Hantek

User Manual

202506

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 DSO2D20 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

Contents I

Figures IX

Tables XI

1 Safety requirement 1

1.1 Summary of general security issues 1

1.2 Security terms and signs 2

1.3 Measurement category 2

1.4 Ventilation Requirement 3

1.5 Working Environment 3

1.6 Care and Cleaning 5

1.7 Environmental Considerations 5

2 Product features 7

3 Document overview 9

4 Quick start 11

4.1 General examination 11

4.2 Appearance and dimension 11

4.3 Preparation before use 12

4.4 Front Panel Overview 16

4.5 Front Panel Function Overview 17
4.6 Rear Panel Overview 22
4.7 Rear Panel Overview 22
4.8 User Interface 23
4.9 Help System....24
5 To Set the Vertical System....25
5.1 To Enable or Disable the Analog Channel....26
5.2 To Adjust the Vertical Scale 26
5.3 To Adjust the Vertical Offset....27
5.4 Channel Coupling....27
5.5 Bandwidth Limit....27
5.6 Input Impedance 28
5.7 Probe Ratio 28
5.8 Waveform Invert....29
5.9 Fine tuning 30
6 To Set the Horizontal System 31
6.1 To Adjust the Horizontal Position 32
6.2 To Adjust the Horizontal Time Base 32
6.3 Pan or zoom single acquisition or stopped acquisition 32

6.4 Delayed Sweep 32
6.5 Horizontal Mode 33
6.5.1 YT Mode 33
6.5.2 XY Mode 34
6.5.3 Roll Mode 35
7 To Set the Sample System 37
7.1 Acquire Type 38
7.2 Acquire Mode 38
7.2.1 Normal Mode 38
7.2.2 Average Mode 38
7.2.3 Peak Mode 39
7.2.4 High Resolution 40
7.3 Memory Depth 40
8 To Trigger the Oscilloscope 42
8.1 Trigger LEVEL 43
8.2 Trigger Source 43
8.3 Trigger Mode 43
8.4 Trigger Type 45
8.4.1 Edge Trigger 45

8.4.2 Pulse Trigger 47
8.4.3 Video Trigger 50
8.4.4 Slope Trigger 52
8.4.5 Overtime Trigger 53
8.4.6 Window Trigger 55
8.4.7 Pattern Trigger 57
8.4.8 Interval Trigger 59
8.4.9 Under Amp Trigger 60
8.4.10 UART Trigger 62
8.4.11 LIN Trigger 65
8.4.12 CAN Trigger 67
8.4.13 SPI Trigger 69
8.4.14 I2C Trigger 71
9 Math Operation 75
9.1 Arithmetic Operations 76
9.2 FFT 77
10 Measure 79
10.1 Source 80
10.2 Type 80

10.3 DVM 82
10.4 Statistics 82
10.5 Clear All 82
10.6 All Measure 82
10.7 Settings 83
10.8 Gate 83
10.9 Quick Measurement after AUTO 83
11 Cursor Measurement 85
11.1 Manual Mode 86
11.2 Track Mode 87
12 Display 90
12.1 Type 91
12.2 Intensity 91
12.3 Grid 91
12.4 To Set the Grid Brightness 91
12.5 To Set the Screen Brightness 91
12.6 Persist 91
13 System Utility Function Setting 93
13.1 Language 94

13.2 Sound 94
13.3 Update 94
13.4 Pass/Fail 94
13.4.1 To Enable or Disable the Pass/Fail Test Function 95
13.4.2 To Select the Source 95
13.4.3 Regular 95
13.4.4 Message 96
13.4.5 Mode 96
13.4.6 Run 96
13.4.7 Stop 96
13.4.8 Out Stop 96
13.5 System Info 97
13.6 Calibrate 97
13.7 FrontPanel Selftest 97
13.8 Legal Information 97
14 Decode 98
14.1 UART Decode 99
14.2 LIN Decode 101
14.3 CAN Decode 102

14.4 SPI Decode 103

14.5 IIC Decode 104

15 Store and Load 106

15.1 Internal SAVE and RECALL 107

15.1.1 Internal SAVE 107

15.1.2 Internal RECALL 108

15.2 External SAVE and RECALL 108

15.2.1 External SAVE 108

15.2.2 External RECALL 109

15.3 File Management 110

15.3.1 New File 110

15.3.2 Delete File 111

15.3.3 Rename File 111

16 Quik key 112

16.1 Auto Scale 113

16.2 Default Setup 114

17 Arbitrary Waveform Generator 116

17.1 Output waveform type 117

17.2 Modulation 117

17.3 Burst 119

18 Remote Control 121

18.1 Remote Control via USB 121

19 Troubleshooting 123

19.1 After pressing the power switch, the oscilloscope did not turn on 123

19.2 Serious distortion of input signal 123

19.3 No signal appears on the screen 123

19.4 USB flash drive cannot be recognized 124

19.5 Waveform cannot be triggered 124

20 Appendix 125

20.1 Appendix A: Accessories 125

20.2 Appendix B: Warranty summary 126

Figures

Figure 4.1 Front view 11
Figure 4.2 Side view 12
Figure 4.3 To Connect to AC Power 13
Figure 4.4 To Use the Compensation Signal 13
Figure 4.5 Square Waveform Signal 14
Figure 4.6 Probe Compensation 15
Figure 4.7 Adjusting capacitance 15
Figure 4.8 Front Panel 16
Figure 4.9 Rear Panel 22
Figure 4.10 User Interface 23
Figure 5.1 "Invert" Off 29
Figure 5.2 "Invert" On 30
Figure 6.1 Delayed Sweep Mode 33
Figure 6.2 Delayed Sweep Mode 34
Figure 6.3 Delayed Sweep Mode 35
Figure 7.1 Delayed Sweep Mode 39
Figure 7.2 Delayed Sweep Mode 39
Figure 7.3 Memory Depth 40
Figure 8.1 Schematic Diagram of the Acquisition Memory 44
Figure 8.2 Rising/Falling 45
Figure 8.3 Positive Pulse Width/Negative Pulse Width 47

Figure 8.4 Positive Slope Time/Negative Slope Time 52

Figure 8.5 Overtime Trigger..... 54

Figure 8.6 Window Trigger..... 55

Figure 8.7 Window Trigger..... 57

Figure 8.8 Runt Trigger..... 61

Figure 8.9 Schematic Diagram of UART Protocol..... 63

Figure 8.10 Data Frame Format of the LIN Bus..... 65

Figure 8.11 Data Frame Format of the CAN Bus..... 67

Figure 8.12 Sequential Chart of SPI Bus..... 70

Figure 8.13 Schematic Diagram of I2C Protocol..... 72

Figure 10.1 All Measure..... 83

Figure 11.1 Manual Cursor Measurement Example 87

Figure 11.2 Track Measurement (before Horizontal Expansion)..... 89

Figure 11.3 Track Measurement (after Horizontal Expansion)..... 89

Figure 14.1 Schematic Diagram of UART Serial Bus 99

Figure 14.2 Schematic Diagram of Negative Logic..... 99

Figure 14.3 SPI Bus Schematic..... 103

Figure 14.4 Schematic diagram of I2C serial bus 105

Figure 17.1 AM 118

Figure 17.2 FM 119

Tables

Table 3.1 Model 10

Table 4.1 Front Panel Description..... 16

Table 4.2 Rear Panel Description 22

Table 5.1 Probe Ratio 28

Table 8.1 Video Standard 51

Table 9.1 Window Function..... 78

Table 16.1 Auto Scale 113

Table 16.2 Default Setup 114

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

- Support 1MΩ/50Ω impedance switching to ensure signal integrity;
- The sampling rate of the entire system is 2GSa/s, with a maximum bandwidth of 500MHz;
- Built in arbitrary waveform generator, capable of outputting 5 standard waveforms, supports custom arbitrary waveform output, and supports burst output;
- Protocol Trigger: Comes with five standard serial protocol triggers and decoding, supporting protocols including UART, LIN, CAN, SPI, IIC, making it easy to analyze serial bus data;
- 32 automatic measurement and threshold measurement functions, with test results supporting statistical analysis;
- Two sets of digital voltmeter functions and hardware frequency meter functions;Standard SCPI remote control instructions, convenient for users to build testing systems;
- Save and export function, which can save settings CSV, Information such as images, reference waveforms, and waveforms;
- Four sampling methods: normal, average, peak, and high-precision;
- 14 operating languages, supporting over 90% of countries and regions worldwide;
- There are two cursor measurement modes: manual and tracking, and the Math function also supports cursor measurement;
- 500uV-10V vertical measurement gear, 300VCAT Ⅱ withstand voltage input;
- FFT scale display, convenient for reading results;
- Manual, single time, one click AUTO measurement, simple and fast waveform measurement;
- XY mode supports dual window display, making it easy to understand waveform phase changes;

The DSO2D20 series digital oscilloscope has comprehensive functions and outstanding performance, with a sampling rate of 2GSa/s and a maximum bandwidth of 500MHz across the entire range. Support 1MΩ/50 Ω impedance switching to ensure signal integrity; Built in arbitrary waveform generator, capable of outputting 5 standard waveforms and supporting custom arbitrary waveform output; Standard configuration includes 9 triggering modes including edge, pulse, video, slope, timeout, window, code

pattern, interval, and under amplitude, as well as 5 bus analysis and protocol decoding functions including UART, LIN, CAN, SPI, and IIC; 32 automatic measurement and threshold measurement functions, with test results supporting statistical analysis; Two sets of digital voltmeter functions and hardware frequency meter functions; Standard SCPI remote control instructions, convenient for users to build testing systems.

3 Document overview

This document describes how to quickly understand the front and back panels, user interfaces, and basic operation methods of the DSO2D20 series digital oscilloscopes.

Tip:

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

Document number:

202506

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, **Sound** means turn on or off sound of the machine operation.

3 Operation steps

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

Document content conventions:

DSO2D20 series tablet oscilloscope consists of the following models. Unless otherwise specified, this manual uses DSO2D50 as an example to describe the DSO2D20 series and basic operations.

Model	Channel	Sampling Rate	Bandwidth	Signal Source
DSO2C20	2	2GSa/s	200MHz	-
DSO2C35	2	2GSa/s	350MHz	-
DSO2C50	2	2GSa/s	500MHz	-

Model	Channel	Sampling Rate	Bandwidth	Signal Source
DSO2D20	2	2GSa/s	200MHz	1
DSO2D35	2	2GSa/s	350MHz	1
DSO2D50	2	2GSa/s	500MHz	1

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

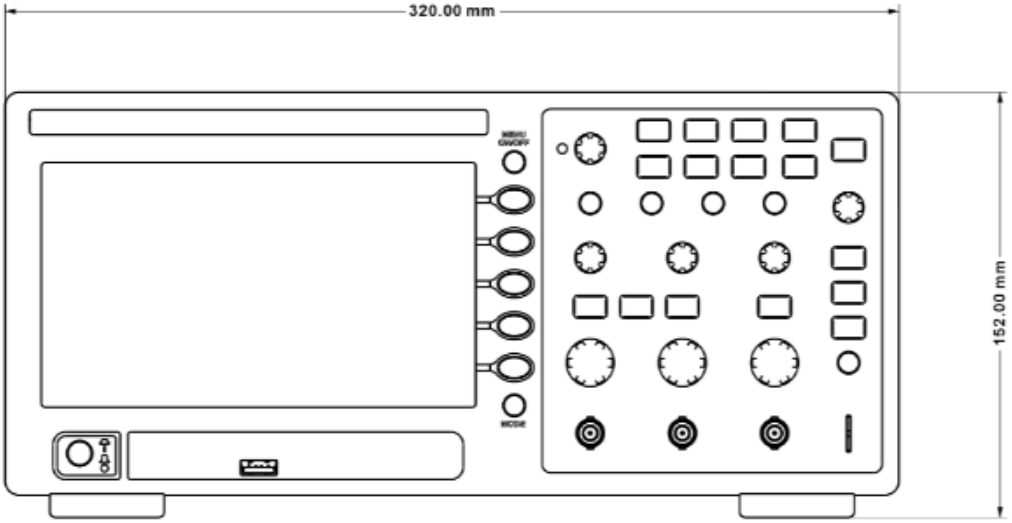


Figure 4.1 Front view

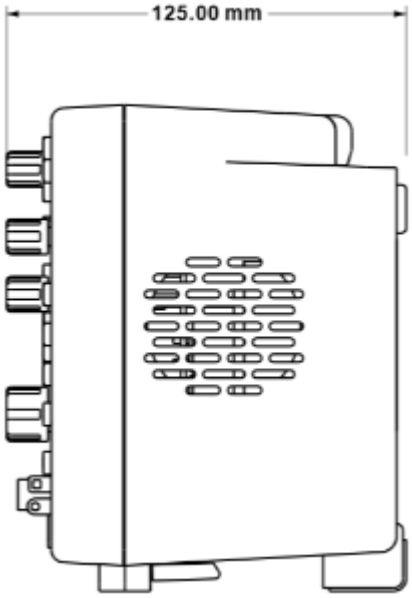


Figure 4.2 Side view

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；50Watts MAX. 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 SET]**.
- Observe the pictures on the waveform to see if the square wave signal is displayed normally. As shown in the following figure.

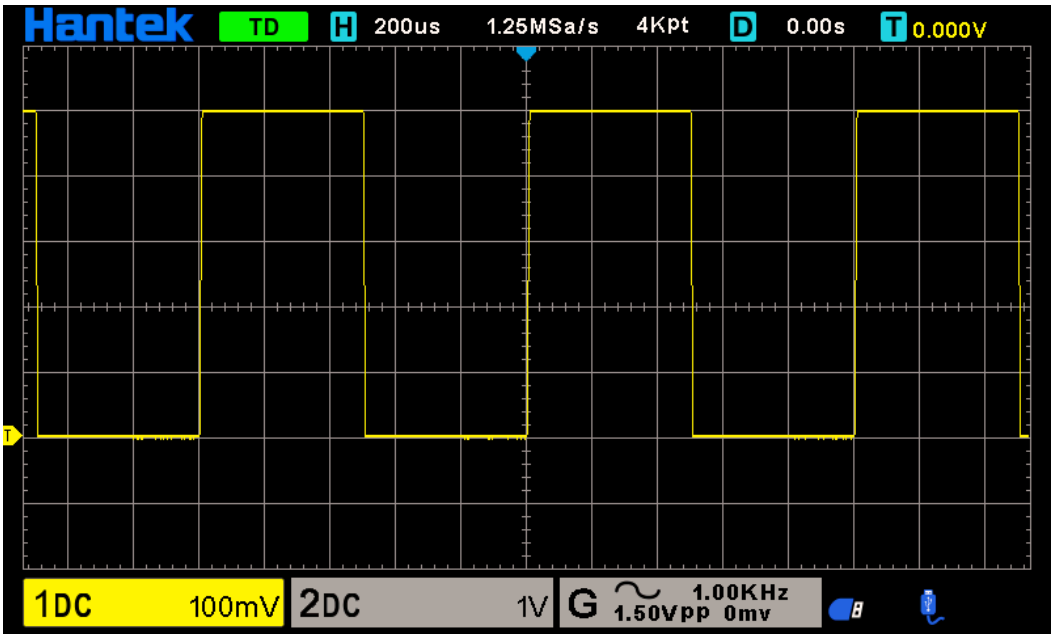


Figure 4.5 Square Waveform Signal

- Check CH2 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.

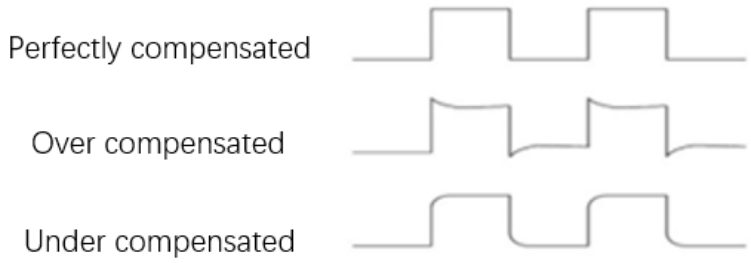


Figure 4.6 Probe Compensation

- 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.

