AEGIS® Shaft Voltage Tester
Handheld Digital Oscilloscope

AEGIS-OSC-9200

Introduction

This user manual includes important safety information and basic instructions for use of the AEGIS-OSC-9200 handheld digital oscilloscope.

General Safety Summary

Carefully read the following safety precautions to avoid any personal injury or damage to the instrument and any device connected to it. To avoid potential hazards, please use as specified.

Precautions

Before operating this product, special training is required. Improper use may cause personal injury or material damage.

Before moving or transporting this product, please read the "Safe Transportation" chapter.

As with all industrial products, the use of substances that cause allergic reactions such as nickel cannot generally be ruled out. If you experience an allergic reaction while using this product, please consult a doctor.

Before you perform mechanical or heat treatment or disassemble the oscilloscope, please be sure to read and pay special attention to the "Waste Disposal/Environmental Protection" chapter.

In the event of a fire, this product may release harmful substances (gas, liquids, etc.), causing health problems. Therefore, appropriate measures must be taken, for example, protective masks and protective clothing must be worn.

Safety Terms and Symbols

Symbols and Meanings

Symbol	Meaning	Symbol	Meaning
	Warning		Power Switch
	Hazardous Voltage		Double or reinforced insulation
	Earth Ground		Indoor use only
	Lithium battery failure		EU label for separately recycled electrical and electronic equipment

Cues and Meanings

DANGER: Risk of injury or hazard

WARNING: Potential risk of injury or hazard

CAUTION: Potential risk of damage to the instrument or other property

Wherever the Warning symbol occurs, pay close attention to the potential HAZARDS and any protective measures to take.

Safe Operation

The product can only be operated under the safe conditions and positions specified by the manufacturer. Do not obstruct the ventilation of the oscilloscope during use. Failure to comply with the manufacturer's specifications may result in electric shock, fire, serious personal injury, or even death. Applicable local safety regulations and accident prevention rules must be observed in all applications.

Do not use the oscilloscope outdoors on AC power.
Do not place the oscilloscope on unstable or unlevel surfaces.
Do not use the oscilloscope in or near a heat source, and the ambient temperature cannot exceed the maximum temperature in the document or datasheet

Electrical Safety

If you ignore the electrical safety precautions, electric shock, fire and/or serious personal injury or death may occur.

Only use the power adapter and battery specified by the manufacturer to power the oscilloscope. The power adapter can only work within its rated input voltage range.
Only a probe specified by the manufacturer can be used for testing. The use of other probes for may result in electric shock.
Damage to the insulating protective layers of the oscilloscope and accessories may cause electric shock. If an extension cord or terminal block is used for the power supply, it must be checked regularly to ensure safety.
Before use, check whether the power cord or probe is damaged. If the power cord or probe is damaged, do not use.
Do not insert the AC plug of the power adapter into a dusty or dirty socket. Be sure that the plug is firmly inserted into the socket to avoid fire and/or injury due to sparks.
Do not overload any sockets, extension cords or connectors to avoid risk of fire or electric shock.
When the measured voltage Vrms exceeds 30V, take appropriate precautions to avoid any danger.
Unless expressly permitted, do not remove the cover or any part of the casing while the oscilloscope is in operation. Doing so would expose circuits and components, reduce the overvoltage level of the measurement, and may cause personal injury, fire, or damage to the oscilloscope.
Anything that is not designed to be placed on the external interface of this product should not be placed there. Doing so may cause an internal short circuit and/or electric shock, fire, or injury to the oscilloscope.
Use or placement of the oscilloscope in an environment exceeding IP51 may damage the oscilloscope. This oscilloscope must be used in a dry environment to avoid risk of electric shock.
Similarly, the oscilloscope should not be used when condensation has formed or may form inside or on the surface. For example, when the oscilloscope moves from a cold environment to a warm environment, the penetration of water increases the risk of electric shock and damage to the scope.

Safe Transportation

For short-distance transportation under supervision, please ensure that the oscilloscope is turned off before putting it in the carrying case. For long-distance transportation under unsupervised conditions, detach the battery first.
The fabric handle on the oscilloscope is meant to be held in your hand. This handle is not designed for use as a focus point to be fixed on transportation equipment, such as cranes, forklifts, trucks, etc. It is the user's responsibility to fasten the oscilloscope firmly to the transportation or lifting tool.
If you use the oscilloscope in a vehicle, the driver is responsible for driving the vehicle safely and correctly. The manufacturer is not responsible for any accidents or collisions. Do not use the oscilloscope in a moving vehicle, to avoid distracting the driver. Use the oscilloscope appropriately in the vehicle to prevent injury or other damage in the event of an accident.

Battery Safety

This product contains a rechargeable lithium battery pack. If misused, there is a risk of explosion, fire and/or serious personal injury, and even death in some cases.

The battery cannot be disassembled or crushed.
The battery cannot be exposed to high temperature or fire, and it must not be stored in direct sunlight. Keep the battery clean and dry. Use a dry, clean cloth to clean contaminated connectors.
The battery must not be short-circuited. Batteries should not be stored in an environment that can easily cause a short circuit, such as boxes and drawers containing metal debris. The battery cannot be taken out of its original packaging before use.
Batteries should not be exposed to any excessive mechanical shock.
If the battery leaks, do not let the liquid contact the skin or eyes. If contact occurs, wash the contact area with plenty of water and seek medical assistance.
Only the power adapter provided by the manufacturer should be used for charging. Use of a mismatched power supply may cause a fire or cause personal injury or death.
The battery should be charged in a well-ventilated room. During the charging process, the oscilloscope should not be covered by objects (such as blankets, towels, clothes), which will stifle heat dissipation and cause a risk of fire.
Used batteries and battery packs must be recycled and separated from residual waste. Batteries contain hazardous waste, and local regulations on waste disposal and recycling must be followed.

Waste Disposal

Used oscilloscope batteries must not be disposed of together with unsorted municipal waste. They must be collected separately and placed in designated recycling points.
If the scope and its components are mechanically and/or thermally processed in a way beyond the intended use, hazardous substances may be released. Therefore, this scope can only be disassembled by professional personnel. Improper disassembly may endanger your health. Local waste disposal regulations must be followed.

Safety Compliance

This oscilloscope complies with the following safety standards:

U.S. nationally recognized testing laboratory listing

- UL 61010-1:2012/R:2018-11. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 1: General Requirements.
- UL 61010-2-030:2018. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part2-030: Particular requirements for testing and measuring circuits.
- UL 61010-2-033:2020. Safety Requirements for Electrical Equipment for Measurement, Control and Laboratory Use – Part 2-033: Particular Requirements for Hand-Held Multimeters for Domestic and Professional Use, Capable of Measuring Mains Voltage.

