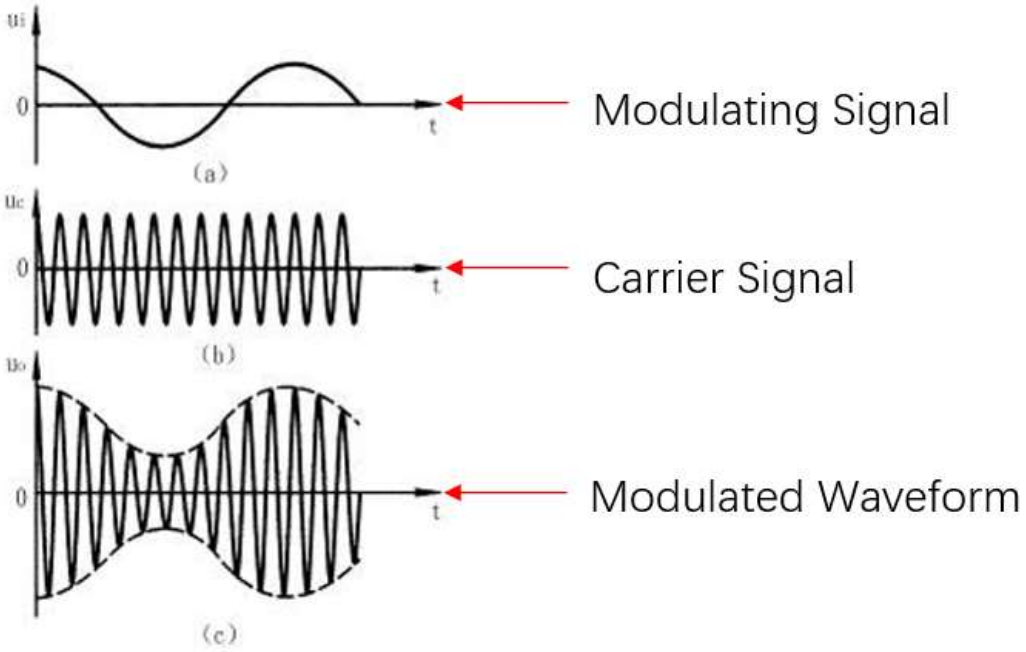


Figure 21.15 AM

- Modulation waveform:

Click on the **Waveform** menu tab to select waveforms such as sine wave, square wave, triangular wave, and noise.

- Frequency:

Click on the **Frequency** menu tab, and you can set the frequency value directly by rotating the knob V0 or on the pop-up numeric keypad.

- Modulation depth:

Click on the **AM Depth** menu tab, and you can set the modulation depth by rotating the knob V0 or directly setting it on the pop-up numeric keyboard. The modulation depth represents the strength of amplitude modulation. At 0% modulation, the output amplitude is half of the carrier signal amplitude; At 100% modulation, the output amplitude is equal to the carrier signal amplitude; When the modulation is greater than 100%, envelope distortion will occur, which needs to be avoided in actual circuits.

2. FM:

Frequency Modulation (FM) refers to the frequency of a carrier wave that changes with the modulation wave.