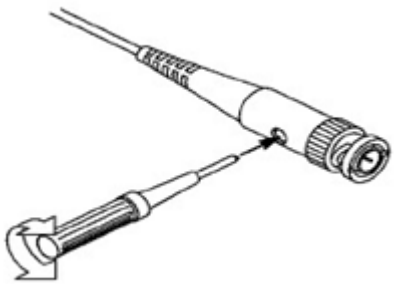


Figure 4.7 Adjusting capacitance

4.4 Front Panel Overview

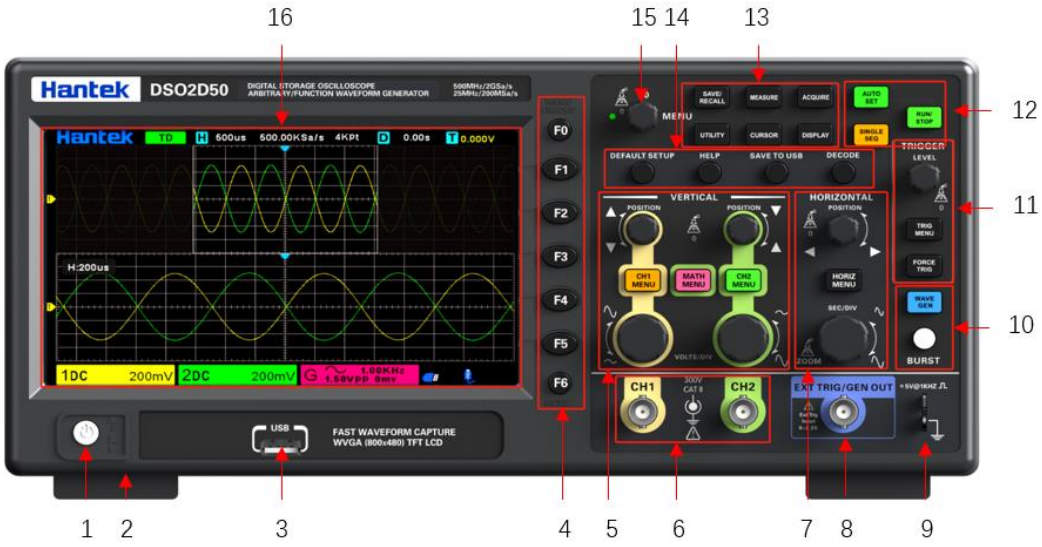


Figure 4.8 Front Panel

Table 4.1 Front Panel Description

No.	Description
1	Power Key
2	Bracket
3	USB HOST Interface
4	Menu Selection Key
5	Vertical Control System, Waveform Control System
6	Analog Channel Input
7	Horizontal Control System
8	Signal Source Output (limited to models with signal source)/ External Trigger Input Channel
9	Probe Compensation Signal Output Terminal/Ground Terminal
10	Signal Source (limited to models with signal source)
11	Trigger Control System
12	Quick Keys for Operating Mode

13	Menu Function Buttons
14	Function Shortcut Keys
15	Multi Functional Knob
16	Display

4.5 Front Panel Function Overview

1. **Horizontal:**

-  :

  Horizontal time base knob. Rotate this knob to modify the horizontal time base. Clockwise rotation reduces the time base, counterclockwise rotation increases the time base. 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. Pressing this knob can quickly switch the horizontal time base adjustment mode to "coarse adjustment" or "fine adjustment".

-  :

  Horizontal displacement knob. Rotate this knob to modify the horizontal displacement (i.e. trigger displacement). When turning the knob, the trigger point moves 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 above the screen changes in real-time. Pressing this knob can quickly reset the horizontal displacement (or

delay the scanning displacement).

2. **Vertical:**

- **CH1 MENU and CH2 MENU analog input channel switches:**

  Two channels are labeled with different colors, and the color of the waveform 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".

- **MATH MENU:**

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

3. **Wave Gen:**

- **WAVE GEN:**

  Press the WAVE GEN button to open the signal generator channel.

- **BURST:**

  Burst button.

4. **TRIGGER:**

- **Trigger knob:**

  Modify the trigger level or threshold level. Clockwise rotation increases the level, counterclockwise rotation decreases the level. During the modification process, the trigger level moves up and down, and the trigger level/threshold level value in the upper right corner of the screen changes in real-time. Pressing this knob can quickly set the trigger level value to 50% of the waveform peak to peak value.

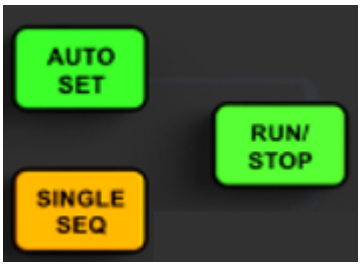
- **TRIG MENU:**

  Press the button to open the trigger operation menu.

- **FORCE TRIG:**

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

5. **Quick keys for operating mode:**



- **AUTO SET:**

  Press this key to enable the automatic waveform setting function. The oscilloscope will automatically adjust the vertical gear, horizontal time base, and trigger mode based on the input signal to achieve the best waveform display state.

- **SINGLE SEQ:**

  Press this key to set the triggering mode of the oscilloscope to single shot.

- **RUN/STOP:**

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

6. **Menu function buttons:**

- **SAVE/RECALL:**

  This key can perform the function of saving and retrieving.

- **MEASURE:**

  Press this key to enter the measurement settings menu.

- **ACQUIRE:**

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

- **UTILITY:**

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

- **CURSOR:**

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

- **DISPLAY:**

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

7. **Function shortcut keys:**

- **DEFAULT SETUP:**

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

- **HELP:**

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

- **SAVE TO USB:**

  Save the screenshot to a USB drive.

- **DECODE:**

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

8. **Multi functional knob:**

Modify setting parameters by rotating the knob.

4.6 Rear Panel Overview

Figure 4.9 Rear Panel

Table 4.2 Rear Panel Description

No.	Description
1	USB DEVICE
2	Power Socket
3	Handle

4.7 Rear Panel Overview

1. **USB DEVICE:**

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

2. **Power Socket:**

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

3. **Handle:**

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

4.8 User Interface

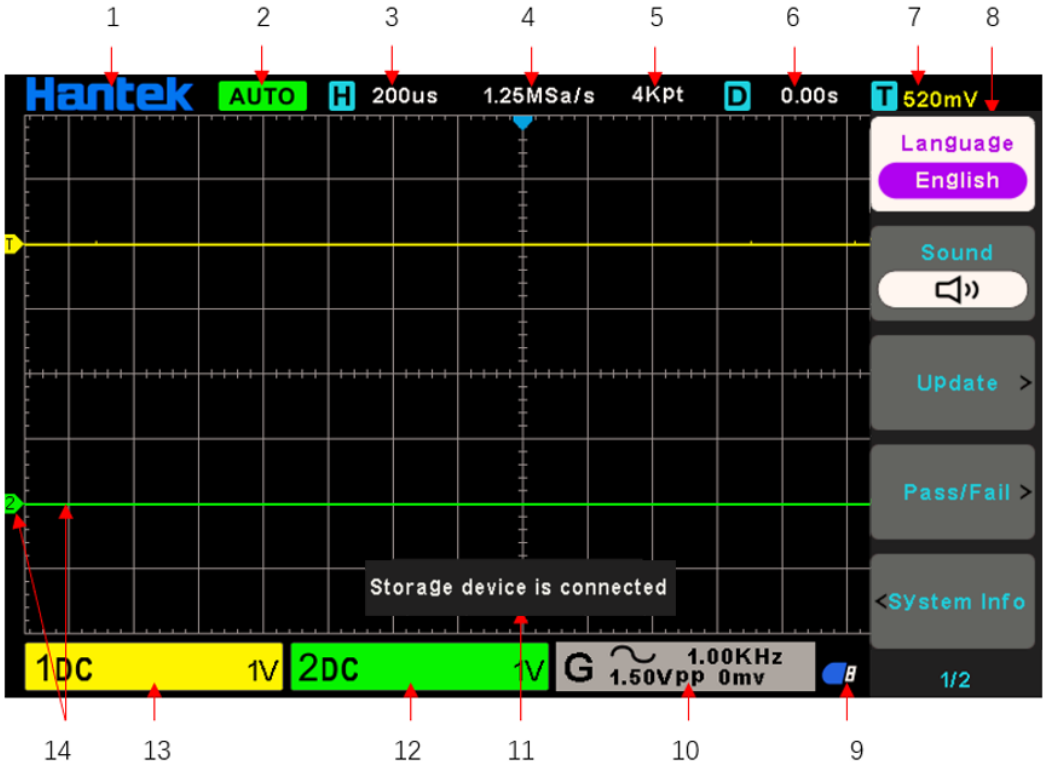


Figure 4.10 User Interface

1. **Hantek logo:**

   Representing the Hantek trademark.

2. **Running status:**

   The operating status of the oscilloscope includes READY, TD, STOP, and AUTO.

3. **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.

4. **Sampling rate:**

   Display the current sampling rate of the analog channel. The sampling rate will change with the variation of the horizontal time base.

5. **Storage depth:**

   Display the current storage depth of the simulated channel. The storage depth will change with the variation of the horizontal time base.

6. **Horizontal displacement:**

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

7. **Trigger level:**

   Display the current trigger level of the analog channel.

8. **Operation menu:**

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

9. **USB:**

   If the icon is lit or activated, it indicates that the USB drive is connected.

10. **Signal generator:**

    Display the open status of the signal generator.

11. **Message box:**

    Display prompt message.

12. **CH2 status label:**

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

13. **CH1 status label:**

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

14. **Simulate channel labels/waveforms:**

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

4.9 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 **[UTILITY]** button again, and the explanation menu for **[UTILITY]** will pop up on the screen.

5 To Set the Vertical System

This series of oscilloscopes provides 2 input channels CH1-CH2, each with an independent vertical control system. The control systems for two channels are the same, and this chapter mainly introduces various setting methods for CH1. Please read this chapter carefully to understand the vertical system settings, functions, and operations of the tablet oscilloscope.

Contents in this chapter:

- To Enable or Disable the Analog Channel
- To Adjust the Vertical Scale
- To Adjust the Vertical Offset
- Channel Coupling
- Bandwidth Limit
- Input Impedance
- Probe Ratio
- Waveform Invert
- Fine tuning

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 in the following way.

- Press the **[CH1]** button to open the channel, and the light on this button will turn on.

If CH1 is already open and currently selected, the label will be as 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

There are several methods to close the simulation channel:

- If CH1 channel is already open and currently selected, press **[CH1]** once to close CH1 channel.
- If the CH1 channel is already open but not selected, press **[CH1]** once to make CH1 selected, and then press the CH1 button again to close the CH1 channel.

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

When 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 the adjustable range. Rotate clockwise to decrease the gear and counterclockwise to increase the gear.

5.3 To Adjust the Vertical Offset

Vertical offset refers to the deviation 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.

Adjust vertical offset:

- Rotate the offset knob in the button area to adjust the vertical offset.

5.4 Channel Coupling

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

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

Tips:

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 limitation function. Setting bandwidth limits can reduce noise in the displayed waveform.

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

Click to open **[CH1]**, then click on the **BW** menu and select the bandwidth limit. You can choose to turn off, 20M, 100M, 200M, 350M, and default to off. After opening the bandwidth limit, the BW flag is displayed at the channel label.

Tips:

Bandwidth limitation reduces noise while also attenuating or eliminating high-frequency components in the signal.

5.6 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.

5.7 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-1000X.

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)
1X	1:1
10X	10:1
50X	50:1
100X	100:1
500X	500:1
1000X	1000:1

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.

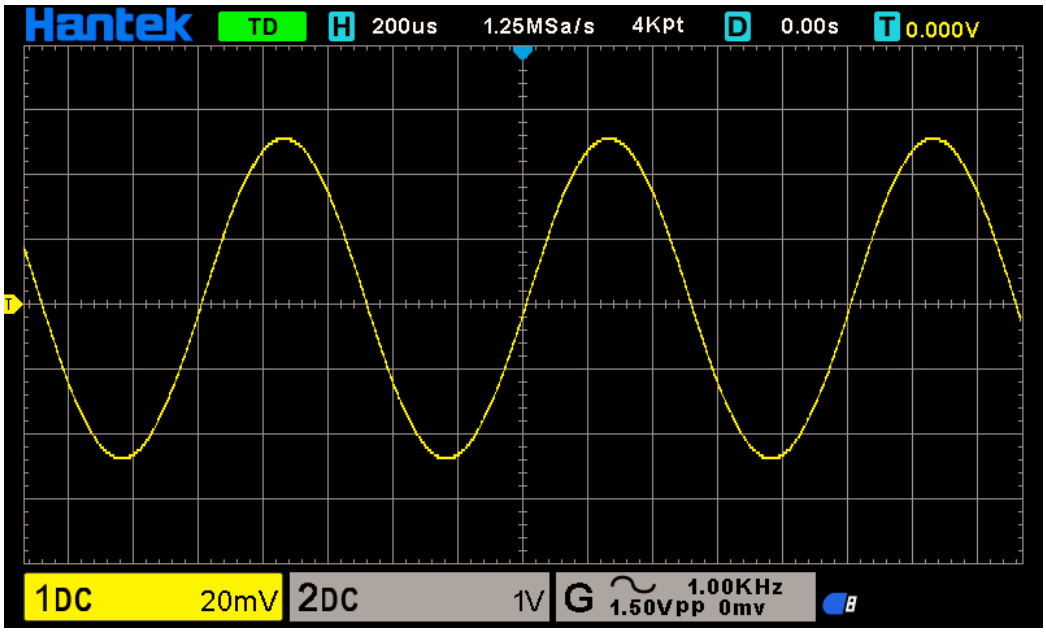


Figure 5.1 "Invert" Off

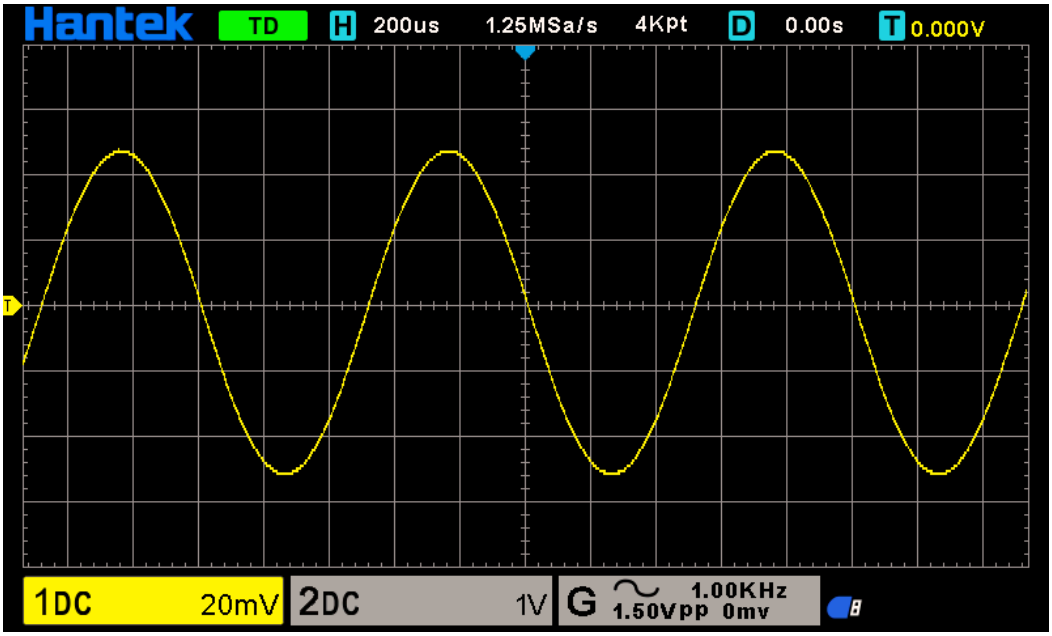


Figure 5.2 "Invert" On

5.9 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.

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:

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

This chapter includes:

- To Adjust the Horizontal Position
- To Adjust the Horizontal Time Base
- Pan or zoom single acquisition or stopped acquisition
- Delayed Sweep
- Horizontal Mode

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.

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

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 2ns/div~100s/div. The default value is 1 us/div.
Changing the horizontal time base will display real-time information on the screen.
The horizontal time base can be adjusted by the following methods:

- Adjust the horizontal time base by rotating the horizontal time base knob.