Canadian certification

- CAN/CSA-C22.2 No. 61010-1:2012/A1:2018-11. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 1: General Requirements.
- CAN/CSA-C22.2 No. 61010-2-030:2018. Safety Requirements for Electrical Equipment for Measurement, Control, and Laboratory Use – Part 2-030: Particular requirements for testing and measuring circuits.
- CSA C22.2 No. 61010-2-033-2020. Safety Requirements For Electrical Equipment For Measurement, Control And Laboratory Use — Part 2-033: Particular Requirements For Hand-Held Multimeters For Domestic And Professional Use, Capable Of Measuring Mains Voltage

General Care and Cleaning

Care

Do not store or leave the instrument in direct sunshine for extended periods of time.

To avoid damages to the instrument or probes, please do not expose them to fog, liquid, or solvents.

Cleaning

Please perform the following steps to clean the instrument and probes regularly in accordance with its operating conditions.

Disconnect the instrument from all power sources and then clean with a soft wet cloth.
Clean the loose dust on the outside of the instrument and probe with a soft cloth. When cleaning the LCD screen, take care to avoid scratching it.

To avoid damage to the surface of the instrument and probe, please do not use any corrosive liquid or chemical cleansers.

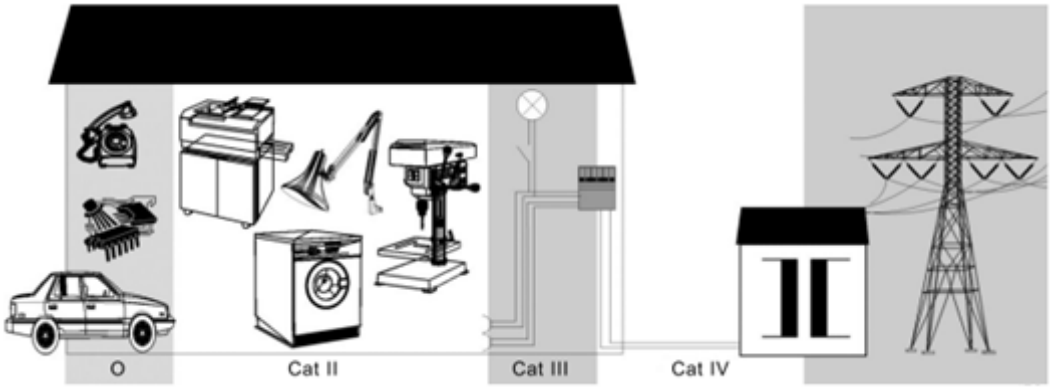
Make sure that the instrument is completely dry before restarting it to avoid potential short circuits or personal injury.

Measurement Category

WARNING
Measurement overvoltage can lead to electric shock to the operator.

IEC61010-2-030 defines the measurement category to rate the ability of measuring instruments to withstand short-term transient overvoltage outside of the working voltage. This product and its accessories can only be used in the environment of the nominal measurement category.

- 0: An instrument with no rated measurement category is used to measure circuits not directly connected to the mains, such as a circuit board powered by a battery or a secondary circuit with special protection. This measurement category is also called CAT Ⅰ.
- CAT II：
  For measurements of branch circuits that power plug loads, such as appliances, portable tools, and other equipment that plugs into an outlet.
- CAT III:
  For measurements on equipment permanently installed in the building, such as motor control centers, motors with permanent connections to the electrical distribution system, breaker panels, and junction boxes.
- CAT IV:
  For measurements performed at the service entrance or on powerlines (overhead or buried). This includes measurements before the service equipment overcurrent device.



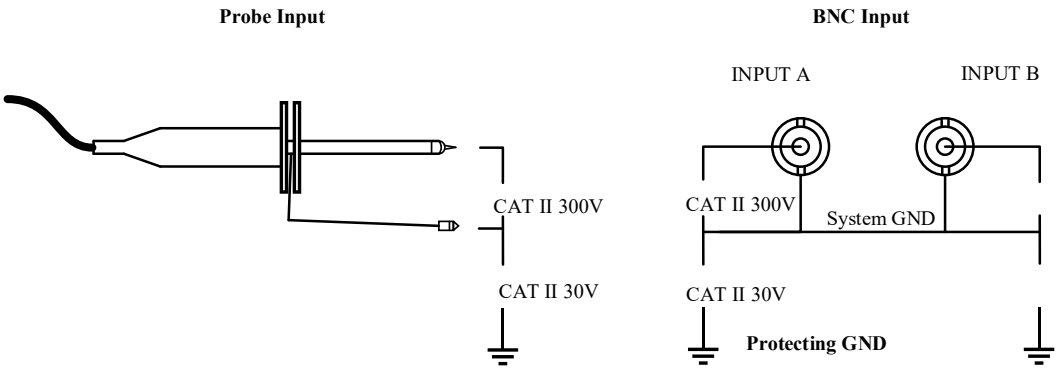
Measurement Categories

The oscilloscope can measure in each category below within the listed voltage ranges. For the combination of a probe assembly and an accessory, use the lower of the measurement categories' limits.

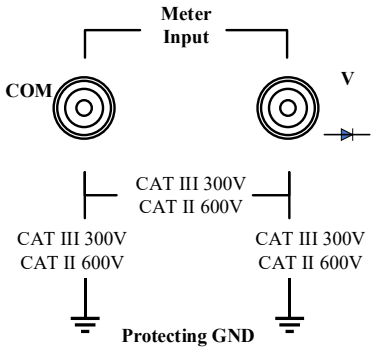
AEGIS-OSC-9200	Max Input Voltage
Scope（BNC input）	CAT II 300Vrms Between BNC Signal & Protective Earth CAT II 30Vrms Between BNC GND & Protective Earth CAT II 300Vrms Between BNC Signal & BNC GND
Scope（Probe input）	CAT II 300Vrms Between Signal & Protective Earth CAT II 30Vrms Between Probe GND & Protective Earth CAT II 300Vrms Between Probe Signal & Probe GND
Multimeter	CAT III 300Vrms, CATII 600Vrms.
SCD10A[1]	CAT III 60Vrms.
SCD600MA[1]	CAT III 60Vrms.

Note: SCD10A is a 10A current adapter for the meter using to measure a current up to 10A, SCD600MA is a 600mA current adapter for the meter using to measure a current up to 600mA. Refer to the DCI/ACI chapter for more information.

Note: Although measurements within a permanently installed motor would be Cat III, shaft voltage measurement is considered Cat II, as neither the motor shaft or frame is directly energized under normal (non-fault) operation.



Measurement Categories for Oscilloscope



Measurement Categories for Multimeter

WARNING

Electrical Shock Hazard!

The reference grounds of CH1 and CH2 are connected together. Connecting one to a voltage higher than 30Vrms and touching the other may results in personal injury.

Working Environment

General requirements

Environment

This product is intended for indoor use and should be operated in a clean, dry environment.
It can be stored in an environment with a waterproof/dustproof rating better than IP51.

Temperature

Operating: 32˚ to 105˚F (0˚ - 40˚ C)
Non-operating: -5˚ to 140˚F (-20˚ - 60˚ C)

Note: Direct sunlight, radiators, and other heat sources should be taken into account when assessing the ambient temperature.