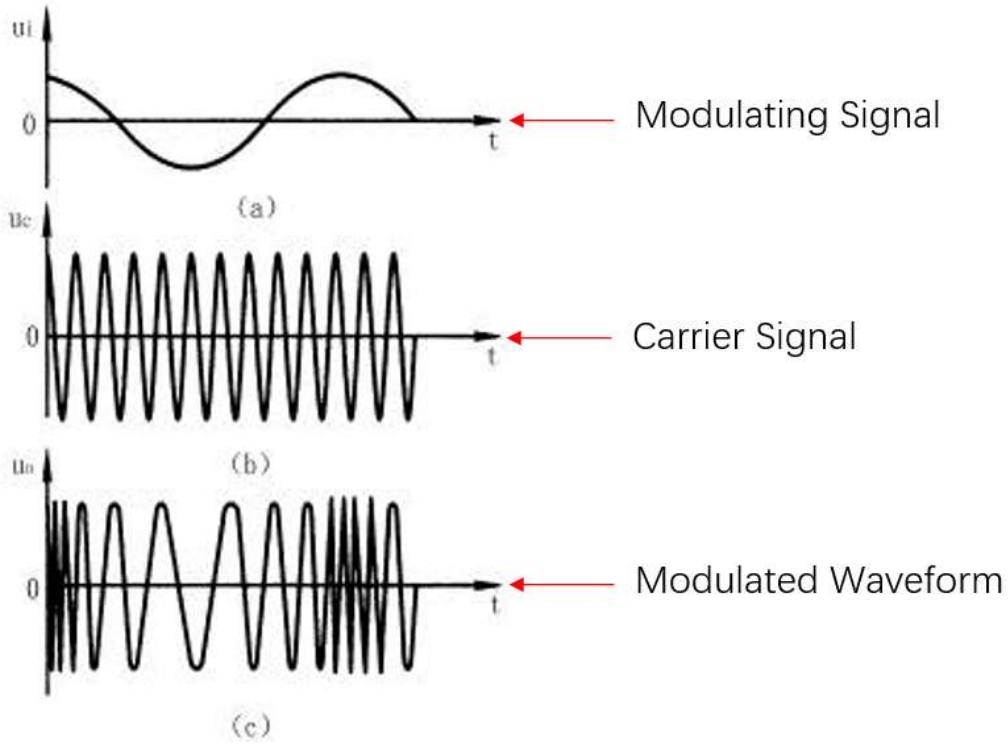


Figure 21.16 FM

- Modulation waveform:

Click on the **Waveform** menu tab to select waveforms such as sine wave, square wave, triangular wave, and noise.

- Frequency:

Click on the **Frequency** menu tab, and you can set the frequency value directly by rotating the knob V0 or on the pop-up numeric keypad.

- Deviation:

Click on the **Deviation** menu label, and you can set the deviation value by rotating the knob V0 or directly setting it on the pop-up numeric keyboard.

3. PM:

The modulation method in which the deviation of the carrier phase from its reference phase varies proportionally with the instantaneous value of the modulated signal is called phase modulation, or phase modulation. There is a close relationship between phase modulation and frequency modulation. During phase modulation, frequency modulation occurs simultaneously; During frequency modulation, phase modulation also occurs simultaneously, but the changing patterns of the two are different.

- Modulation waveform:

Click on the **Waveform** menu tab to select waveforms such as sine wave, square wave,

triangular wave, and noise.

- Frequency:

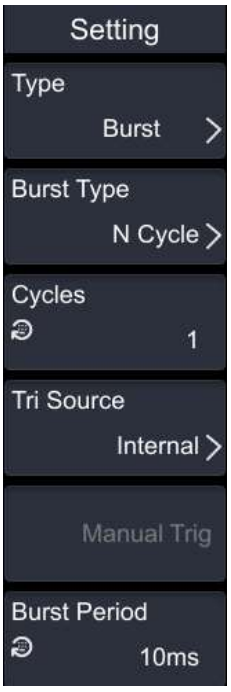
Click on the **Frequency** menu tab, and you can set the frequency value directly by rotating the knob V0 or on the pop-up numeric keypad.

- Modulation depth:

Click on the **PM Depth** menu tab, and you can set the modulation depth by rotating the knob V0 or directly setting it on the pop-up numeric keyboard.

21.4 Burst

The signal generator of this series of oscilloscopes supports outputting burst waveforms with a specified number of cycles. This series of oscilloscopes supports internal or manual control of burst waveform output; Supports two burst types: N-loop and infinite.
The burst setting menu can be accessed through the following methods:
Click on the button area **[Burst]** to enter the burst setting menu.
Click on the button area **[G1]**, go to **More** -> **Setting** -> **Type** -> **Burst**, and enter the Burst Settings menu.



1. Burst Type:

Click on the **Burst Type** menu tab to select N cycle or infinite.