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 magnify a segment of waveform for viewing waveform details.

Click the button  in the button area, and the screen will be divided into two display areas.

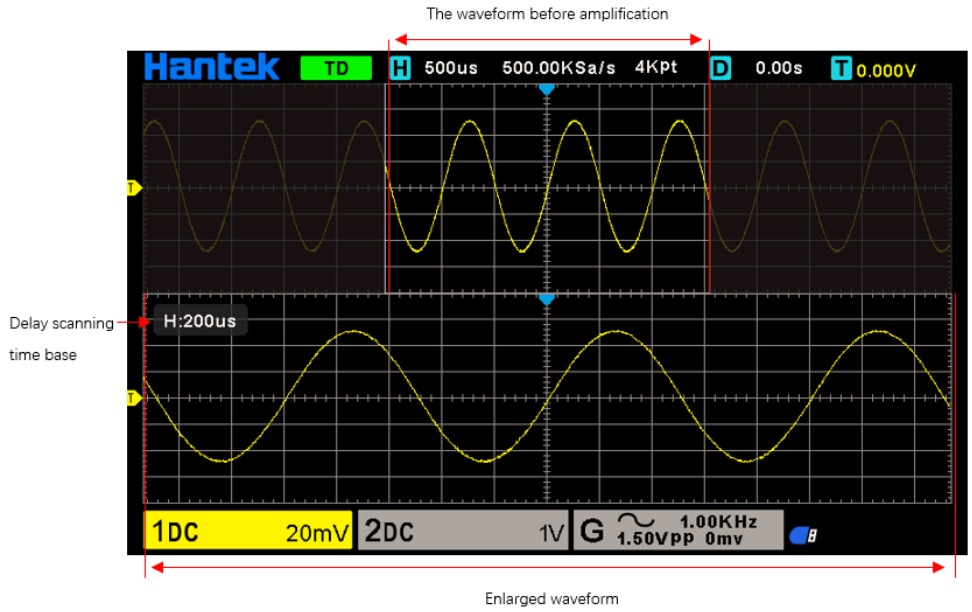


Figure 6.1 Delayed Sweep Mode

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.

6.5 Horizontal Mode

Click the button [**HORIZ MENU**] to set the horizontal mode, and select YT, XY, and scroll.

6.5.1 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 to the left of the trigger point, then waits for the trigger condition to occur, and continues to complete the waveform to the right of the trigger point after the trigger condition occurs, while displaying the currently collected waveform data.

Tips:

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

6.5.2 XY Mode

This series of oscilloscopes supports the XY mode waveform display window "XY window", in which both the X and Y axes represent voltage levels.

Phase difference measurement:

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

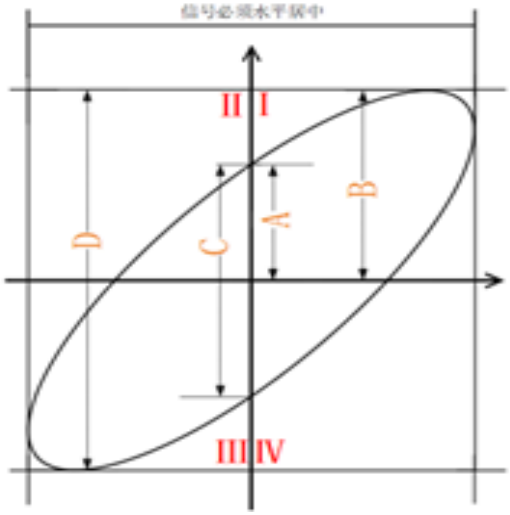


Figure 6.2 Delayed Sweep Mode

According to $\sin\theta = \frac{A}{B}$ or $\sin\theta = \frac{C}{D}$, where θ is the phase difference angle between channels, the definitions of A, B, C, and D are shown in the above figure. Therefore, the phase difference angle can be obtained, that is: $\theta = \pm\arcsin\frac{A}{B}$ 或 $\theta = \pm\arcsin\frac{C}{D}$

If the main axis of the ellipse is within quadrants I and III, the obtained phase difference angle should be within quadrants I and IV, that is, within (0 to π/2) or (3 π/2 to 2 π). If the main axis of the ellipse is within quadrants II and IV, the obtained phase difference angle should be within quadrants II and III, that is, within (π/2 to π) or (π to 3 π/2). The

X-Y function can be used to test the phase changes 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.

Tips:

In general, longer sampling waveforms can result in better display effects, but due to the limitation of storage depth, longer waveform length means that the sampling rate needs to be reduced. Therefore, in this measurement process, reducing the sampling rate appropriately can obtain a better display effect of Li Shayu graphics.

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 XY mode, and rotate the knob to adjust the sampling rate appropriately to obtain a better Li Shayu graph for better observation and measurement.

3 Adjust the vertical knobs corresponding to channels CH1 and CH2 to make the signal easier to observe. At this point, the circle shown in the following figure should be obtained.

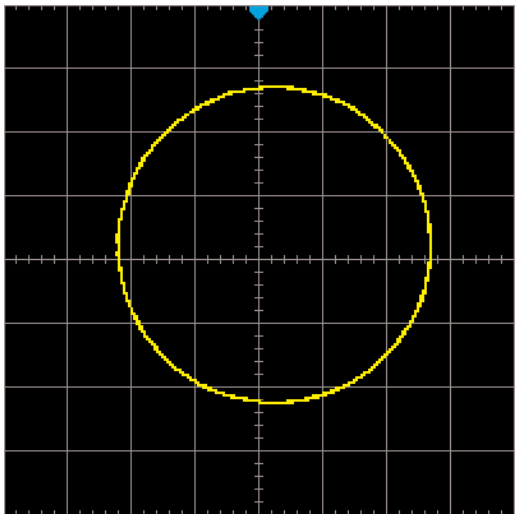


Figure 6.3 Delayed Sweep Mode

4 Observing the measurement results in the above figure and based on the phase difference measurement principle diagram (Figure 6.3), it can be concluded that A/B (C/D)=1, which means the phase difference angle between the input signals of the two channels is = arcsin1=90 °.

6.5.3 Roll Mode

Click on **[HORIZ MENU]** -> **Mode** -> **Roll**, select the scroll mode, and in this mode, the waveform will scroll from right to left to refresh the display. The adjustment range of the

horizontal gear is 100ms to 100s.

7 To Set the Sample System

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

This chapter includes:

- Acquire Type
- Acquire Mode
- Memory Depth

7.1 Acquire Type

Open **[Acquire]**, click on the **Acquire type** menu, and select the collection type: real-time sampling, equivalent sampling.

- Real time sampling: Real time digital technology is used to collect waveforms.
- Equivalent sampling: Reconstruct waveforms using equivalent sampling techniques.

7.2 Acquire Mode

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

7.2.1 Normal Mode

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 Mode

In this mode, the oscilloscope averages multiple sampled waveforms 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 response of the displayed waveform to waveform changes is also slower.

After selecting the **Average mode**, click the "Average Times" button and continuously press the menu button to set it to 4, 8, 16, 32, 64, 128. The default is 4.

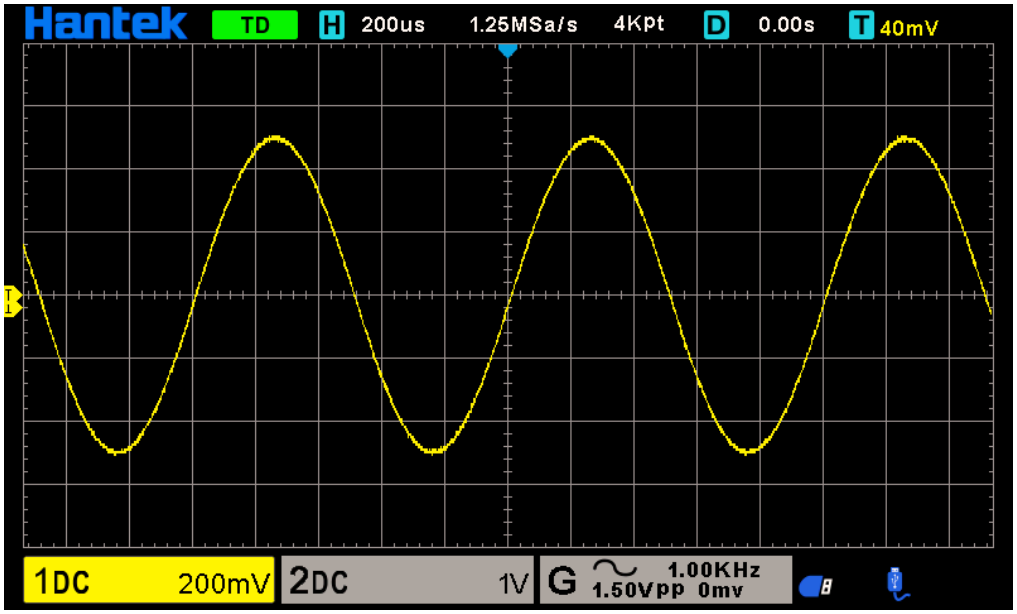


Figure 7.1 Delayed Sweep Mode

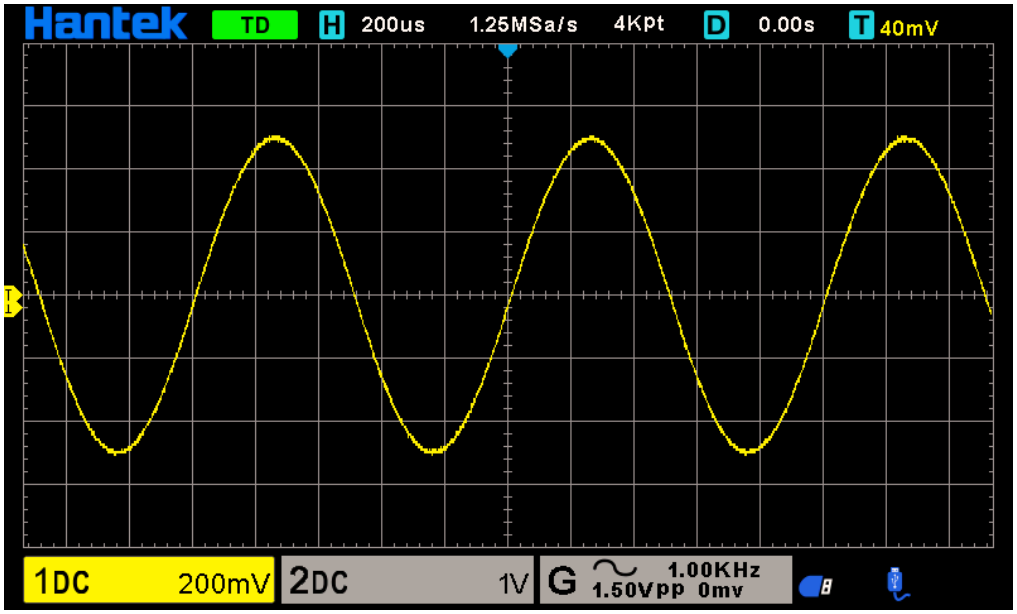


Figure 7.2 Delayed Sweep Mode

7.2.3 Peak Mode

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.

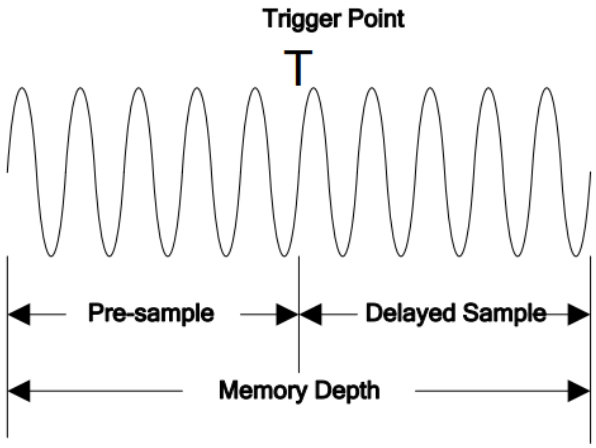


Figure 7.3 Memory Depth

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: 4K, 40K, 80K.

- Full channel mode: Storage depth options: 4K, 40K.

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.

This chapter includes:

- Trigger LEVEL
- Trigger Source
- Trigger Mode
- Trigger Type

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 V1, level V2, and level V1V2.

8.2 Trigger Source

Click on the button area **[TRIG MENU]** to enter the **Source** menu. The available data sources include: CH1~CH2, External, Line.

Analog channel: The input signals of analog channels CH1-CH2 can all be used as trigger sources, and only by selecting the channel with already connected signals as the trigger source can stable triggering be obtained.

External source: External trigger input, triggered on the **[EXT TRIG]** terminal of the oscilloscope front panel. The external trigger signal must be a 0-3.3V CMOS waveform.

Line: Triggered at 50% level of the AC power signal.

Tips:

To avoid damaging this product, do not input signals exceeding 3.3 volts to the **[EXT TRIG/GEN OUT]** connector.

8.3 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.

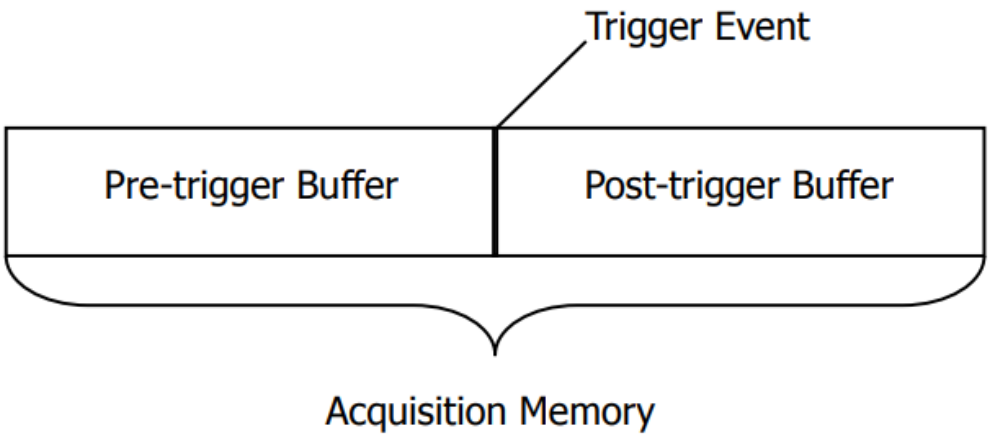


Figure 8.1 Schematic Diagram of the Acquisition Memory

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 **[SINGLR SEQ]** 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.

When a signal characteristic is not understood, the oscilloscope should be set to "automatic" mode, which can ensure that the oscilloscope also displays waveforms when 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 particular signal, especially when the time interval to meet the triggering conditions is relatively long, we need to set the triggering mode to "normal" to prevent the oscilloscope from automatically triggering.

8.4 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
- Pattern Trigger
- Interval Trigger
- Under Amp Trigger
- UART Trigger
- LIN Trigger
- CAN Trigger
- SPI Trigger
- I2C Trigger

8.4.1 Edge Trigger

Edge triggering is triggered at the trigger threshold specified by the input signal for the edge.

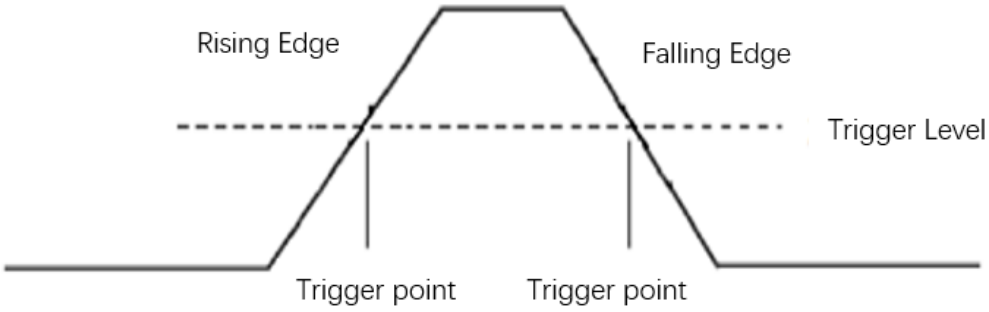


Figure 8.2 Rising/Falling

Click on **[TRIG MENU]** in the button area to open the trigger menu.