Humidity

Operating: 85% RH, 105˚F (40˚ C) , 24 hours
Non-operating: 85% RH, 150˚F (65˚ C), 24 hours

Altitude

Operating: less than 1.2 mi/2 km
Non-operating: less than 3 mi/5 km

Installation (overvoltage) Category

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

WARNING
Overvoltage while operating on line power (from lightning, for instance) creates the risk of electric shock for the user.

Installation (overvoltage) Category Definitions

Installation (overvoltage) category I refers to signal level. This applies to circuits with measures in place to limit the transient voltage to the corresponding low level.

Installation (overvoltage) category II refers to the local power distribution level. This applies to equipment running on "wall power".

Degree of Pollution

The oscilloscope may be operated in environments up to Pollution Degree II.

Note: Pollution Degree II refers to a dry working environment and non-conductive pollution. Occasional temporary conductivity caused by condensation is expected.

IP Rating

IP51 (as defined in IEC 60529)

Ventilation Requirement

It is best to allow free air circulation while the scope is in use. When using it in a benchtop or rack setting, provide at least 10 cm clearance beside, above and behind the instrument for adequate ventilation.

WARNING
Inadequate ventilation may cause temperature increase which could damage the oscilloscope. So please keep it well ventilated during operation.

AC Power Requirement

The power adapter operates with a single-phase, 100 to 240 Vrms (+/-10%) AC power at 50/60 Hz (+/-5%).

No manual voltage selection is required.

Depending on the type and number of options and accessories (probes, PC port plug-in, charging etc.), the scope can consume up to 35 W while on AC power.

Note: The power adapter automatically adapts to the AC line input within the following ranges:

Voltage Range:	90 - 264 Vrms
Frequency Range:	47 - 63 Hz

Please use the power cord and adapter provided by the manufacturer. Use of other adapters/chargers may cause damage, fire, or injury.

WARNING
Electrical Shock Hazard!
Use of an adapter or power supply not specified by the manufacturer may cause personal injury.

To turn the oscilloscope completely off, unplug the power cord from the AC socket and remove the battery.

The battery should be removed from the scope when it is not to be used for an extended time.

Maintenance and Service

Only authorized and specially trained personnel can open the oscilloscope. When performing maintenance on the oscilloscope, you must remove the power adapter and make sure that the instrument is turned off. Otherwise an internal short circuit may damage the oscilloscope or shock the user.

Adjustment, replacement of parts, maintenance and repair can only be performed by operators authorized by the manufacturer. Safety-related parts can only be replaced with original parts. Safety tests must be carried out after replacing parts.

Table of Contents

Quick Start

Contents:

Appearance and Dimensions (mm)

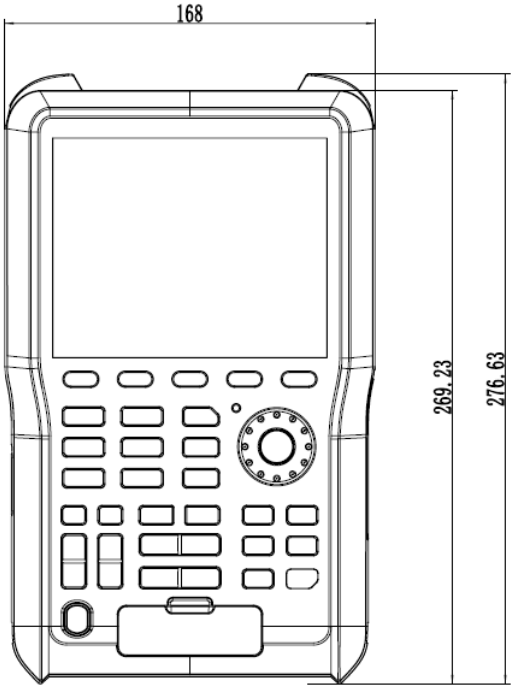


Figure 1 Front View

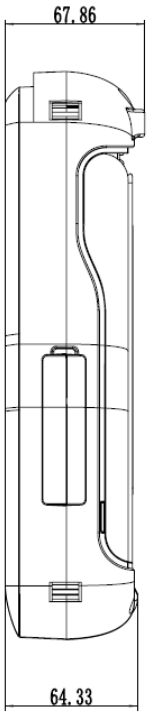


Figure 2 Side view

Preparation for Use

Adjust the Supporting Leg

Adjust the supporting leg and lean the oscilloscope back on it for stability.

Figure 3 Adjust the Supporting Leg

Battery Installation

When the oscilloscope leaves the factory, the battery is disconnected. Connect the battery as follows:

1. Remove the three screws on the battery cover with a screwdriver, as shown in Figure 4.
2. Remove the battery cover, as shown in Figure 5.
3. Put the battery into the battery slot and close the battery cover, as shown in Figure 6.
4. Tighten the screws with the screwdriver, and then turn on the oscilloscope to confirm proper connection.

Figure 4

Figure 5

Figure 6

Important tips:

- Pay attention to the direction of the battery cover, with the serial number at the bottom.
- The battery compartment is designed NOT to allow the battery to be inserted reversed. Insert the battery gently; if it does not slide in easily, it may be backwards.
- If the scope fails to power on after inserting the battery, it may be that the battery has run out. Please charge it.
- If the oscilloscope sits in storage, the battery will need to be recharged every three months.

Connect the Power Supply

The power supply accepts input of 100-240 V, 50/60Hz. Use the power cord provided to connect the oscilloscope to the power source.

Figure 7 Connect to Power Supply

Power-on Inspection

Press the power button at the lower left of the front panel to start the oscilloscope. During the start-up process, the scope performs a series of self-tests and you can hear the sound of relays switching. After the self-test is finished, the welcome screen will be displayed.

Connect the Probe

Connect the Probe:

1. Connect the BNC terminal of the probe to Channel 1's BNC connector.
2. Connect the probe tip to the circuit point to be tested and connect the ground alligator clip of the probe to the circuit ground terminal.