- N cycle: When receiving a trigger signal, outputs a burst waveform with a specific number of cycles.
- Infinite: It is equivalent to setting the number of waveform cycles to infinity, and outputting a continuous waveform upon receiving the trigger signal.

2. Cycles:

Click on the **Cycles** number menu tab, and you can set the cycle number by rotating the knob V0 or directly setting it on the pop-up numeric keypad. The number of cycles can only be set when multiple cycles are selected for the burst type.

3. Trigger source:

Click on the **Tri Source** menu tab to select either internal or manual. The signal generator generates a burst output when receiving a trigger signal, and then waits for the next trigger signal.

- Internal: When triggered internally, the signal generator outputs continuous multiple cycles.
- Manual: When manually triggered, each press of the manual trigger button immediately triggers a burst output in the corresponding channel. Under the infinite burst type, the number of cycles menu is disabled.

4. Burst Period:

Click on the **Burst Period** menu tab to set the cycle by rotating the knob V0 or directly setting the cycle on the pop-up numeric keypad. The burst period is the time from the beginning of one burst to the beginning of the next burst.

22 Remote Control

The oscilloscope can be remotely controlled in the following three methods:

1. User-defined programming:

Users can program and control the oscilloscope through the standard SCPI (Standard Commands for Programmable Instruments) command. For detailed instructions on commands and programming, please refer to the DPO8000 Programming Manual.

2. Using IO software:

Users can use IO software to send commands for remote control of the oscilloscope. It is recommended to use the PC software IO provided by Keysight. You can log in to the Keysight official website (www.keysight.com) to download the software.

Operating steps:

- Establish communication between the oscilloscope and the computer.
- Run IO and search for oscilloscope resources.
- Open the remote command control panel and send commands.

This oscilloscope can communicate with a PC through the following interfaces:

- Controlled through USB
- Control through LAN
- Controlled through RS232/485

This chapter will provide a detailed introduction to how to use the IO software provided by Keysight to remotely control the oscilloscope through various interfaces.

Note:

Before connecting the communication cable, please turn off the instrument to avoid damaging its communication interface.

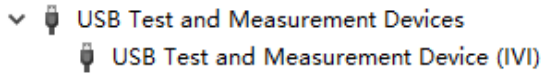
22.1 Remote Control via USB

1. Connect the device

Connect the oscilloscope (USB VIEW interface) to the PC (USB HOST interface) using a USB data cable.

2. Install USB driver

After the oscilloscope is correctly connected to the PC for the first time and turned on, the PC will automatically install the driver, as shown in the Device Manager:



3. Search for device resources

Open IO and the software will automatically search for resources currently connected to the PC through a USB interface. You can also click Rescan to search.

4. View Device Resources

The searched resources will appear in the USB (USB0) directory, displaying the machine model, serial number, version information, and USB interface information.

5. Control the instrument remotely

Click on Interactive IO in the IO interface to open the remote command control panel, where you can send commands and read data.

22.2 Remote Control via LAN

1. Connect the device

Connect the oscilloscope to your local area network using a network cable.

2. Configure network parameters

Configure the network parameters of the oscilloscope according to the instructions in the section on LAN Configuration.

3. Search for device resources

Open IO and click Rescan to search for devices.

4. View Device Resources

The name of the oscilloscope found is displayed on LAN (TCPIP0).

5. Control the instrument remotely

Click on Interactive IO in the IO interface to open the remote command control panel, where you can send commands and read data.

6. Load webpage

The DPO8000 oscilloscope has a LAN interface and a built-in oscilloscope web server, which allows you to remotely:

- You can view information about the oscilloscope, such as its model, serial number, IP address, firmware version, etc.
- Use the remote front panel to control the oscilloscope.
- Send SCPI commands.
- View and modify the network configuration and status of the oscilloscope.

Before you can use the web interface, you must turn on the oscilloscope and establish its LAN connection.