1. Type:

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

2. Source:

Click on the **Source** menu tab to select CH1, CH2, external, and Line.

3. Slope Type:

Click on the **Slope** 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. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

5. 50%:

Click on the **50%** menu tab and set the trigger level to the vertical midpoint of the peak to peak value of the trigger signal. The trigger level value is displayed in the upper right corner of the screen.

6. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

Tips:

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

8.4.2 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.

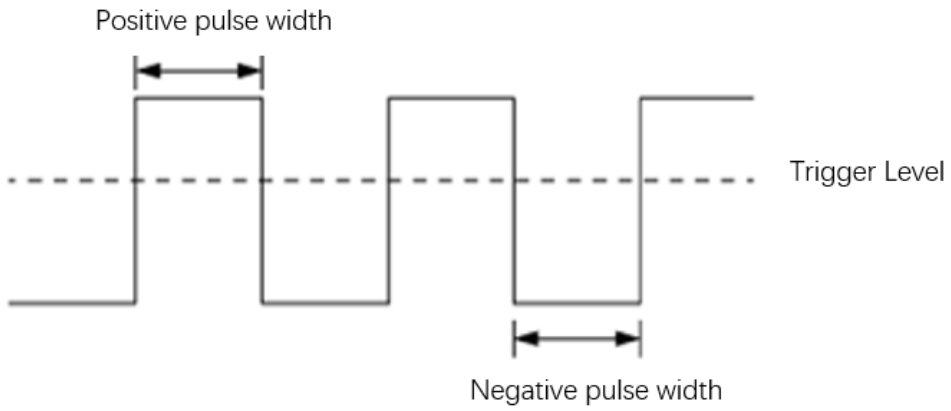


Figure 8.3 Positive Pulse Width/Negative Pulse Width

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 CH1 and CH2 data sources.

3. Polarity:

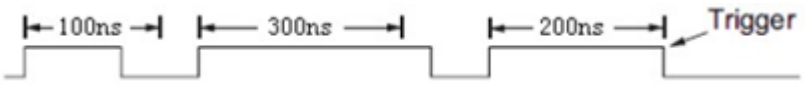
Click on the **Polarity** menu tab to select either positive or negative polarity.

- 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:

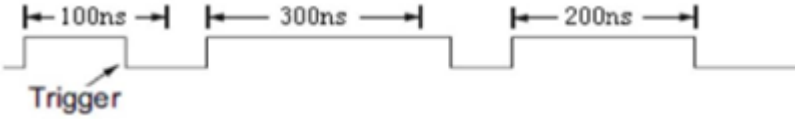
Click on the **When** menu tab and select Trigger Condition.

- =(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. 50%:

Click on the **50%** menu tab and set the trigger level to the vertical midpoint of the peak to peak value of the trigger signal. The trigger level value is displayed in the upper right corner of the screen.

7. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

8. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.3 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 Trigger", and set the trigger.

2. Source:

Click on the **Source** menu tab to select CH1 and CH2 data sources.

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.

Table 8.1 Video Standard

Video Standard	Frame Frequency	Scan Type	TV Scan Line
NTSC	30	Interlaced Scan	525
PAL	25	Interlaced Scan	625

5. Sync:

Click on the **Sync** menu tab and select the desired field or line (ScanLine, LineNum, OddField, EvenField, 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.
- LineNum: 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.
- OddField: displays multiple fields and the oscilloscope only triggers on odd fields.
- EvenField: 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).

7. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

8. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.4 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.

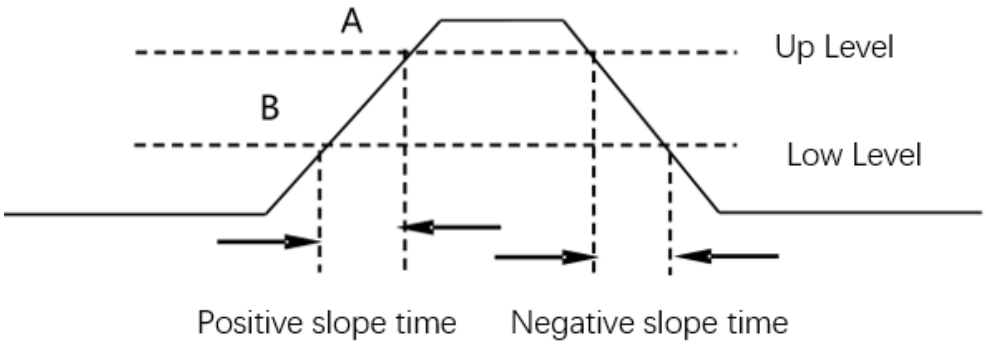


Figure 8.4 Positive Slope Time/Negative Slope Time

Click on **[TRIG MENU]** in the button area to open the trigger menu.

1. Type:

Click on the **Type** menu tab and select "Slope Trigger" to set the trigger.

2. Source:

Click on the **Source** menu tab to select CH1 and CH2 data sources.

3. Slope Type:

Click on the **Slope** 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. Level:

Click on the **Level** menu tab, continuously press the level tab, and select the set level 1, level 2, or level 12.

5. 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.

6. 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.

7. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

8. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.5 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:

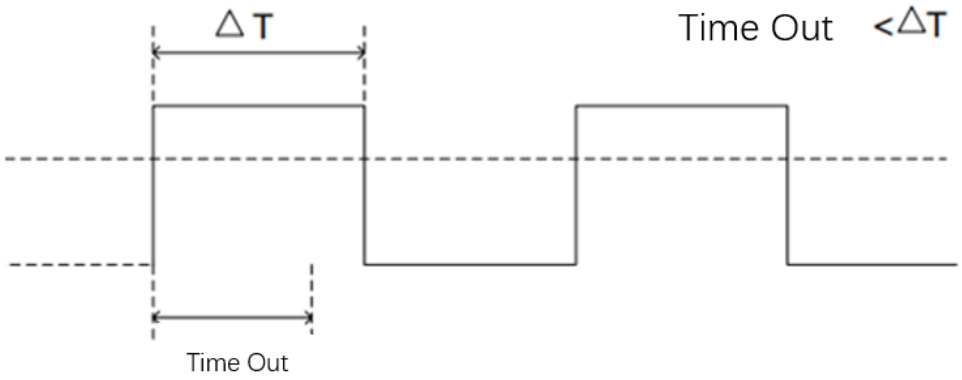


Figure 8.5 Overtime Trigger

Click on **[TRIG MENU]** in the button area to open the trigger menu.

1. Type:

Click on the **Type** menu tab, select "Overtime trigger", and set the trigger.

2. Source:

Click on the **Source** menu tab to select CH1 and CH2 data sources.

3. Slope type:

Click on the **Slope Type** menu tab to select Positive and Negative edges.

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

4. Time:

Click on the **Time** menu tab and set the time by rotating the multifunctional knob V0. The timeout period can be set within the range of 8ns to 10s.

5. 50%:

Click on the **50%** menu tab and set the trigger level to the vertical midpoint of the peak to peak value of the trigger signal. The trigger level value is displayed in the upper right corner of the screen.

6. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

7. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.6 Window Trigger

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

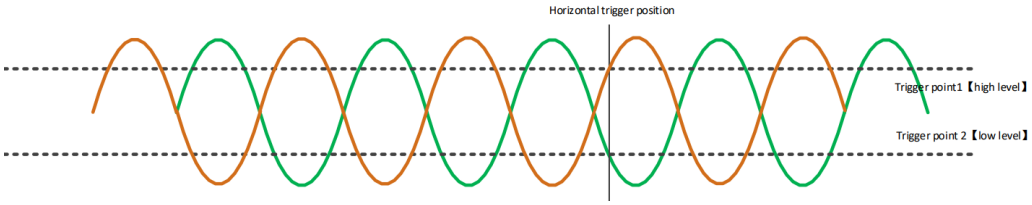


Figure 8.6 Window Trigger

- 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 tab, select "Window Trigger", and set the window trigger.

2. Source:

Click on the **Source** menu tab to select CH1 and CH2 data sources.

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** type from the level selection menu tab.

- Level 1: Only adjust the upper limit of the trigger level, while keeping the lower limit of the trigger level unchanged.
- Level 2: Only adjust the lower limit of the trigger level, while keeping the upper limit of the trigger level unchanged.
- Level 12: 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. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform

is displayed.

5. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.7 Pattern Trigger

Identify triggering conditions by searching for specified code patterns. The code pattern is a combination of channel logic 'and', and each channel can be set as H (high), L (low), X (ignored). You can also specify one channel in the pattern as the rising edge or falling edge (only one edge can be specified). After specifying the edge, if the code patterns of other channels are judged as "true" (i.e. the actual code pattern is consistent with the preset code pattern), the oscilloscope will trigger on the specified edge. If no edge is specified, the oscilloscope will trigger on the last edge that makes the code pattern "true". If all channel codes are set to 'ignore', the oscilloscope will not trigger.

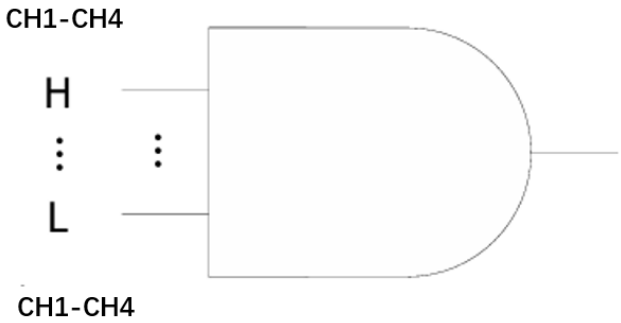
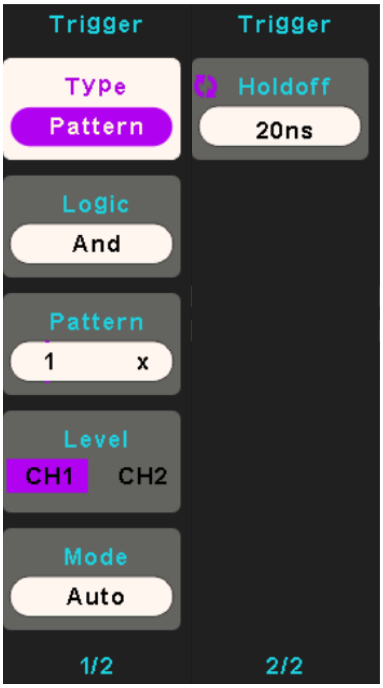


Figure 8.7 Window Trigger

Click on **[TRIG MENU]** in the button area to open the trigger menu.



1. Type:

Click on the **Type** menu tab, select "Pattern trigger", and set the code trigger.

2. Logic:

Click on the **Logic** menu tab and set the logic to AND, OR, or.

3. Pattern:

Click on the **Pattern** menu tab and select: x, 1, 0, Rising, Falling, or Either.

- 1: Set the channel code to "H", which means the level is higher than the trigger level of the channel.
- 0: Set the channel code to "L", which means the level is lower than the trigger level of the channel.
- X: Set the channel code type to "X", which means that this channel is not part of the code type. When all channels of the code are set to "x", the oscilloscope will not trigger.
- :Set the code pattern of the selected channel as rising edge.
- :Set the code pattern of the selected channel as a falling edge.
- :Set the code pattern of the selected channel as rising edge or falling edge.

4. Level:

Click on the **Level** menu tab and select the set level as CH1 or CH2.

5. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the

waveform can be freely collected and run.

- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

6. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.8 Interval Trigger

Triggered when the interval between two consecutive rising (or falling) edges satisfies the set time condition (<,>,!=,=).
Click on **[TRIG MENU]** in the button area to open the trigger menu.

1. Type:

Click on the **Type** menu tab, select "Interval Trigger", and set the trigger.

2. Source:

Click on the **Source** menu tab to select CH1 and CH2 data sources.

3. Slope:

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

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

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

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. Time:

Click on the **Time** menu tab and set the trigger release or pop-up numeric keypad directly by rotating the multifunctional knob V0. The slope time can be set within the range of 8ns to 10s.

6. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

7. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.9 Under Amp Trigger

Underamplitude triggering is used to trigger a pulse that crosses one trigger level but does not cross another trigger level, as shown in the following figure::

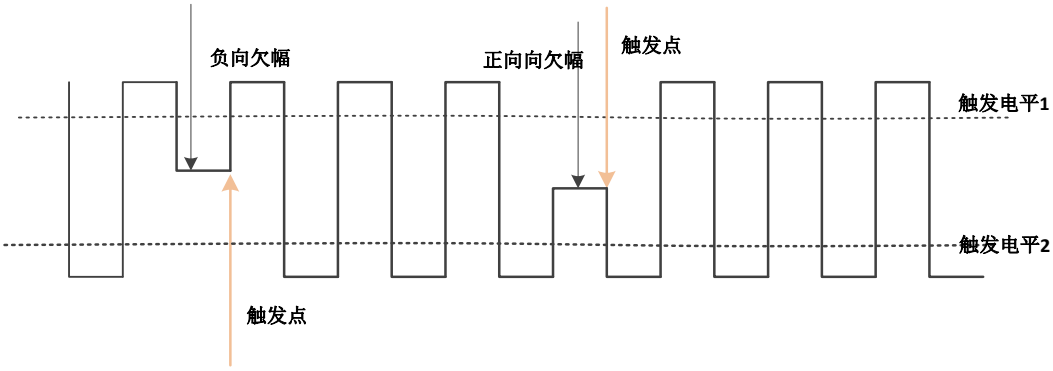


Figure 8.8 Runt Trigger

- Positive underamplitude: The pulse crosses a low level but not a high level.
- Negative underamplitude: The pulse crosses a high level but not a low level.

Click on **[TRIG MENU]** in the button area to open the trigger menu.

1. Type:

Click on the **Type** menu tab and select "Under Amp Trigger" to set the underamplitude pulse trigger.

2. Source:

Click on the **Source** menu tab to select CH1 and CH2 data sources.

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:

Click on the **Level** menu tab, continuously press the level tab, and select the set level 1, level 2, or level 12.

7. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

8. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.10 UART Trigger

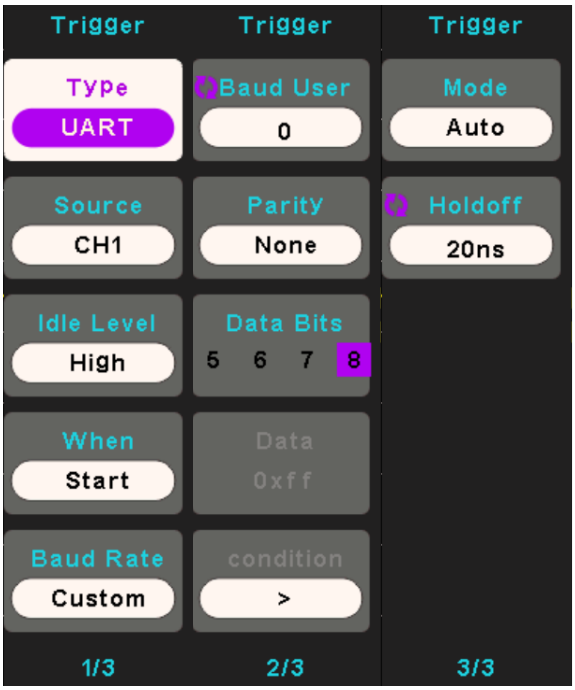
UART bus is a serial communication method used for data transmission between

computers or between computers and terminals.

The UART 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 UART signals.

Figure 8.9 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 tab, select "UART Trigger", and set the UART trigger.

2. Source:

Click on the data source menu tab to select CH1 and CH2 data sources.

3. Idle Level:

Click on the **Idle Level** menu tab and select whether the idle level is high or low.

4. When:

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

- Start: Triggered at the start position of the frame.
- Stop: Triggered at the end of the frame position.
- Spec Data: Triggered in custom data bits. When the data length of the testing equipment is 5-8 bits, it can be used.
  - Press the **condition** menu tab and select the equation qualifier. Choose specific data values that are equal to (=), not equal to (!=), less than (<), or greater than (>).
  - Use the **Data** menu tab to rotate V0 to set the desired data value. The range of data values that can be set is 0x00~0xff. Use the data menu in conjunction with the condition menu.
- Parity ERR: Triggered when the data reception is completed and the verification result does not match the data set by the user.
- COM ERR: Triggered when an error frame is detected.