Function Inspection

1. Press the **Default** button on the front panel to restore the oscilloscope to its default configuration.
2. Connect the ground clip of the probe to the "Ground Terminal" under the probe compensation signal output terminal (1 in Figure 10).
3. Touch the probe tip to the "Compensation Signal Output Terminal".
4. Press the **Auto Setup** button.
5. Observe the waveform on the display. If the probe is properly compensated, the display will show a square waveform as shown in Figure 8:

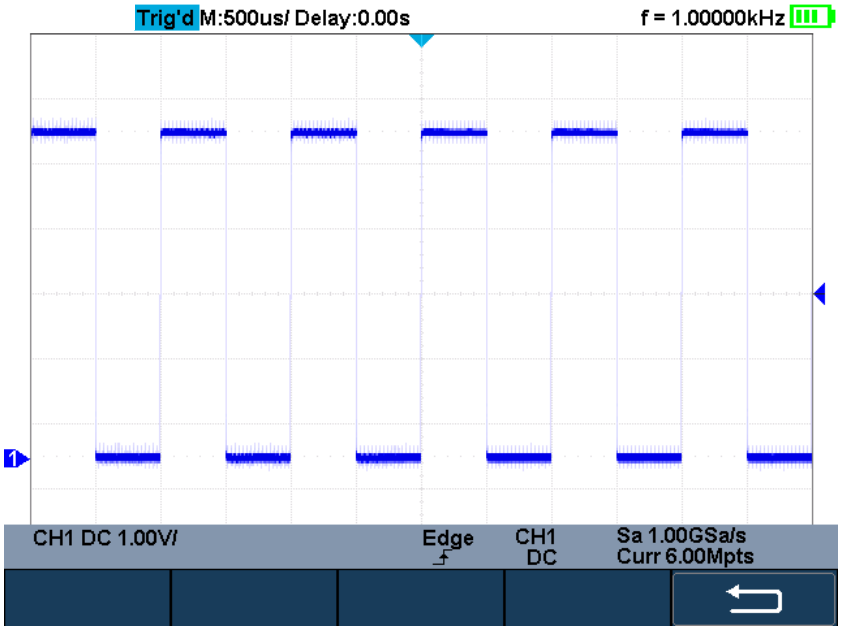


Figure 8 Function Inspection

6. Use the same method to test Channel 2. If the waveforms do not have square corners as shown above, adjust the probe compensation as described in the next section.

WARNING

To avoid electric shock, ensure that the insulated wire of the probe is in good condition, and do not touch the metallic part of the probe when the probe is connected to high voltage source

Probe Compensation

When the probes are used for the first time, you should adjust (compensate) the probes to match the input channels of the oscilloscope. Non-compensated or poorly compensated probes will cause measurement inaccuracy or error. The probes can be compensated as follows.

1. Set the switch to 10X on the probe.
2. Perform steps 1, 2, 3 and 4 of "**Function Inspection**" in the previous section.
3. Check the waveforms displayed and compare them with the following:

Under Compensated | Perfectly Compensated | Over Compensated

4. Use a nonmetallic screwdriver to adjust the low-frequency compensation adjustment hole on the probe until the waveform is square ("Perfectly compensated" in the figure above).

Multimeter Leads

Voltage, resistance, and other properties are measured differently than current. To measure anything but current, plug the multimeter leads directly into the jacks at the bottom of the front of the instrument. Current measurements require an adapter. The leads plug into the adapter, which then plugs into the jacks on the oscilloscope. Attempting to measure current without the adapter may damage the multimeter circuitry.

Front Panel

Figure 9 Scope Front Panel

No.	Description	No.	Description
1	Power Button	7	Universal Knob
2	Vertical Control	8	Trigger Controls
3	Dual Function Buttons	9	Horizontal Control
4	Menu Selection Keys (F keys)	10	Single Function Buttons
5	LCD Display	11	Multimeter Input
6	Analog Channel Input		

Side Panels

Figure 10 Scope Side Panel Overview

1. **Probe Compensation/Ground Terminal**
   Supplies a 0-5 V, 1 kHz square wave for compensating the probes.

2. **USB Port**
   Plug flash drives/USB storage devices in here for data transfer.

3. **USB Device**
   The oscilloscope support SCPI remote control commands. The user can control the oscilloscope through this interface.

4. **Adapter Power Input**

Rear Panel

Figure 11 Scope Rear Panel Overview

1. **Battery Cover**

2. **Supporting Leg**

   Adjust the supporting leg properly to use them as stands to tilt the oscilloscope upwards for stable placement of the oscilloscope as well as better operation and observation.

3. **Handle**

Front Panel Functions

Horizontal Control

Horizontal Position Buttons (< >): adjust horizontal position. Press the button to change the horizontal offset. Waveforms of both channels will move horizontally with the trigger point. Press the **Shift** + **Horizontal Position** to quickly reset the horizontal delay to Zero.

Horizontal Scale Button (s, ns): adjust the horizontal time base. Press **ns** to reduce the time base and press **s** to increase the time base. As the change is made, all waveforms will be expanded or compressed, respectively. The time base message at the upper left of the screen shows the current scale.

Vertical Control

: Select input channel. The two channels are marked by different colors which are also used to mark both the corresponding waveforms on the screen and the channel input connectors.

Vertical Position Buttons (▲ ▼): adjust the vertical position of the current channel waveform. Each arrow moves the waveform in its direction. The position message at the lower left will be updated with the new offset value. Press **Shift** + **Vertical Position** to quickly reset the vertical position to zero.

Vertical Scale Button (V, mV): adjust the vertical scale of the current channel. Press **mV** to decrease the scale and press **V** to increase. During the modification, the amplitude of the waveform will increase or decrease, respectively, and the scale information in the channel setup box will change accordingly.

Trigger

: Press **Trigger Setup** to enter the TRIGGER function menu. The oscilloscope provides numerous advanced trigger functions.

: Press **Trigger Level** and then turn the **Universal Knob** to adjust the trigger level. The trigger level line will visibly move up or down. Press the **Universal Knob** to quickly reset the trigger level to the center of the waveform.

Run Control

Press **Run/Stop** to set the acquisition state to Run (collect and display signal) or Stop (pause collection).
In the RUN state, the key is illuminated in yellow.
In the STOP state, the key is illuminated in red.

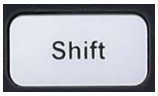
Press **Auto Setup** to enable the waveform auto setting function. The oscilloscope will automatically adjust the vertical scale, horizontal time base and trigger mode according to the input signal to realize optimum waveform display.

Press **Default** to reset the oscilloscope to the default setup.

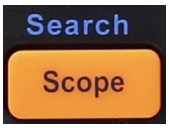
Universal Knob

1. **Adjust the waveform intensity**
   Press **Display/Persist**, then press the **Next Page** F key to go to the second page of the DISPLAY function menu. Press the **Intensity** F key and then turn the **Universal Knob** to adjust the waveform intensity: Turn clockwise to increase the intensity and counterclockwise to decrease it.
2. **Select the desired submenu**
   In menu operation, press any menu F key and turn the **Universal Knob** to select the desired submenu under the menu. Push down the knob to confirm the current submenu.
3. **Modify parameters**
   After having chosen a parameter, turn the **Universal Knob** to modify the value. Turn clockwise to increase it and counterclockwise to decrease it. The universal knob can also be used to adjust scale and offset of MATH and REF.
4. **Choose file or directory or input filename**
   In the file system, turn the **Universal Knob** to select the desired file or directory. When entering a file name, turn the **Universal Knob** to select the desired character and then push the knob to confirm.

Dual Function Buttons

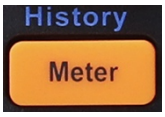


Function switching button for dual function buttons



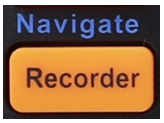
Press **Scope** to enter scope mode.

Press **Shift** and this button to enter the SEARCH function menu.