1) Set up LAN connection for oscilloscope

Before you can use the web interface, you must turn on the oscilloscope and establish its LAN connection. Whenever you modify the IP address of the oscilloscope, it will disconnect the oscilloscope from the local area network. You need to use a new IP address to re-establish communication with the oscilloscope.

Connect to the network manually:

- Open [**UTILITY**]-> **I/O** -> **LAN** -> **Static IP** of the machine to obtain its IP address.
- Insert the oscilloscope into the local area network and connect the LAN cable to the LAN port on the back panel of the oscilloscope.
- Modify the network connection of the computer, set the IP address to 192.168.1.XX (XX is different from the last two digits of the machine), and the subnet mask to 255.255.255.0.

2) Accessing the web interface

To access the web interface of the oscilloscope:

- Connect the oscilloscope to the local area network, refer to the previous section.
- Enter the IP address of the oscilloscope in the browser, and the interface will open the oscilloscope's web interface.

3) Web page control

Using the web interface to operate the oscilloscope:

- Home: Provide specific information about the machine.
- Control instruments: HTML5 controls and instrument IO.
- Configure LAN: Edit network configuration.
- Language : Supports both Chinese and English.
- Help: Click to view quick help.

a) Home:

View information about the oscilloscope, such as model, serial number, host name, VISA tool address, and LAN information.

b) Launch HTML5 Control:

In the web interface of the oscilloscope, select to launch the HTML5 control, and the remote front panel will appear.

In the remote front panel, you can click on the oscilloscope buttons for operation, but you cannot click on the oscilloscope screen for operation.

c) Using instrument IO:

In the web interface of the oscilloscope, select the instrument IO, open the IO interface, and send remote programming commands to the oscilloscope through SCPI.

d) Configure LAN:

In the web interface of the oscilloscope, select Configure LAN to display the current LAN configuration of the instrument or edit a new LAN configuration.

Click 'Edit' to enter the network configuration editing process, where you can reset

the new IP address. The instrument will use the new IP address to connect to the browser.

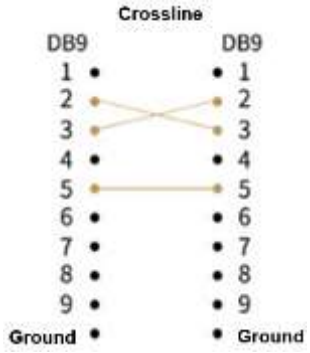
22.3 Remote Control via RS232

1. Connect the device

Connect the oscilloscope to your local area network using an RS232 serial cable.

Note:

Please use RS232 cross serial cable. Refer to the following figure.



2. Configure network parameters

Configure the parameters of the oscilloscope according to the instructions in the section on controlling through RS232 Configuration.

3. Search for device resources

Open IO and click Rescan to search for devices.

4. View Device Resources

The name of the oscilloscope found is displayed in COM.

5. Control the instrument remotely

Click on Interactive IO in the IO interface to open the remote command control panel, where you can send commands and read data.

23 Troubleshooting

The following is a list of possible faults and troubleshooting methods that may occur during the use of an oscilloscope. When you encounter these faults, please follow the corresponding steps to handle them. If you cannot handle them, please contact Hantek and provide the equipment information of your machine.

23.1 The display of waveform is ladder-like

1. The horizontal time base might be too low. Increase the horizontal time base to improve the display effects.
2. If the display type is "Vector", the lines between the sample points may cause ladder-like display results.

23.2 After connecting the power cord, the button light does not light up

1. Check if the power switch of the oscilloscope is turned on.
2. Check if the fuse is intact. If a fuse needs to be replaced, please use a fuse that meets the specifications of this product.
3. If the button board still does not light up, please contact Hantek technical support department in a timely manner.

23.3 No waveform of the signal is displayed on the screen

1. Check if the probe is correctly connected to the BNC connector of the signal input channel.
2. Check if the channel is open (CH1~CH4 menu buttons).
3. Detect whether the signal channel to be tested has signal output.
4. If it is a DC signal and the amplitude is relatively large, please increase the amplitude range.
5. You can press the automatic measurement button to automatically detect the signal first.