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. Parity:

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

7. 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 bit, 6 bit, 7 bit, or 8 bit.

8. Data:

Click on the **Data** menu tab and set the data by rotating the multifunctional knob V0. Set the data using V0. When the arrow in the upper left corner of the data menu displays up and down arrows, rotate V0 to set the value of the current data bit; Press V0 again, and the arrow in the upper left corner of the data menu will change to a left or right arrow. Rotate V0 to select the data bit that needs to be set.
 : Horizontal arrow, rotate V0 to select a number.
 : Vertical arrow, rotate V0 to set the value of the selected number.

9. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the

input waveform; When the triggering conditions are not met, the original waveform is displayed.

10. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.11 LIN Trigger

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.

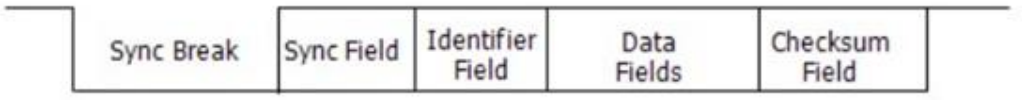
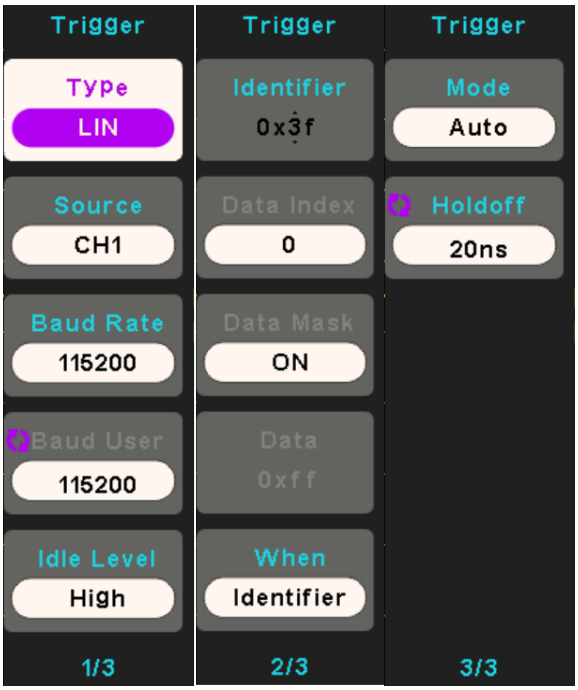


Figure 8.10 Data Frame Format of the LIN Bus

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.

2. Source:

Click on the **Source** menu tab to select CH1 and CH2 data sources.

3. Baud Rate:

Click on the **Baud Rate** menu tab and select the pre-set baud rate from the pop-up sub options.
If there is no baud rate that matches the tested device, you can choose to customize it. Set the baud rate in the custom menu tab.

4. Idle Level:

Click on the **Idle Level** menu tab and select whether the idle level is high or low.

5. Identifier:

Click on the **Identifier** menu tab and rotate V0 to set the identifier. The range is from 0x00 to 0x3f.

6. Data:

Click on the **Data Index** menu tab, set the range from 0 to 3, and you can set four data values.
When clicking on the **Data Mask** menu tab and setting it to "ON", the data in that index will be ignored when triggered; Set to 'OFF', the data on the data line must be consistent with the data in the index in order to trigger.
Click on the **Data** menu tab and set the data by rotating the multifunctional knob V0. Set the data using V0. When the arrow in the upper left corner of the data menu displays up and down arrows, rotate V0 to set the value of the current data bit; Press V0 again, and the arrow in the upper left corner of the data menu will change to a left or right arrow. Rotate V0 to select the data bit that needs to be set.
 : Horizontal arrow, rotate V0 to select a number.
 : Vertical arrow, rotate V0 to set the value of the selected number.

7. 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.

8. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

9. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.12 CAN Trigger

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.

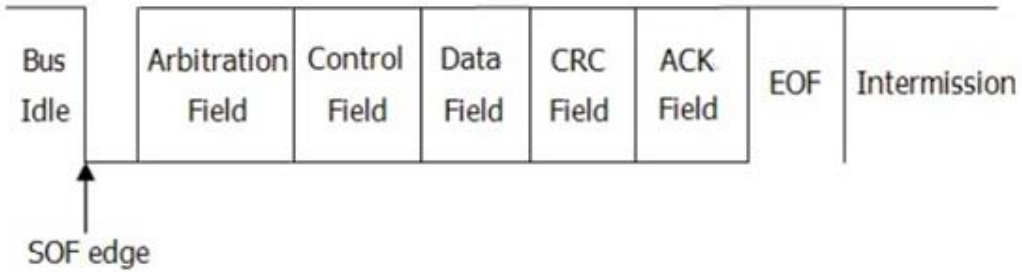


Figure 8.11 Data Frame Format of the CAN Bus

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.

2. Source:

Click on the **Source** menu tab and select CH1-CH2.

3. Baud Rate:

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

4. Idle Level:

Click on the **Idle Level** menu tab and select whether the idle level is high or low.

5. Identifier:

Click on the **Identifier** menu tab and rotate V0 to set the identifier.

6. Data:

Click on the **Data Index** menu tab, set the range from 0 to 3, and you can set four data values.
When clicking on the **Data Mask** menu tab and setting it to "ON", the data in that index will be ignored when triggered; Set to 'OFF', the data on the data line must be consistent with the data in the index in order to trigger.
Click on the **Data** menu tab and set the data by rotating the multifunctional knob V0. Set the data using V0. When the arrow in the upper left corner of the data menu displays up and down arrows, rotate V0 to set the value of the current data bit; Press V0 again, and the arrow in the upper left corner of the data menu will change to a left or right arrow. Rotate V0 to select the data bit that needs to be set.

 : Horizontal arrow, rotate V0 to select a number.
 : Vertical arrow, rotate V0 to set the value of the selected number.

7. 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.
- ID: Triggered on the frame with the specified ID.
- 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.

8. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

9. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.13 SPI Trigger

Under the SPI trigger type, when the 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).

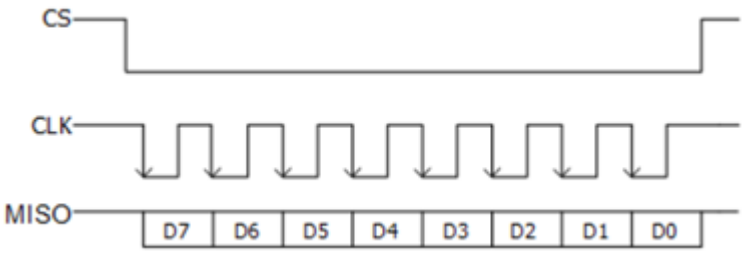


Figure 8.12 Sequential Chart of SPI Bus

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.

2. Source:

Click on the **CLK** and **MISO** menu tabs respectively to select CH1-CH2.

3. Slope type:

Click on the **Slope** 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. Level:

Click on the **Level** menu tab to set the levels for SCK and MOSI/MISO.

5. Data Width:

Click on the **Data Width** menu tab to set the number of bits for the serial data string. Serial data strings can be specified as 4, 8, 16, 24, or 32 bits long.

6. Data:

Click on the **Data Mask** menu tab, which is a hexadecimal number with 0- block, f - not block, and 1- e - block some data.
Click on the **Data** menu tab and set the data by rotating the multifunctional knob V0. Set the data using V0. When the arrow in the upper left corner of the data menu displays up and down arrows, rotate V0 to set the value of the current data bit; Press V0 again, and the arrow in the upper left corner of the data menu will change to a left or right arrow. Rotate V0 to select the data bit that needs to be set.
 : Horizontal arrow, rotate V0 to select a number.
 : Vertical arrow, rotate V0 to set the value of the selected number.

7. Overtime:

Click on the **Overtime** menu tab and set the setting value by rotating the multifunctional knob V0, with a range of 8ns-10s. Before searching for triggers on the oscilloscope, the clock (SCK) signal needs to remain idle for a certain amount of time. When the data (SDA) meets the triggering conditions, the oscilloscope will trigger.

8. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

9. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

8.4.14 I2C Trigger

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.

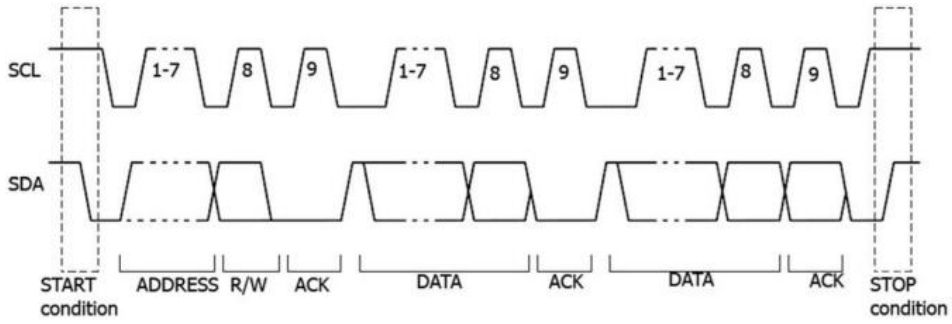
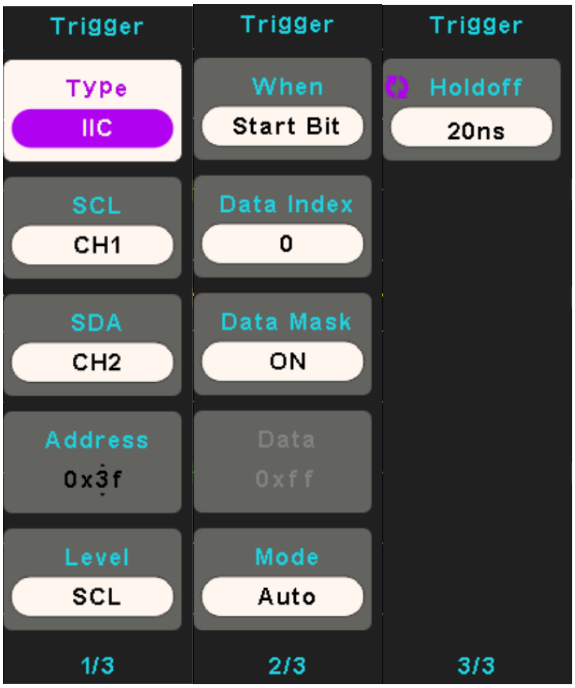


Figure 8.13 Schematic Diagram of I2C Protocol

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.

2. Clock source:

Click on the **SLK** menu tab and select CH1-CH2.

3. Data source:

Click on the **SDA** menu tab and select CH1-CH2.

4. Address:

Click on the **Address** menu tab and set the address by rotating the multifunctional knob V0.

Set the data using V0. When the up and down arrows are displayed in the upper left corner of the data menu, rotate the V0 setting value; Press V0 again, and the arrow in the upper left corner of the data menu will change to a left or right arrow. Rotate V0 to select the desired value.

 : Horizontal arrow, rotate V0 to select a number.

 : Vertical arrow, rotate V0 to set the value of the selected number.

5. Level:

Click on the **Level** menu tab to set the levels of SCL and SDA.

6. When:

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

- Start Bit: IIC start condition trigger detected. (Triggered when SCL is high and SDA data jumps from high level to low level.)
- Stop Bit: IIC stop condition triggered detected. (Triggered when SCL is high and SDA data jumps from low to high.)
- No Ack: Triggered if SDA data is high during any SCL clock bit.
- Address: Trigger the search for the set address value and trigger it on the read/write bit. The address bit width is "7 bits", so the address range can be from 0 to 0x7F.
- Restart: Triggered when a new start condition appears before the stop condition.
- Address and Data: Trigger to search for the set address and data value on the data line (SDA), and trigger on the clock line (SCL) transition edge of the last bit of the data. After selecting 'Address and Data' as the triggering condition:

  Click on the **Data Index** menu tab, set the range from 0 to 3, and you can set four data values.

  When clicking on the **Data Mask** menu tab and setting it to "on", the data in that index will be ignored when triggered; Set to 'off', the data on the data line must be consistent with the data in the index in order to trigger.

  Click on the **Data** menu tab and set the data by rotating the multifunctional knob V0.

  Set the data using V0. When the arrow in the upper left corner of the data menu displays up and down arrows, rotate V0 to set the value of the current data bit; Press V0 again, and the arrow in the upper left corner of the data menu will change to a left or right arrow. Rotate V0 to select the data bit that needs to be set.

   : Horizontal arrow, rotate V0 to select a number.

   : Vertical arrow, rotate V0 to set the value of the selected number.

7. Mode:

Click on the **Mode** menu tab and select the trigger mode as automatic or normal.

- Automatic: When the oscilloscope meets the triggering conditions, it completes one triggered acquisition; When the triggering conditions are not met, the waveform can be freely collected and run.
- Normal: When the oscilloscope meets the triggering conditions, it displays the input waveform; When the triggering conditions are not met, the original waveform is displayed.

8. Holdoff:

Click on the **Holdoff** menu tab and set the Holdoff by rotating the multifunction knob V0. Set the waiting time from one trigger to the next on the oscilloscope to ensure stable display of complex waveforms.

9 Math Operation

This series of oscilloscopes supports algebraic and FFT operations between various analog channel waveforms.

You can enter the **Math** menu through the following methods:

- Click **[Math]** in the button area to enter the mathematical operation menu.

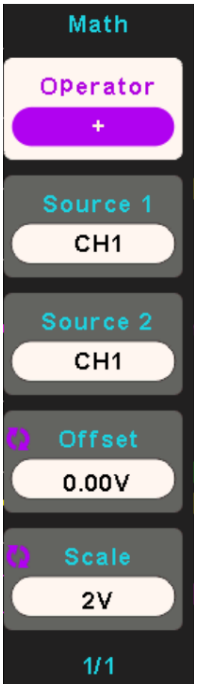
This chapter includes:

- Arithmetic Operations
- FFT

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. Source 1, Source 2:

After clicking on **Source 1** or **Source 2**, a menu will appear where you can select the source as CH1, CH2. Channels in closed state cannot be selected.

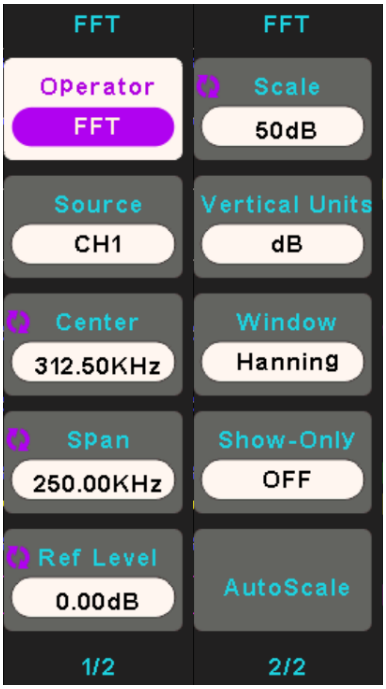
3. Offset:

Click on the **Offset** menu tab and set the vertical offset of the calculation result display window by rotating the multifunctional knob V0.

4. **Scale:**

Click on the **Scale** menu label and set the vertical gear of the calculation result display window by rotating the multifunctional knob V0.

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. Channels in closed state cannot be selected.

3. **Center:**

Click on the **Center** menu tab and rotate the multifunction knob V0 to set the center frequency.

4. **Span:**

Click on the **Span** menu tab and rotate the multifunction knob V0 to set the frequency span.

5. **Ref Level:**

Click on the **Ref Level** menu tab and set the vertical offset of the calculation result display window by rotating the multifunction knob V0.

6. Scale:

Click on the **Scale** menu label and set the vertical gear of the calculation result display window by rotating the multifunctional knob V0.

7. Vertical Units:

Click on the **Vertical Units** menu tab to select dB or VRms.

8. Window:

The use of window functions can effectively reduce spectral leakage effects. This series of oscilloscopes provides the following table, showing 6 FFT window functions, each with different characteristics and suitable waveforms for measurement. Selection should be made based on the measured waveform and its characteristics. Click the **Window** menu to make a selection.