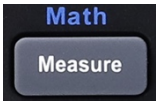
Press **Meter** to enter meter mode.

Press **Shift** and then **Meter** to enter history mode. The history can store at most 80,000 frames of waveforms.



Press **Recorder** to enter recorder mode.

Press **Shift** and then **Recorder** to turn off or turn on the navigate function.



Press **Measure** to enable/disable measurements.

Press **Shift** and then **Measure** to enable/disable MATH mode.

Press **Cursors** to enter the CURSOR function menu. The oscilloscope provides manual and track cursor mode.

Press **Shift** and then **Cursors** to enter the SAVE/RECALL menu.

Press **Print** to save a screenshot to an external storage device. Supported image formats include .bmp, .jpg, and .png.

Press **Shift** and **Print** to enter the DECODE function menu. The oscilloscope supports I2C, SPI, UART, CAN and LIN serial bus decode.

Press **Menu On/Off** to display or hide the current menu.

Press **Shift** and then **Menu On/Off** to enter the REF function menu.

Clear Sweeps clears the data or display in multiple sweeps, including display persistence, measurement statistics, average sweeps and Pass/Fail statistics.

Press **Shift** and **Clear Sweeps** to enter zoom mode.

Other Buttons

Press **Acquire** to enter the ACQUIRE function menu, to set the acquisition mode, memory depth, wave interpolation, and so on.

Press **Display/Persist** to enter the DISPLAY menu. The second press turns on Persist - press again to turn off Persist.

Press **Utility** to enter the UTILITY function menu, to see system status, do self-calibration, set the sound and language, and so on.

Help

The oscilloscope has a help function that supplies multi-language help information.

You can access the help function by pressing any button for 2 seconds and a help window will explain the function. Also, all of the submenus include help information.