If there is still no waveform display, please contact Hantai technical support department in a timely manner.

23.4 The touch functions cannot be used normally

1. Check if the touch screen function is turned on. If it is not, click **[Touch Lock]** in the button area to turn on the touch screen function.
2. Check if there is any dirt on the screen or fingers. If there is, please clean it first.
3. Check if the oscilloscope is close to a strong magnetic field. If it is close to a strong magnetic field, please stay away first.

If you still cannot use the touch screen, please contact Hantek technical support department in a timely manner.

23.5 The USB storage device cannot be recognized

1. Check if the USB drive is functioning properly.
2. Confirm the capacity of the USB flash drive. It is recommended to use a USB flash drive that does not exceed (16G) for this oscilloscope.
3. Confirm that the format of the USB drive is FAT32 format.
4. Reopen the oscilloscope and reinsert the USB drive for inspection.

If you still cannot use the USB drive, please contact Hantek Technical Support Department in a timely manner.

23.6 The instrument cannot start normally

1. Check if the power switch of the machine is turned on.
2. Check if the power plug of the oscilloscope is properly connected.
3. Check if the fuse is intact. If a fuse needs to be replaced, please use a fuse that meets the specifications of this product.
4. Turn on the machine again and see if it can be turned on normally.

If the machine still cannot be opened, please contact Hantek Technical Support Department in a timely manner.

24 Appendix

24.1 Appendix A: Accessories

Order information	Order number
Host model	
8GSa/S, 2Gpts, 1GHz 4-channel oscilloscope + LA + AWG	DPO8104E
8GSa/S, 2Gpts, 800MHz 4-channel oscilloscope + LA + AWG	DPO8084E
8GSa/S, 2Gpts, 500MHz 4-channel oscilloscope+ LA + AWG	DPO8054E
8GSa/S, 2Gpts, 350MHz 4-channel oscilloscope+ LA + AWG	DPO8034E
8GSa/S, 2Gpts, 200MHz 4-channel oscilloscope+ LA + AWG	DPO8024E
8GSa/S, 2Gpts, 1GHz 4-channel oscilloscope	DPO8104C
8GSa/S, 2Gpts, 800MHz 4-channel oscilloscope	DPO8084C
8GSa/S, 2Gpts, 500MHz 4-channel oscilloscope	DPO8054C
8GSa/S, 2Gpts, 350MHz 4-channel oscilloscope	DPO8034C
8GSa/S, 2Gpts, 200MHz 4-channel oscilloscope	DPO8024C
8GSa/S, 2Gpts, 500MHz 4-channel oscilloscope+ LA + AWG	DPO8054U
Standard accessories	
USB cable	--
Power Cord Conforming to the Standard of the Destination Country	--
Oscilloscope probe	PP-100(100MHz) PP-200(200MHz) HT300B(350MHz)

Order information	Order number
	HT500B(500MHz)

24.2 Appendix B: Warranty summary

Qingdao Hantek Electronic Co., LTD. (hereinafter referred to as Hantek) undertakes that the host and accessories of its production shall be free from any material and process defects during the warranty period.

During the warranty period, if the product is proved to be defective, Hantek will repair or replace the product free of charge. Please refer to the description on Hantek official website for detailed warranty regulations. For repair service or full warranty instructions, please contact Hantek repair center or local offices.

Hantek disclaims warranties, express or implied, other than those provided in this summary or any other applicable warranty card, including, but not limited to, any implied warranties of merchantability and fitness for special purpose. In no event shall Hantek be liable for indirect, special or consequential damages.

Addr: #35 Building, No. 780 Baoyuan Road, High-tech Zone, Qingdao, Shandong, China 266114

Switchboard: 400-036-7077

Email: service@hantek.com

Tel: (0086)532-55678770 & 55678772 & 55678773

Zip code: 266114

Website：www.hantek.com

Qingdao Hantek Electronic Co., LTD