Table 9.1 Window Function

Window Function	Characteristics	Waveforms Applicable to the Window Function
Rectangular	A dedicated window for discontinuous waveforms. It is basically similar to the situation without adding a window.	Transient and short pulse waveforms.
Hanning	Better frequency resolution, poorer amplitude resolution.	Period Waveform.
Hamming	Slightly better frequency resolution than the Hanning window.	Transient and short pulse waveforms.
Blackman-Harris	The best amplitude resolution, the worst frequency resolution.	Single frequency signal, searching for higher harmonics.
Triangle	Better frequency resolution.	Strong narrowband signal
Flattop	Better amplitude resolution, poorer frequency resolution.	Period Waveform.

9. Show Only:

Click on the **Show-Only** menu tab and select whether to display the FFT function separately.

10. AutoScale:

Click the **AutoScale** setting button, and the instrument will automatically adjust the vertical gear and offset of the calculation result to the optimal value based on the current configuration, making it easier for users to observe.

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 **[Measure]** to enter the measurement menu.

This chapter includes:

- Source
- Type
- DVM
- Statistics
- Clear All
- All Measure
- Settings
- Gate
- Quick Measurement after AUTO

10.1 Source

Click on the **Source** source menu tab and select the measured data sources CH1 and CH2. Only open data sources can be selected.

10.2 Type

Click on the **Type** menu tab and select the measurement item using the multifunction knob V0.

1. PkPk: The voltage value from the highest point to the lowest point of the waveform.

   PkPk = Vmax - Vmin

2. Frequency: defined as the reciprocal of period.
3. Average: the arithmetic mean value of the whole waveform or gate area.
4. Vmax: The voltage value from the highest point of the waveform to GND (ground).
5. Vmin: The voltage value from the lowest point of the waveform to GND (ground).
6. Period: defined as the time between the middle threshold points of two consecutive, like-polarity edges.
7. Vtop: The voltage value from the flat top of the waveform to GND (ground).
8. Vmid: The actual voltage value corresponding to the intermediate value of the measurement threshold.
9. Vbase: The voltage value from the flat bottom of the waveform to GND (ground).
10. Vamp: The voltage value from the top to the bottom of the waveform.

    Amplitude = Vtop - Vbase

11. RMS: The root mean square value on the entire waveform or gating region.
12. OverShoot: The ratio of the difference between the maximum value of the rising edge and the top value of the waveform to the amplitude
13. PreShoot: The ratio of the difference between the minimum value of the falling edge of the waveform and the bottom value to the amplitude.
14. PerRms: The root mean square value within a period.

    $$\mathrm{RMS} = \sqrt{\frac{\sum_{i=1}^{n} x_i^2}{n}}$$

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

15. PrdAvg: The arithmetic mean 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 points measured.

16. Rise Time: indicates the time for the signal amplitude to rise from the threshold lower limit to the threshold upper limit.
17. Fall Time: indicates the time for the signal amplitude to rise from the threshold upper limit to the threshold lower limit.
18. +Width: indicates the time between the threshold middle value of a rising edge to the threshold middle value of the next falling edge.
19. -Width: indicates the time between the threshold middle value of a falling edge to the threshold middle value of the next rising edge.
20. +Duty: indicates the ratio of the positive pulse width to the period.

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

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

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

22. FRR: ↑ A -> ↑ B, the time difference between the first rising edge of data source A and data source B.
23. FFF: ↓ A -> ↓ B, the time difference between the first falling edge of data source A and data source B.
24. FOV: The ratio of the difference between the maximum value of the falling edge and the top value of the waveform to the amplitude.
25. FPRE: The ratio of the difference between the minimum value of the rising edge and the bottom value of the waveform to the amplitude.
26. Bandwidth: The time from the first edge of the data source to the last edge of the data source.
27. FRF: ↑ A -> ↓ B, the time difference between the first rising edge of data source A and the first falling edge of data source B.
28. FFR: ↓ A -> ↑ B, the time difference between the first falling edge of data source A and the first rising falling edge of data source B.
29. LRR: ↑ A -> ↑ B, the time between the first rising edge of data source A and the last rising edge of data source B.
30. LRF: ↑ A -> ↓ B, the time between the first rising edge of data source A and the last falling edge of data source B.
31. LFR: ↓ A -> ↑ B, the time between the first falling edge of data source A and the last rising edge of data source B.

32. LFF: ↓ A -> ↓ B, the time between the first falling edge of data source A and the last falling edge of data source B.

10.3 DVM

Click on the **DVM** menu tab to enter the settings menu of the digital voltmeter.

1. Enable:

Click on the **CH1 Enable** or **CH2 Enable** menu tab and select to turn on or off CH1 Enable or CH2 Enable. Only open channels can be set.

2. Type:

Click on the **CH1 Type** or **CH2 Type** menu tab and select the type displayed on the digital voltmeter: DC RMS, AC RMS, DC.

- DC RMS: Display the root mean square value of the collected data.
- AC RMS: displays the root mean square value of the collected data with the DC component removed.
- DC: Display the DC value of the collected data.

3. Return:

Click on the **Return** menu tab to go back to the previous menu.

10.4 Statistics

Click on the **[Meas]** -> **Statistics** menu tab and select to turn on or off the statistics function. Opening the statistics function will display the current value, average value, minimum value, maximum value, root mean square error, and count of the opened measurement items.

10.5 Clear All

Click on **[Meas]** ->**Clear All** Menu Tags, and clear all measurement parameter statistical functions displayed on the screen.

10.6 All Measure

Click on the **[Meas]** -> **All Measurement** menu tab, and all measurement items will be displayed on the screen.

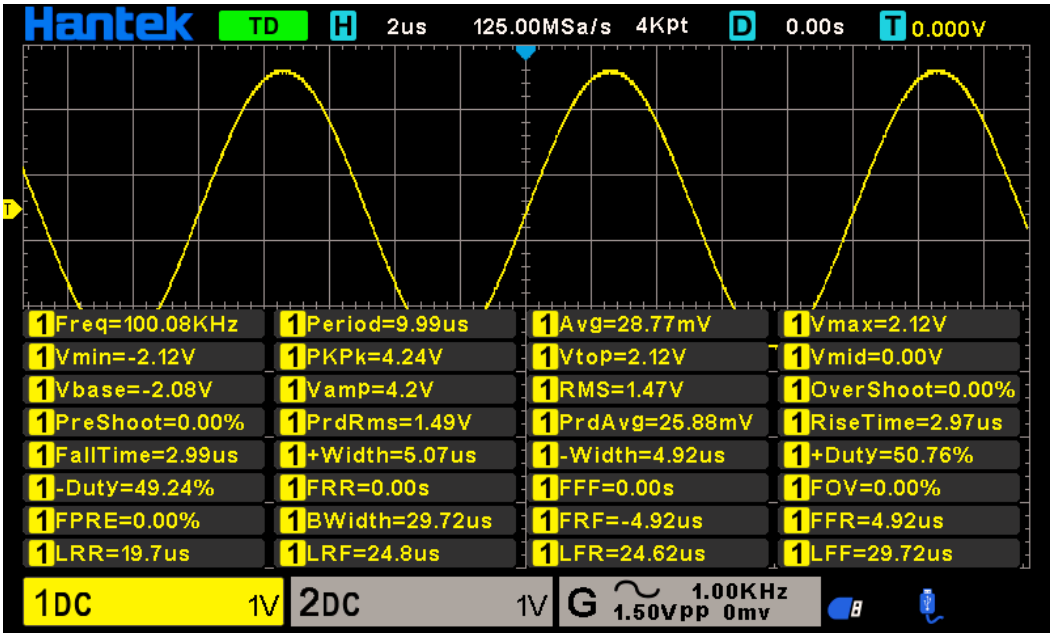


Figure 10.1 All Measure

10.7 Settings

Click on **[Meas]** -> **Settings** menu tab to open the settings menu. When selecting 8 types of delay measurements: FRR, FFF, FRF, FFR, LRR, LRF, LFR, and LFF, the data source selected in the measurement menu is set as data source A. In the settings menu, the opened channel can be set as data source B for delay measurement.

10.8 Gate

Click on the **[Meas]** -> **Gate** Control Menu tab to open the Gate Control Settings menu. Gate control measurement can only be opened when the measurement type is enabled. After opening the door control measurement, the measurement result only measures the waveform between cursor A and B.

10.9 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.

11 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.

- 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 source channel amplitude unit).

Click the **[Cursor]** button in the key area to enter the cursor settings menu. The selected modes are: none, manual, and tracking. When selecting manual cursor or tracking cursor, a window for cursor measurement will appear on the screen, displaying the measurement results.

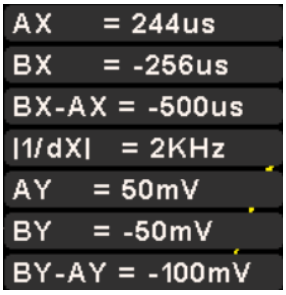
- Manual Mode
- Track Mode

11.1 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.

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

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

AY: The Y value at cursor A.

BY: The Y value at cursor B.

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

1. Source:

Click on the **Source** menu tab, and the optional source channels are CH1, CH2, MATH.Only the currently open channels are selectable.

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. 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.

6. 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.

7. 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.

8. Test example:

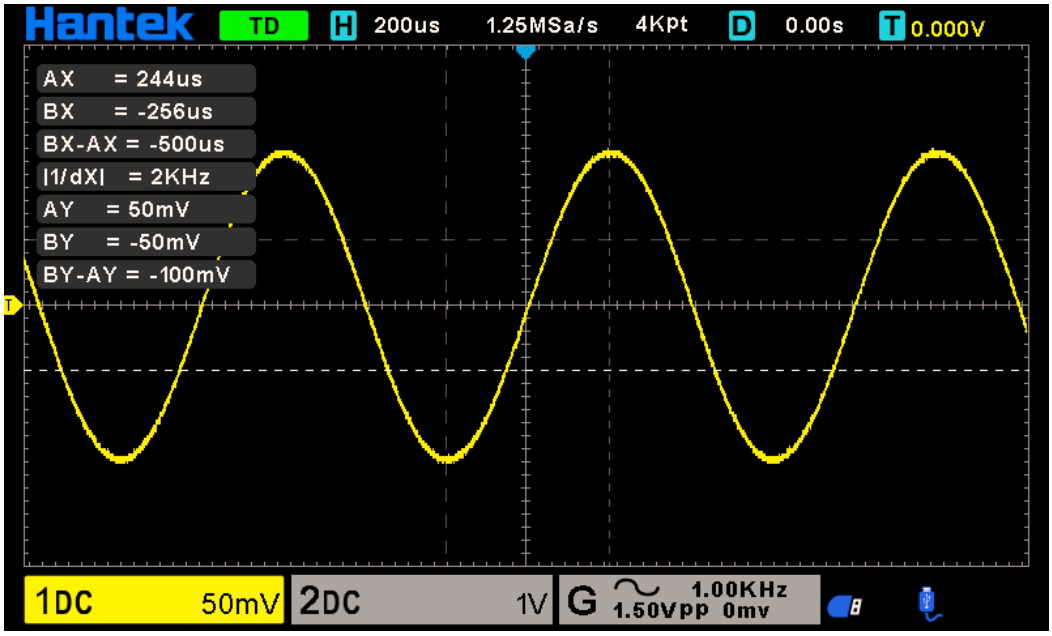


Figure 11.1 Manual Cursor Measurement Example

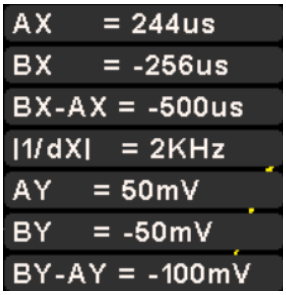
11.2 Track Mode

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.

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

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

AY: The Y value at cursor A.

BY: The Y value at cursor B.

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

1. Source:

Click on the **Source** menu label, and the available information sources include CH1~CH2 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:

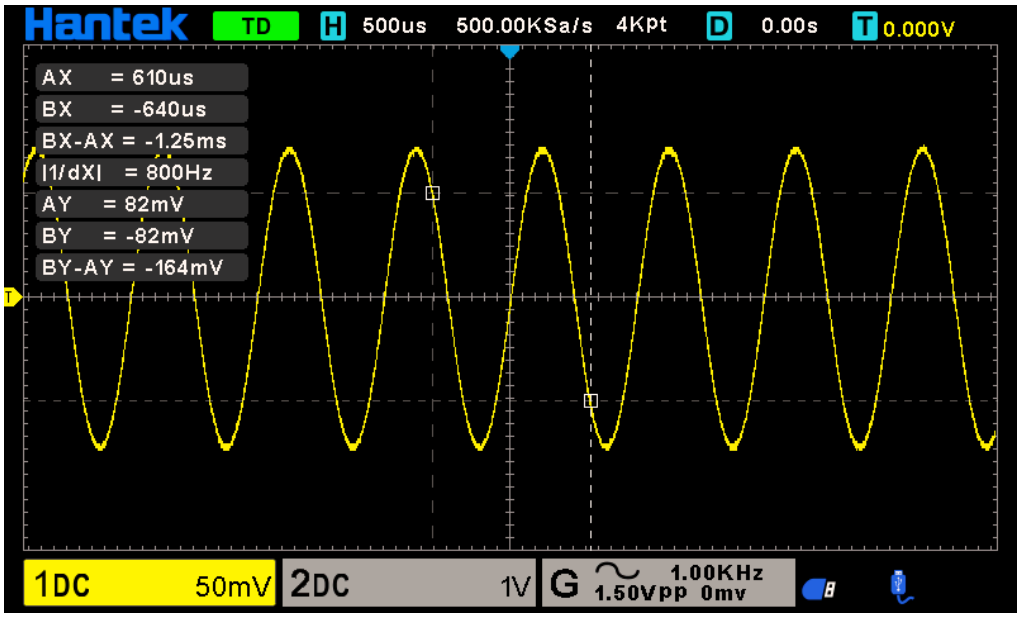


Figure 11.2 Track Measurement (before Horizontal Expansion)

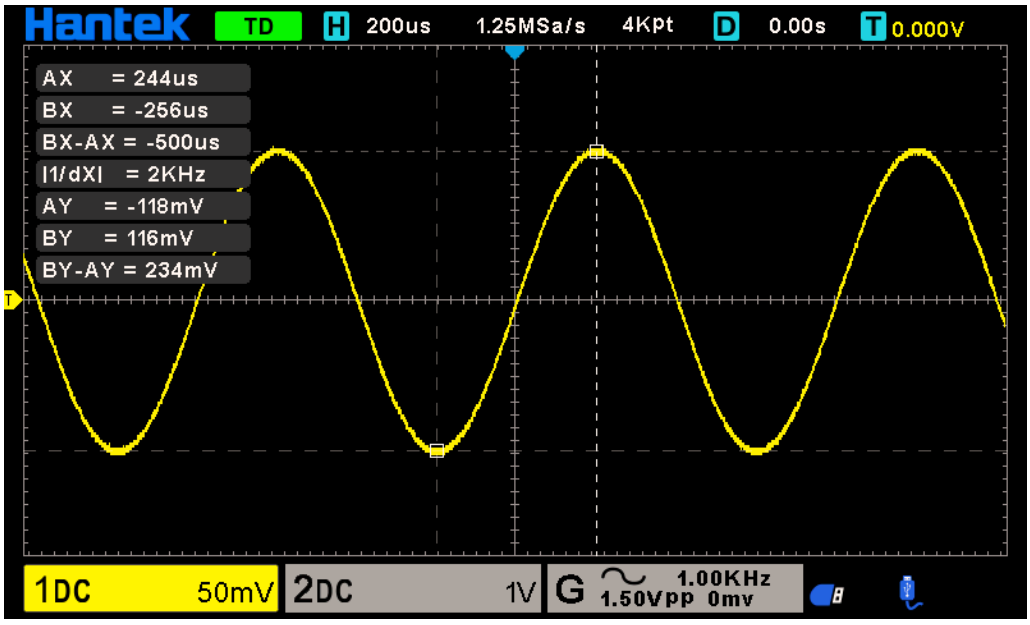
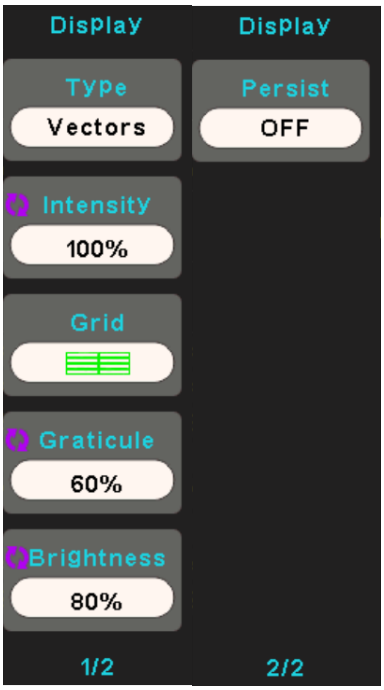


Figure 11.3 Track Measurement (after Horizontal Expansion)

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:

- Type
- Intensity
- Gird
- To Set the Gird Brightness
- To Set the Screen Brightness
- Persist

12.1 Type

Click on the **Type** menu tab to select between Vector and Point. Choose whether to display waveforms as vectors or dots. Vector settings will fill the gaps between adjacent sampling points in the display; The point setting only displays sampling points.