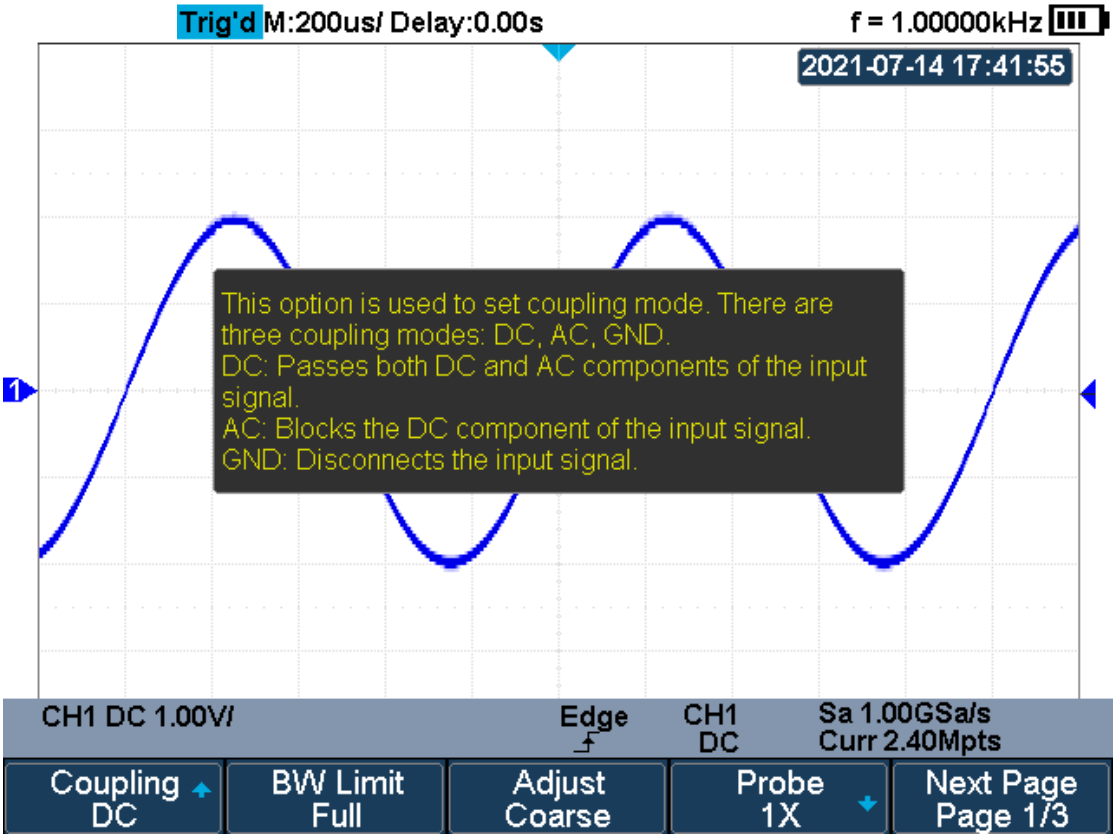


Figure 12 Help Message

User Interface

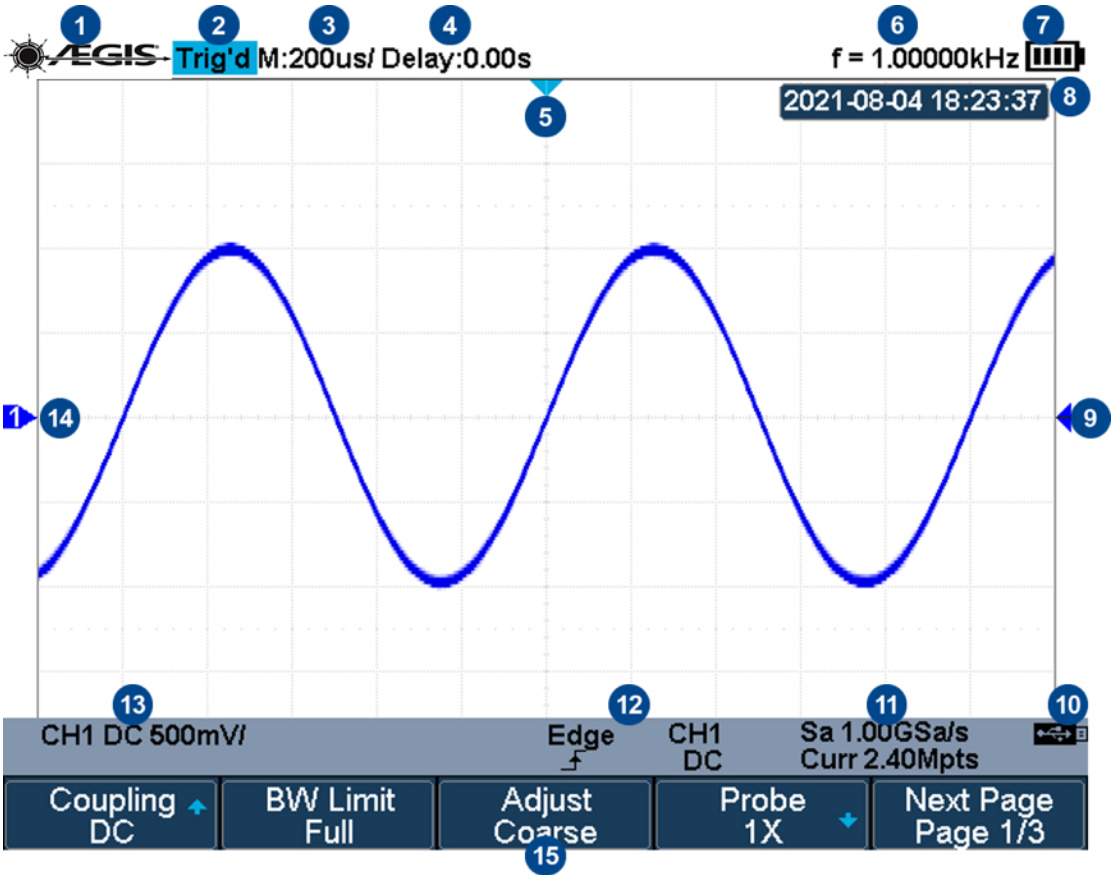


Figure 13 User Interface

1. **Logo**

   AEGIS® is a registered trademark of **Electro Static Technology, an ITW Company**

2. **Acquisition Status**

   Possible trigger status values: Arm, Ready, Trig'd, Auto, Stop, FStop, Roll.

3. **Horizontal Time Base**

   - Represent the time per grid on the horizontal axis on the screen.
   - Use the **Horizontal Scale** button to adjust the time scale. The available range is from 1.0 ns to 100 s.

4. **Horizontal Position (Delay)**

   Turn the **Horizontal Position** button to adjust the time offset. Press **Shift** and the **Horizontal Position** button to set the value to 0 automatically.

5. **Trigger Delay Label**

   Indicates the trigger delay on the waveform.

6. **Frequency Counter**
   Display the frequency value of the trigger channel.

7. **Battery Level**

8. **Date/Time**

9. **Trigger Level Label**
   Displays the position of trigger level. Each channel has its own color-coded trigger level label. It can move from +4.5div to -4.5div from the screen center.

10. **I/O status**
    Indicates that a device is connected to the USB port.

11. **Sampling Rate/ Memory Depth**
    Displays the current sampling rate and memory depth. Sa is the sampling rate and Curr is the current memory depth.

12. **Trigger Setup**
    Edge Trigger Type: displays the current trigger type. The trigger type names display by the abbreviation when the name is too long to display.
    CH1 Trigger source: displays the channel the trigger is acting on.
    Trigger condition: display the current trigger condition.
    DC Trigger coupling: display the current trigger coupling. Available trigger coupling modes: DC, AC, HF Reject, LF Reject.

13. **Channel Setup**
    DC Channel coupling: display the current channel coupling of the channel. Available channel coupling modes: DC, AC, and GND.
    1.00V/ Vertical Scale: displays the current voltage scale of the channel. Press the **Vertical Scale** button to adjust the value.

14. **Channel Label/Waveform**
    Different channels are marked by different colors. Typically Channel 1 is in blue and Channel 2 in red.

15. **Menu**
    Display the corresponding function menu of the selected button. Press the corresponding F key to set the oscilloscope.

Vertical Control

Contents:

Enabling the Channel

The oscilloscope includes 2 analog input channels, and each channel shares the same vertical control system. Since the vertical system setting methods of each channel are the same, this chapter uses CH1 as an example to show how to control the vertical settings.

Connect the signal to the CH1 channel connector, and then press **CH1** in the vertical control area (VERTICAL) on the front panel to enable CH1.

The channel setting menu and the channel information bar are displayed at the bottom of the screen.

After the channel is turned on, adjust the parameters such as vertical scale (V/div), horizontal time base (s/div), and the trigger mode according to the input signal to make the waveform display clearly.

Note: to turn off the channel, press the channel button twice.

Adjusting the Scale

The vertical scale can be adjusted in **Coarse** or **Fine** mode.

- **Coarse** adjustment (take counterclockwise as an example): set the vertical scale in 1-2-5 step namely 2 mV/div, 5 mV/div, 10 mV/div …10 V/div.
  Fine adjustment: further adjust the vertical scale within a relatively smaller range to improve vertical resolution. For example: 2 V/div, 1.98V/div, 1.96V/div, 1.94 V/div …1 V/div.
- If the amplitude of the input waveform is too large at the current scale, but would be too small if the next scale up was used, fine adjustment can be used to improve the amplitude.

Press **CH1** on the front panel, then press the **Adjust** F key to choose the desired mode. Press **Vertical Scale** to adjust the vertical scale (**V** to increase the scale and **mV** to reduce).

The scale information in the channel information bar at the bottom of the screen will change during the adjustment process. The adjustable range of the vertical scale is related to the probe attenuation ratio. By default, the probe attenuation factor is 1:1 (1X) and the adjustable range of the vertical scale is from 2 mV/div to 100 V/div.

Vertical Position

Press **Vertical Position** to adjust the vertical position of the channel waveform. Press up to move the waveform up on the screen, and press down to move the waveform down. Press **Shift** and **Vertical Position** to set the vertical position of the channel waveform to zero.

The table below shows the range of vertical positions according to the volt scale.

Volt Scale	Range of Vertical Position
2 mV/div ~ 296 mV/div	±5 V
302 mV/div ~ 7.5 V/div	±80 V
7.6 V/div ~ 100 V/div	±400 V

Coupling

Set the coupling mode to filter out undesired signals. For example, the signal under test is a square waveform with DC offset.

- When the coupling mode is set to **DC**: the DC and AC components of the signal under test can both pass, and both will be shown.
- When the coupling mode is set to **AC**: the DC components of the signal under test are blocked. Any DC offset present will not be displayed.
- When the coupling mode is set to **GND**: the DC and AC components of the signal under test are both blocked.

Press **CH1** on the front panel, then press the **Coupling** F key and turn the **Universal Knob** to select the desired coupling mode. The default setup is **DC**.

The current coupling mode is displayed in the channel information bar at the bottom of the screen. You can also press the **Coupling** F key repeatedly to switch the coupling mode.

Bandwidth Limit

You can set the bandwidth limit to reduce display noise. For example, if the signal under test is a pulse with high frequency oscillation.

- When the bandwidth limit is set to **Full**, the high frequency components of the signal under test can pass through the channel.
- When the bandwidth limit is set to **20M**, the components over 20 MHz will be attenuated.

Press **CH1** on the front panel; then press the **BW Limit** F key to select **Full** or **20M**. The default setup is **Full**. When the bandwidth limit is enabled, the character **B** will be displayed in the channel information bar at the bottom of the screen.

Probe

Set the probe attenuation factor to match the type of probe that you are using to ensure correct vertical readouts. For most purposes, 10X (10:1) is a good ratio.

Press **CH1** on the front panel; then press the **Probe** F key, turn the **Universal Knob** to select the desired value, and push the knob to confirm. The default setup is **1X**.

You can also repeatedly press the **Probe** F key to switch the probe attenuation factor.