Vector type sampling points are displayed by connecting lines between them. 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 Intensity

Click the **Intensity** menu button and set the brightness value through the multifunctional knob V0, with a range of 0% to 100%.

12.3 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.

12.4 To Set the Grid Brightness

Click on the **Gridicule** 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%.

12.5 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%.

12.6 Persist

In the display menu, click on the afterglow menu tab and select the afterglow time (off, infinite, 1s, 5s, 10s, 30s).

- Minimum value: The waveform that changes at a high refresh rate can be observed.

- Specific value: Can observe burrs with slow changes or low probability of occurrence. The afterglow time can be set to 1 s, 5 s, 10 s, and 30 s.
- Infinite: When the oscilloscope displays a newly acquired waveform, it will not clear the previously acquired waveform. The collected waveforms will be displayed in colors with lower brightness, while newly collected waveforms will be displayed in normal brightness and color. Using infinite afterglow to measure noise and jitter, capturing occasional events.

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:

- Language
- Sound
- Update
- Pass/Fail
- System Info
- Calibrate
- FrontPanel Selftest
- Legal Information

13.1 Language

Click on the **[Utility]** -> **Language** menu tab to switch between 14 languages, including Chinese or English.

13.2 Sound

Click on the **[Utility]** -> **Sound** menu tab, and the sound can be turned off or on.

13.3 Update

Click on **[Utility]** -> **Upgrade** menu tab.
This series of oscilloscopes can experiment with upgrading firmware for USB storage devices, and the entire process takes approximately 5 minutes.
Follow these steps to upgrade the firmware:

- Insert the USB storage device with the upgrade package into the USB HOST interface on the front panel of the oscilloscope.
- Press the **[Utility]** button to enter the auxiliary menu.
- Press the **Upgrade** -> **Upgrade Firmware** software button.
- Rotate V0 to select the upgrade file and press V0 to confirm. Then press the **Start Upgrade** soft key to upgrade the firmware.
- After the upgrade is completed, the oscilloscope will restart and the firmware version will also be updated. The oscilloscope should self calibrate after upgrading.

13.4 Pass/Fail

Testing functions can better monitor changes in signals. You can enter the testing menu through the following methods:

- In **[Utility]** -> **Pass/Fail**, 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 CH2.

13.4.3 Regular

Click on the **Regular** menu tab in the pass test menu to create pass/fail rules.

1. Vertical

Click on **Regular** -> **Vertical**, use the V0 knob to set the vertical tolerance range: 0.020div -4.00div.

2. Level

Click on **Regular** -> **Horizontal**, use the V0 knob to set the horizontal tolerance range: 0.025div-8.00div.

3. Create

Click on **Regular** -> **Create**, and create a rule template based on the vertical and horizontal settings.

4. Save

Click on **Regular** -> **Save**, and select the storage location for the rules.

4.1 Save Directory

Click on **Regular** -> **Save** -> **Save To**, select the location to save, and internal flash memory should be stored in 1-10 locations.

4.2 Save

Click on **Regular** -> **Save** -> **Save** to save the rule settings.

4.3 Transfer out

Click on **Regular** -> **Save** -> **Recall** to retrieve the waveform rule settings.

13.4.4 Message

Click on **Pass/Fail** -> **Message**, select to turn on or off the display information for the number of failed attempts.

13.4.5 Mode

Click on **Pass/Fail** -> **Mode**, select Pass, Fail, Pass Ring, Fail Ring.

- Pass: Output a negative pulse train during testing.
- Fail: Negative pulse train output during failed test.
- Pass Ring: When passing the test, output a negative pulse train accompanied by ringing.
- Fail Ring: During the failed test, a negative pulse train is output, accompanied by a ringing sound.

13.4.6 Run

Click on **Pass/Fail** -> **Run**, run pass/fail test.

13.4.7 Stop

Click on **Pass/Fail** -> **Stop** to stop the pass/fail test.

13.4.8 Out Stop

Click on **Pass/Fail** -> **Out Stop**, select on or off, enter STOP state when there is output or continue running when there is output.

13.5 System Info

Click on **[UTILITY]** -> **System Info** to display the software version, hardware version, serial number, and other information of the oscilloscope.

13.6 Calibrate

Click on **[UTILITY]** -> **Calibrate**, press this soft key, and a self calibration dialog box will pop up. Follow the prompts to self calibrate or cancel.
The self calibration program can optimize the oscilloscope signal path with maximum measurement accuracy. You can run this program at any time, but it should be run if the ambient temperature changes by more than 5 ℃ or more. To calibrate more accurately, please turn on the oscilloscope power, preheat for 20 minutes, and then perform self calibration. Follow the on-screen prompts to operate.
To compensate for the signal path, disconnect any probes or cables connected to the front panel input connector, press the **[Utility]** button, select the "Self Calibration" option, and follow the instructions on the display screen.

13.7 FrontPanel Selftest

Click on **[Utility]** -> **FrontPanel Selftest** to test the functionality of all buttons, knobs, and LED lights on the front panel.

13.8 Legal Information

Click on **[Utility]** -> **Legal Information** to display the source code license.

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 UART, 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:

- UART Decode
- LIN Decode
- CAN Decode
- SPI Decode
- IIC Decode

14.1 UART Decode

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

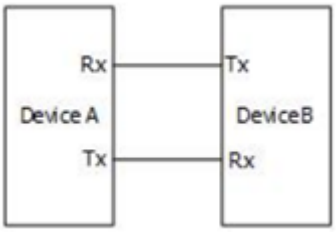


Figure 14.1 Schematic Diagram of UART Serial Bus

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

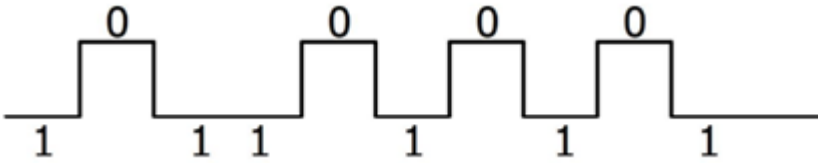
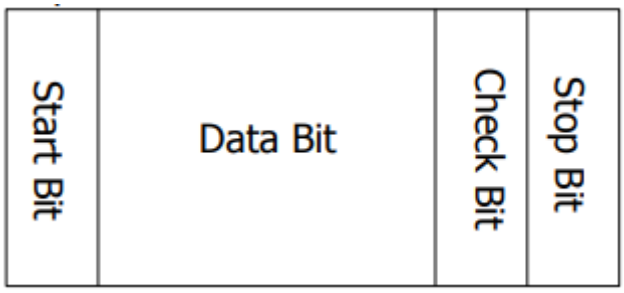


Figure 14.2 Schematic Diagram of Negative Logic

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

2. Source:

Click on the **Source** menu tab and select CH1, CH2.

3. Idle Level:

Click on the **Idle Level** menu tab and set the idle level to low or high.

4. Label Position:

Click on the **Label Position** menu label and set the position using the V0 knob.

5. Baud Rate:

Click on the **Baud Rate** menu tab and select baud rate from the pop-up menu.
Select Custom, you can set the **Baud User** through the V0 knob in the Custom menu tab.

6. Parity:

Click on the **Parity** menu tab, and the available check methods are: no check, odd check, and even check.

- None: There is no checksum during transmission.
- Odd: The total number of "1" in the data bit and check bit is odd. For example, if 0x55 (01010101) is sent, the verification position needs to be filled with 1.
- Even: The total number of "1" in the data bits and check bits is even. For example, if 0x55 (01010101) is sent, it is necessary to fill in 0 in the verification position.

7. Data Bits:

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

8. Disp:

Click on the **Disp** menu tab and select synchronous decoding or protocol monitoring.

14.2 LIN Decode

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 **Type** menu tab and select LIN from the pop-up menu to configure LIN decoding.

2. Source:

Click on the **Source** menu tab and select CH1, CH2.

3. Label Position:

Click on the **Label Position** menu label and set the position using the V0 knob.

4. Baud Rate:

Click on the **Baud Rate** menu tab and select baud rate from the pop-up menu. Select Custom, you can set the **Baud User** through the V0 knob in the Custom menu tab.

5. Idle Level:

Click on the **Idle Level** menu tab and set the idle level to low or high.

6. Disp:

Click on the **Disp** menu tab and select synchronous decoding or protocol monitoring.

14.3 CAN Decode

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 **Type** menu label and select CAN from the pop-up menu to configure CAN decoding.

2. Source:

Click on the **Source** menu tab and select CH1, CH2.

3. Label Position:

Click on the **Label Position** menu label and set the position using the V0 knob.

4. **Baud Rate:**

Click on the **Baud Rate** menu tab and select baud rate from the pop-up menu. Select Custom, you can set the **Baud User** through the V0 knob in the Custom menu tab.

5. **Idle Level:**

Click on the **Idle Level** menu tab and set the idle level to low or high.

6. **Disp:**

Click on the **Disp** menu tab and select synchronous decoding or protocol monitoring.

14.4 SPI Decode

SPI bus communication is based on master-slave configuration and generally consists of 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).

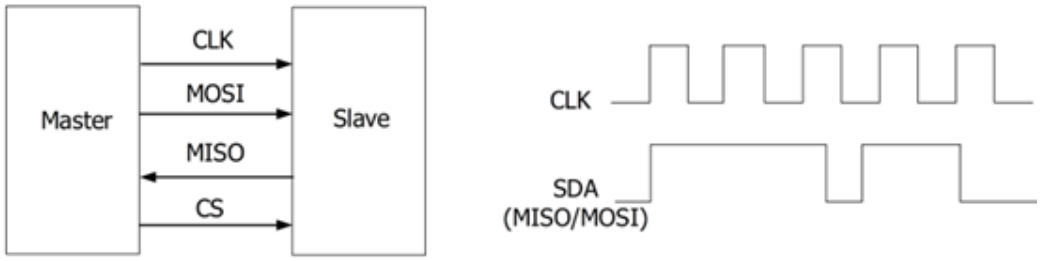


Figure 14.3 SPI Bus Schematic

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. **Source:**

Click on the **SCK** or **MOSI/MISO** menu tab and select CH1, CH2.

3. **Slope:**

Click on the **Slope** menu tab and select whether the slope is rising or falling.

4. **Label Position:**

Click on the **Label Position** menu label and set the position using the V0 knob.

5. **Data Width:**

Click on the **Data Width** menu tab and select bit widths of 4, 8, 16, 24, and 32.

6. **Overtime:**

Click on the **Overtime** menu tab and set the timeout time using the V0 knob.

7. **Disp:**

Click on the **Disp** menu tab and select synchronous decoding or protocol monitoring.

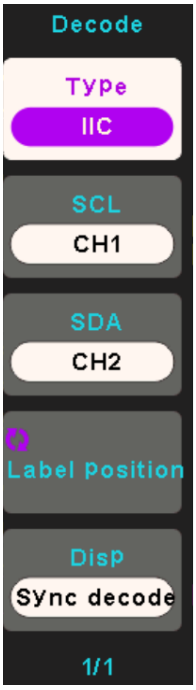
14.5 IIC Decode

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

2. Source:

Click on the **SCL** or **SDA** menu tab and select CH1 or CH2.

3. Label Position:

Click on the **Label Position** menu label and set the position using the V0 knob.

4. Disp:

Click on the **Disp** menu tab and select synchronous decoding or protocol monitoring.

15 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:

- Internal SAVE and RECALL
- External SAVE and RECALL
- File Managerment

15.1 Internal SAVE and RECALL

Click the **[SAVE/RECALL]** button in the button area to open the save/recall menu.

15.1.1 Internal SAVE

1. Save settings:

Click on the **Type** menu tab, select SetUp, and save the settings.

- Save Directory: Click on the **Save To** menu tab, select Internal, and save the current settings of the oscilloscope to the internal memory.
- Location: Click on the **SetUp** menu label, rotate V0 to select the save location. The internal flash memory can store up to 9 files, No.1～No.9.
- Save: Click on the **Save** menu tab to save the current settings of the oscilloscope to the specified location. After a few seconds, a prompt message saying 'Save Successfully' will pop up.

2. Save waveform:

Click on the **Type** menu tab, select the waveform, and save the waveform.

- Save Directory: Click on the **Save To** menu tab, select Internal, and save the current waveform of the oscilloscope to the internal memory.
- Location: Click on the **SetUp** menu label, rotate V0 to select save waveform. The internal flash memory can store up to 9 files, No.1～No.9.
- Save: Click on the **Save** menu tab to save the current waveform of the oscilloscope to the specified location. After a few seconds, a prompt message saying 'Save Successfully' will pop up.

3. Save reference:

Click on the **Type** menu tab, select Reference, and save the reference.

- Data Source: Click on the **Source** menu tab and select CH1, CH2, Math.
- Save Directory: Click on the **Save To** menu tab, select Internal, and save the current reference waveform of the oscilloscope to the internal memory.
- Location: Click on the **SetUp** menu label, rotate V0 to select save reference waveform. The internal flash memory can store up to 9 files, No.1～No.9.
- Save: Click on the **Save** menu tab to save the current reference waveform of the oscilloscope to the specified location. After a few seconds, a prompt message saying 'Save Successfully' will pop up.

15.1.2 Internal RECALL

1. **Recall SetUp:**

Click on the **Type** menu tab, select SetUp, and save the settings.

- Recall From: Click on the **Recall From** menu tab, select Internal, and transfer the saved file to the oscilloscope.
- Location: Click on the **SetUp** menu tab, rotate V0 to select the previously saved location, and bring up the settings.
- Recall: Click on the **Recall** menu tab to transfer out the settings. After completion, a prompt message "Recall Successfully" will pop up.

2. **Recall Waveform:**

Click on the **Type** menu tab, select the waveform, and save the waveform.

- Recall From: Click on the **Recall From** menu tab, select Internal, and transfer the saved file to the oscilloscope.
- Location: Click on the **SetUp** menu label, rotate V0 to select the previously saved location, and bring up the waveform.
- Recall: Click on the **Recall** menu tab to transfer out the waveform. After completion, a prompt message "Recall Successfully" will pop up.

3. **Recall Reference:**

Click on the **Type** menu tab, select Reference, and save the reference.

- Channel: Click on the **Channel** menu tab and select RefA or RefB.
- Recall From: Click on the **Recall From** menu tab, select Internal, and transfer the saved file to the oscilloscope.
- Location: Click on the **SetUp** menu label, rotate V0 to select the previously saved location, and bring up the reference waveform.
- Recall: Click on the **Recall** menu tab to bring up the reference waveform. After completion, a prompt message "Recall Successfully" will pop up.

15.2 External SAVE and RECALL

15.2.1 External SAVE

1. **Save SetUp:**

Click on the **Type** menu tab, select SetUp, and save the settings.

- Save Directory: Click on the **Save To** menu tab, select External, and save the current settings of the oscilloscope to a USB storage device.
- Save: Click on the **Save** menu tab to save the current settings of the oscilloscope to a USB storage device. After a few seconds, a prompt message saying 'Save

Successfully' will pop up.

2. **Save waveform:**

Click on the **Type** menu tab, select the waveform, and save the waveform.

- Save Directory: Click on the **Save To** menu tab, select External, and save the current waveform of the oscilloscope to a USB storage device.
- Save: Click on the **Save** menu tab to save the current waveform of the oscilloscope to a USB storage device. After a few seconds, a prompt message saying 'Save Successfully' will pop up.

3. **Save reference:**

Click on the **Type** menu tab, select Reference, and save the reference.

- Source: Click on the **Source** menu tab and select CH1, CH2, Mathematics.
- Save Catalog: Click on the **Save To** menu tab, select External, and save the current reference waveform of the oscilloscope to a USB storage device.
- Save: Click on the **Save** menu tab to save the current reference waveform of the oscilloscope to a USB storage device. After a few seconds, a prompt message saying 'Save Successfully' will pop up.

4. **Save CSV:**

Click on the **Type** menu tab, select CSV, and save CSV.

- Save: Click on the **Save** menu tab to save the current CSV of the oscilloscope to a USB storage device. After a few seconds, a prompt message saying 'Save sSuccessfully' will pop up.

5. **Save image:**

Click on the **Type** menu tab, select the image, and save the image.

- Picture Invert: Click on the **Picture Invert** menu tab and select whether to turn on or off the image reverse when saving the image.
- Save: Click on the **Save** menu tab to save the current image of the oscilloscope to a USB storage device. After a few seconds, a prompt message saying 'Save Successfully' will pop up.

Press the shortcut key **[SAVE TO USB]** on the front panel to automatically take a screenshot and save the image to an external storage device.

15.2.2 External RECALL

1. **Recall SetUp:**

Click on the **Type** menu tab, select SetUp, and save the settings.

- Recall From: Click on the **Recall From** menu tab, select External, and transfer the saved file to the oscilloscope.
- Recall: Click on the **Recall** menu tab to access the settings. After completion, a

prompt message saying "Recall Successful" will pop up.

2. **Recall waveform:**

Click on the **Type** menu tab, select the waveform, and save the waveform.

- Recall From: Click on the **Recall From** menu tab, select External, and transfer the saved file to the oscilloscope.
- Recall: Click on the **Recall** menu tab to transfer out the waveform. After completion, a prompt message of "Recall Successful" will pop up.

3. **Recall reference:**

Click on the **Type** menu tab, select Reference, and save the reference.

- Channel: Click on the channel menu tab and select RefA or RefB.
- Recall From: Click on the **Recall From** menu tab, select External, and transfer the saved file to the oscilloscope.
- Recall: Click on the **Recall** menu tab to retrieve the reference waveform. After completion, a prompt message will pop up saying "Recall Successful".

15.3 File Management

15.3.1 New File

This operation is only valid when stored externally. This oscilloscope supports English input method. The file or folder name can consist of letters, numbers, and underscores. Below, we will introduce the method of creating files or folders through examples.
Create a file named 'DSOXXXX01'.

1. After correctly inserting the USB flash drive into the oscilloscope, press the **Save** soft key under the **[SAVE/RECALL]** menu to enter the USB storage interface.
2. Press the **Type** soft key and rotate V0 to select a type.
3. Use the **Save To** soft key and select 'External'. Press the **Save** soft key to enter the file management interface.
4. Press the **New** soft key to open the New Folder interface as shown below. Divided into two areas: name input area and keyboard area. Default switch to "keyboard area".
5. Rotate the multifunctional knob V0 to select "Aa" and press V0 to confirm in order to switch the current lowercase English letter to uppercase. Rotate V0 again and select "DSOXXXX01" in sequence, then press V0 to confirm.
6. To delete a name from the current name input area, press the **Switch Focus To** soft key and select 'Name' to switch to the name input area. You can continuously press the **Delete** soft key to delete the characters to the left of the cursor one by one. Rotate V0 to move the cursor position.
7. After completing the input, press the **Save** soft key, and the oscilloscope will create a specified type of file with the file name in the current path.

15.3.2 Delete File

This operation is only valid for external storage (USB storage).

1. After correctly inserting the USB flash drive into the oscilloscope, press the save soft key under the **[SAVE/RECALL]** menu to enter the USB storage interface.
2. Press the **Type** soft key and rotate V0 to select a type.
3. Use the **Save To** soft key and select 'External'. Press the **Save** soft key to enter the file management interface.
4. Use the multifunctional knob V0 to select the file or folder that needs to be deleted, and press the **Delete** soft key.

15.3.3 Rename File

This operation is only valid for external storage (USB storage).

1. After correctly inserting the USB flash drive into the oscilloscope, press the save soft key under the **[SAVE/RECALL]** menu to enter the USB storage interface.
2. Press the **Type** soft key and rotate V0 to select a type.
3. Use the **Save To** soft key and select 'External'. Press the **Save** software to enter the file management interface.
4. Use the multifunctional knob V0 to select the file or folder that needs to be renamed, and press the **Rename** soft key to open the rename interface. For specific operation methods, please refer to the instructions in "New File".

16 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:

- Auto Scale
- Default Setup

16.1 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 16.1 Auto Scale

Parameter	Settings
Acquire	Normal
Cursor	Close
Timebase Mode	YT
Display	Vector
Horizontal	Adjusted
SEC/DIV	Adjusted
Holdoff	20ns
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

16.2 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 16.2 Default Setup

Horizontal	
Horizontal Time Base	200us
Horizontal Position	0s
Delayed Sweep	Off
Timebase Mode	YT
Fine	Off
Horizontal Expansion	Center
Vertical	
Vertical Scale	1V
Vertical Offset	CH1: 2V, CH2: -2V
CH1/CH2	CH1-CH2 On
Channel Coupling	DC
BW Limit	Off
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	4K
Trigger	

Trigger Type	Edge Trigger
Source Selection	CH1
Edge Type	Rising
Trigger Holdoff	1us
Display	
Display Type	Vector
Persistence Time	Off
Intensity	100%
Grid	Line
Gird Brightness	60%
Brightness	80%
AWG	
AWG	Off
Wave	Sine
Frequency	1KHz
Amplitude	1.5V
Offset	0V
Start Phase	0°
Setting Type	Off
Impedance	HighZ
Cursor	
Mode	Off
Math	
Operator	A+B
Operation	Off

17 Arbitrary Waveform Generator

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:

- Output Waveform type
- Modulation
- Burst

17.1 Output waveform type

Click on **[WAVE GEN]** in the button area to enter the signal generator settings interface and select the output of the basic wave.

1. Waveform:

Click on the **Wave** menu tab and select Sine wave, Square wave, Ramp wave, Exp wave, Noise, DC, and Arb1-Arb4 from the pop-up menu.

- Frequency:
  Click on the **Frequency** menu tab to set the frequency value by rotating knob V0.
  DC and noise: no frequency parameters.
- Amplitude:
  Click on the **Amplitude** menu tab to set the amplitude value by rotating knob V0.
- Offset:
  Click on the **Offset** menu tab to set the offset value by rotating knob V0.
- Symmetry/duty cycle:
  Click on the **Symmetry/Duty** menu tab. When the waveform is a square wave, set the waveform duty cycle; When the waveform is a triangular wave, set the waveform to symmetry.
- Impedance:
  Click on the **Impedance** menu tab to select the output impedance of the signal generator as high impedance or 50 Ω.

17.2 Modulation

The signal generator of this series of oscilloscopes has two modulation methods: amplitude modulation and frequency modulation. The modulated waveform consists of a carrier wave and a modulated wave. The carrier signal is a waveform signal output by a signal generator, and the modulation signal can be selected from the built-in sine wave, square wave, or triangular wave of the signal generator.

Click on the button area **[WAVE GEN]** to enter the modulation settings menu.

Click on the **Type** menu tab to select amplitude modulation (AM), frequency modulation (FM), or phase modulation (PM).

1. Amplitude Modulation (AM):

Amplitude Modulation (AM) refers to the variation of the amplitude of the carrier wave with the modulation wave.

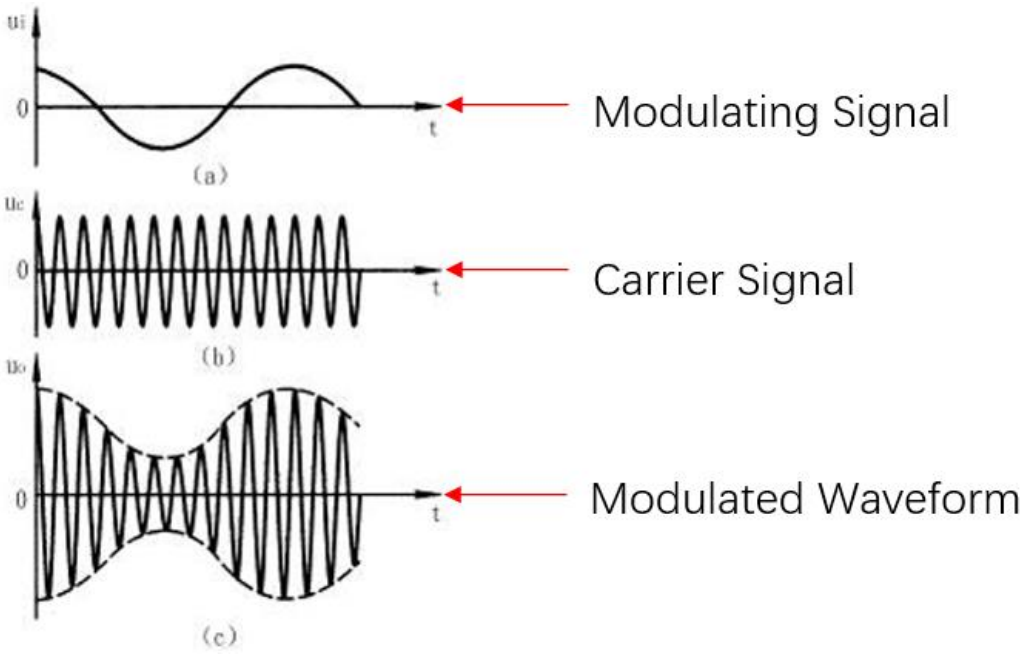


Figure 17.1 AM

- Modulation waveform:
  Click on the **Wave** menu tab to select waveforms including sine wave, square wave, and triangular wave.
- Modulation frequency:
  Click on the **Mod Freq** menu tab to set the frequency value by rotating knob V0.
- Modulation depth:
  Click on the **Depth** menu tab to set the modulation depth from 0-120 by rotating knob V0.

2. Frequency modulation (FM):

Frequency Modulation (FM) refers to the variation of the carrier frequency with the modulation wave.

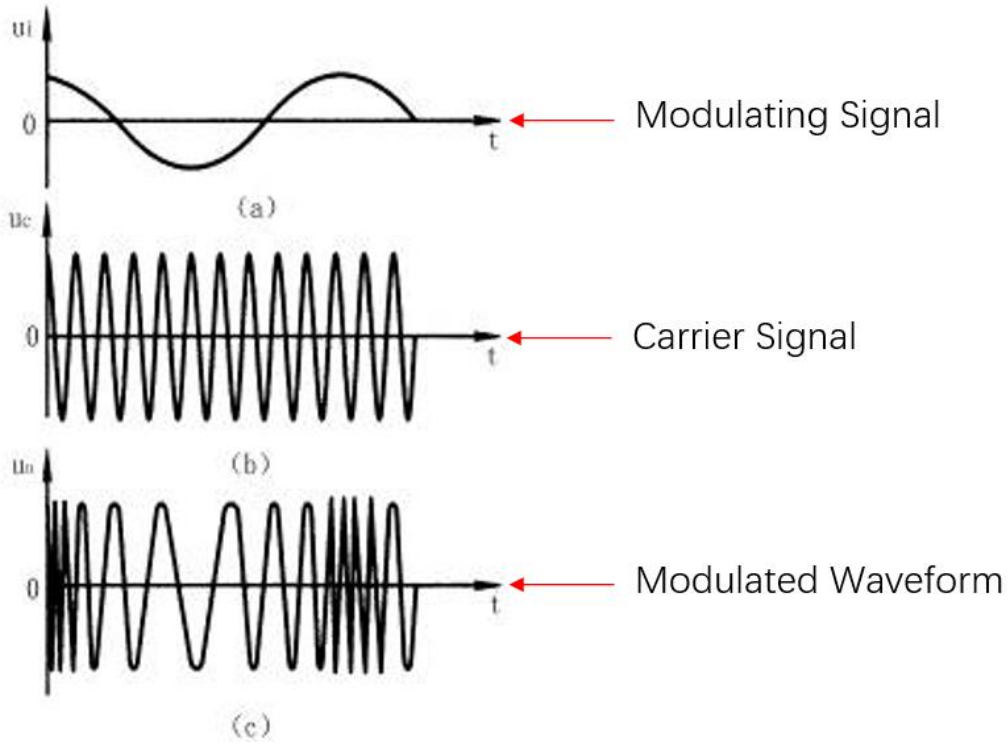


Figure 17.2 FM

- Modulation waveform:

Click on the **Wave** menu tab to select from sine wave, square wave, and triangular wave waveforms.

- Modulation frequency:

Click on the **Mod Freq** menu tab to set the frequency value by rotating knob V0.

- Deviation:

Click on the **Deviation** menu tab to set the offset value by rotating knob V0.

17.3 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.

1. Type of burst:

Click on the **Type** menu tab to choose between multiple cycles or infinite.

- N cycle: When receiving a trigger signal, output 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 when receiving a trigger signal.

2. Counting:

Click on the **count** menu tab to set the number of cycles by rotating knob V0. Only

when multiple cycles are selected for the burst type, can the number of cycles be set.

3. Trigger:

Click on the **Trigger** menu tab, and press the trigger button to output a pulse train with a specified number of cycles.

18 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 DSO2D20 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

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.

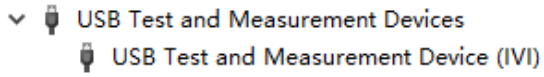
18.1 Remote Control via USB

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.

19 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.

19.1 After pressing the power switch, the oscilloscope did not turn on

1. Check if the power cord on the side of the oscilloscope is connected properly.
2. Check if the power switch is pressed.
3. After completing the above checks, restart the instrument.
4. If the oscilloscope still cannot be turned on normally, please contact your local dealer or directly contact the Hantai technical support department.

19.2 Serious distortion of input signal

1. Check if the oscilloscope probe is properly connected to the BNC channel.
2. Check if the probe is properly connected to the phenomenon being measured.
3. Check if the oscilloscope probe is calibrated properly. If it is not calibrated, please follow the relevant instructions in the manual for calibration.

19.3 No signal appears on the screen

1. Check if the probe is properly connected to the BNC connector of the signal input channel.
2. Check if the channel is open (CH1-CH2 menu buttons).
3. Check if there is signal output from the tested signal channel.
4. If it is a DC signal with a relatively large amplitude, please increase the amplitude range.
5. You can press the automatic measurement button to automatically detect the signal first.
6. If there is still no waveform display, please contact the Hantai technical support department in a timely manner.

19.4 USB flash drive cannot be recognized

1. Check if the USB flash 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 16GB for this oscilloscope.
3. Confirm that the format of the USB flash drive is FAT32 format.
4. Open the oscilloscope again and insert the USB drive for inspection.
5. If you still cannot use the USB drive, please contact the Hantai technical support department in a timely manner.

19.5 Waveform cannot be triggered

1. Check if the triggering data source is consistent with the signal input channel.
2. Check if the trigger level is adjusted correctly. You can press the trigger level knob to return the trigger level to the signal center point.
3. Check if the triggering method is correct, as the default triggering is edge triggering. For different input signals, the triggering method should be selected correctly.

20 Appendix

20.1 Appendix A: Accessories

Order information	Order number
Host model	
2GSa/S, 200MHz 2-channel oscilloscope	DSO2C20
2GSa/S, 350MHz 2-channel oscilloscope	DSO2C35
2GSa/S, 500MHz 42-channel oscilloscope	DSO2C50
2GSa/S, 200MHz 2-channel oscilloscope + AWG	DSO2D20
2GSa/S, 350MHz 2-channel oscilloscope + AWG	DSO2D35
2GSa/S, 500MHz 2-channel oscilloscope + AWG	DSO2D50
Standard accessories	
USB cable	--
power Line	--
Oscilloscope probe	200MHz machine PP200B * 1 350MHz machine PP300B * 1 500MHz machine PP500B * 1
Crocodile Clip Line	DSO2C20 series * 1 DSO2D20 series * 2

20.2 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