The table shows the probe attenuation factor

Menu	Attenuation Factor
0.1X	0.1 : 1
0.2X	0.2 : 1
0.5X	0.5 : 1
1X	1 : 1
2X	2 : 1
…	…
5000X	5000 : 1
10000X	10000 : 1
Custom	1000000:1~0.000001:1

You can also customize the probe attenuation factor. Press the **Probe** F key, select **Custom**, and then press the **Custom** F key to select **Custom** or **Custom Fine**. First, select **Custom** and turn the **Universal Knob** to achieve rough adjustment, and then select **Custom Fine** and turn the **Universal Knob** to achieve fine adjustment.

Unit

Select the amplitude display unit for the current channel. The available units are **V** and **A**. When the unit is changed, the unit displayed in the channel information bar will change accordingly.

1. Press **CH1** on the front panel to enter the CH1 function menu.
2. Press the **Next Page** F key to enter the second page of the CH1 function menu.
3. Press the **Unit** F key to select the desired unit **V** or **A**.

The default setting is **V.**

Deskew

The valid range of each channel is ±100ns.

1. Press **CH1** button on the front panel to enter the CH1 function menu.
2. Press the **Next Page** F key to enter the second page of the CH1 function menu.
3. Press the **Deskew** F key. Then turn the **Universal Knob** to change the deskew setting.

Invert

When **Invert** is set to **On**, the voltage values of the displayed waveform are inverted (multiplied by -1). Invert affects how a channel is displayed while keeping the trigger settings.

Inverting a channel also changes the result of any math function selected and measure function.

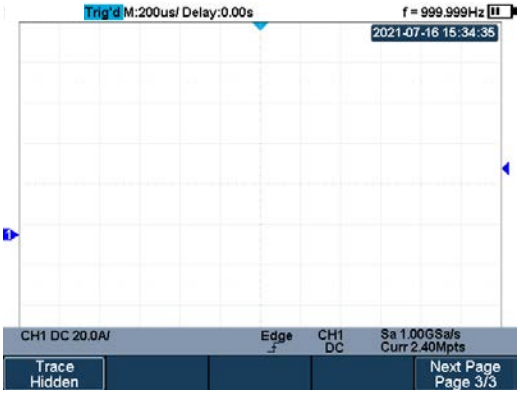
1. Press **CH1** button on the front panel to enter the CH1 function menu.
2. Press the **Next Page** F key to enter the second page of the CH1 function menu.
3. Press the **Invert** F key to turn on or off the invert display.

Trace Visible/Hidden

Set whether to hide the current channel waveform

Pressing **CH1** button repeatedly will toggle the trace (waveform) on and off.

Trace Visible



Trace Hidden

Horizontal Control

Contents:

Horizontal Scale

Press **Horizontal Scale** on the front panel to adjust the horizontal time base (seconds per division). Press **ns** to decrease the horizontal time scale and **s** to increase it.

The current time base is displayed in the upper left corner. The oscilloscope's horizontal scale ranges from 1ns/div to 100s/div.

The **Horizontal Scale** button works (in the Normal time mode) while acquisitions are running or when they are stopped. When in run mode, adjust the horizontal scale knob to change the sample rate. When stopped, adjust the horizontal scale knob to zoom in the acquired data.

Horizontal Delay

Press **Horizontal Position** on the front panel to set the trigger delay. You can then move the trigger point (solid inverted triangle) horizontally by pressing the arrow keys < and >. Press the **Shift** and **Horizontal Position** button to reset the trigger delay.

When changing the horizontal delay, the delay time in the information bar at the top of the screen changes in real time. The delay time indicates the duration between the time reference point and the trigger point. The available delay range depends on the selected time/div and memory depth.

Roll Mode

Press **Acquire** to enter the acquire menu and press the **Roll Mode** F key to enable roll mode.

In Roll Mode the waveform moves slowly across the screen from right to left. It only operates on time base settings of 50 ms/div or slower.

In Roll Mode there is no trigger. The fixed reference point on the screen is the right edge of the screen and refers to the current moment in time. Events that have occurred are scrolled to the left of the reference point. Since there is no trigger, no pre-trigger information is available.

To exit Roll Mode, press the **Run/Stop** button. To clear the display or restart an acquisition in Roll Mode, press the **Run/Stop** button again.

Using Roll mode on low-frequency waveforms gives a display much like a strip chart recorder. It allows the waveform to roll across the display.

Zoom Mode

Zoom is a horizontally expanded version of the normal display. You can use Zoom to locate and (horizontally) expand part of the normal window for a more detailed (higher-resolution) view.

Press **Shift** and **Clear Sweep** on the front panel to turn on the zoom function, and press them again to turn it off. When Zoom is on, the display window splits in half. The top half shows the main window and the bottom half displays a faster (less time/div) zoom window.

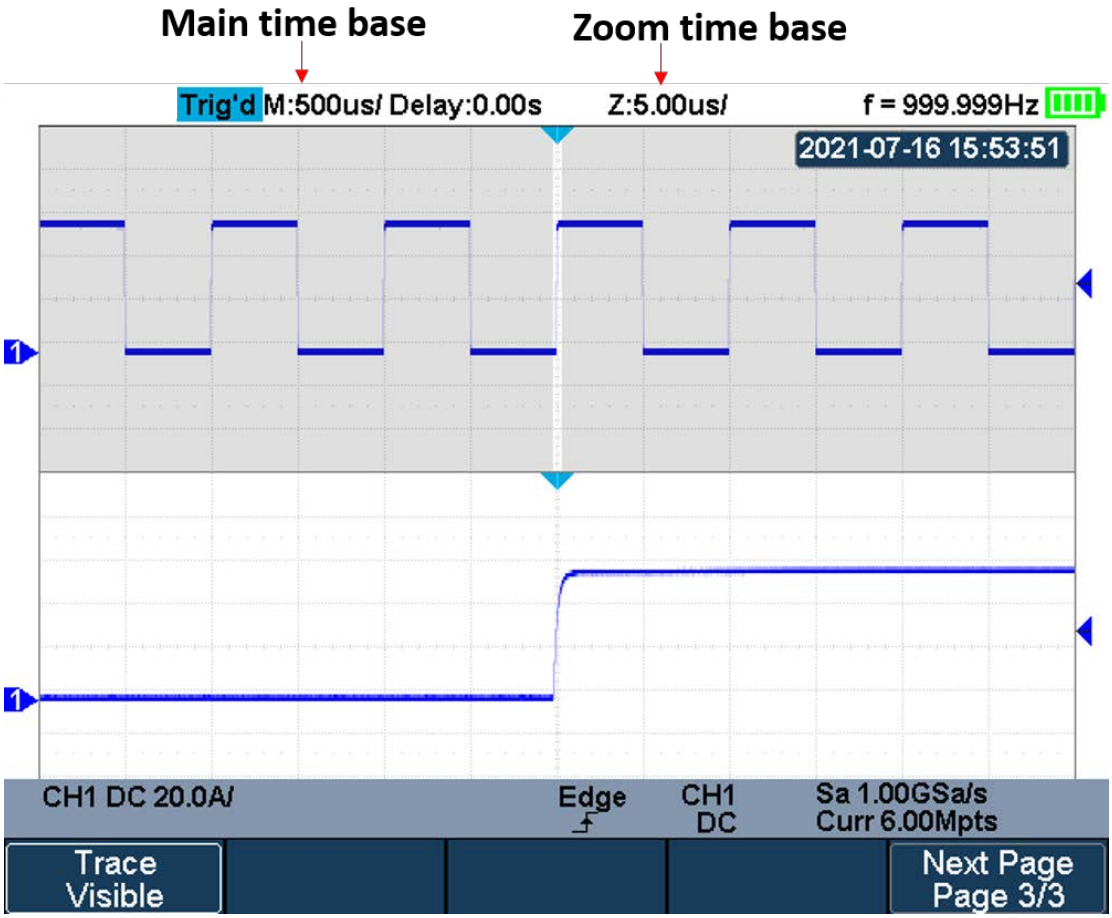


Figure 14 Split Screen Zoom

The zoomed-in area of the normal window is outlined with a border, and the rest of the normal window is shaded. The selected area is zoomed in on at the bottom of the screen.

To change the time base for the Zoom window, press **Horizontal Scale**. The **Horizontal Scale** button controls the size of the border area. The **Horizontal Position** button sets the left-to-right position of the zoom window. The delay value, which is the time displayed relative to the trigger point, is momentarily displayed in the upper-right corner when you press **Horizontal Position**. Negative delay values indicate you're looking at a portion of the waveform before the trigger event, and positive values indicate you're looking at the waveform after the trigger event.

To change the time base of the normal window, turn off Zoom; then press **Horizontal Scale**.

Acquisition

Contents

Overview

To understand the oscilloscope's sampling and acquisition modes, it is helpful to understand sampling theory, sample rate, and oscilloscope bandwidth.

Sampling Theory

The Nyquist sampling theorem states that for a limited bandwidth (band-limited) signal with maximum frequency f_{MAX}, the equally spaced sampling frequency f_S must be greater than twice the maximum frequency f_{MAX}, in order to uniquely reconstruct the signal without aliasing.

$f_{MAX} = f_{S/2}$ = Nyquist frequency (f_N) = folding frequency

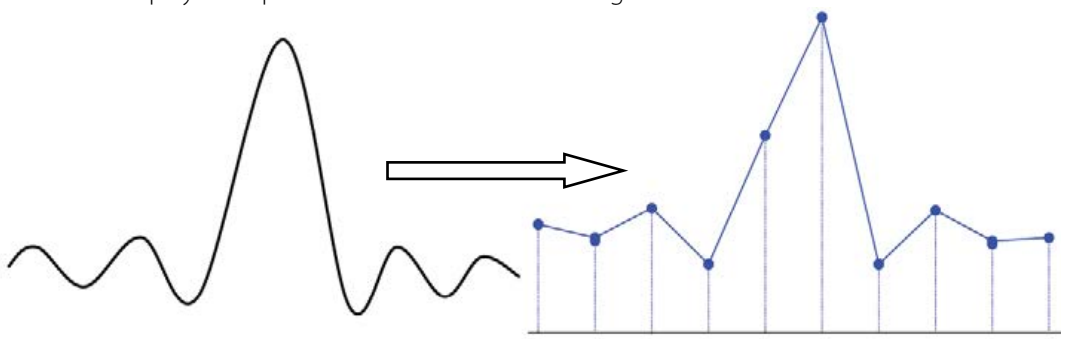
Sample Rate

The maximum sample rate of the oscilloscope is 1GSa/s. The actual sample rate of the oscilloscope is determined by the horizontal scale. Press **Horizontal Scale** to adjust the sample rate.

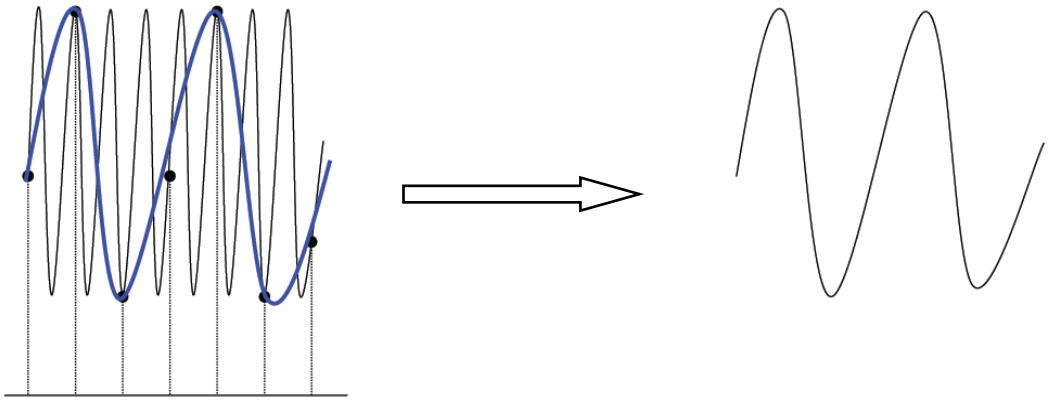
The actual sample rate is displayed in the information area at the bottom right of the screen.

The effects of low sampling rate on the waveform:

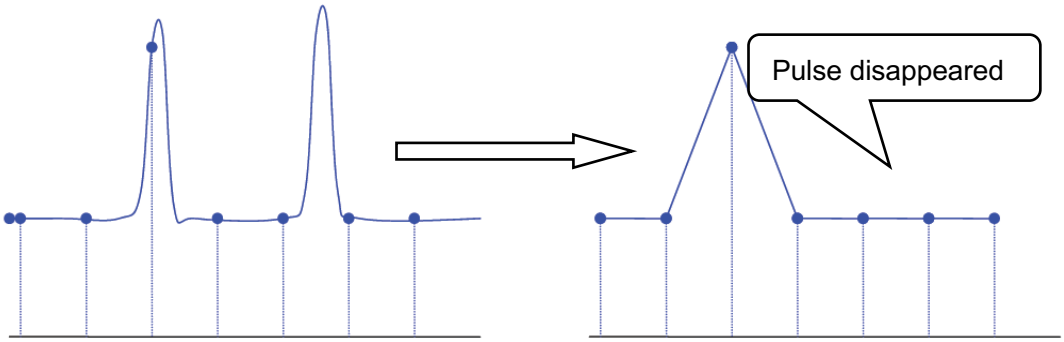
1. **Waveform Distortion:** when the sample rate is too low, some waveform details are lost and the waveform displayed is quite different from the actual signal.



2. **Waveform Confusion (Aliasing):** when the sample rate is lower than twice the actual signal frequency (Nyquist Frequency), the waveform frequency reconstructed from the sample data is lower than the actual signal frequency. The most common aliasing is jitter on a fast-changing edge.



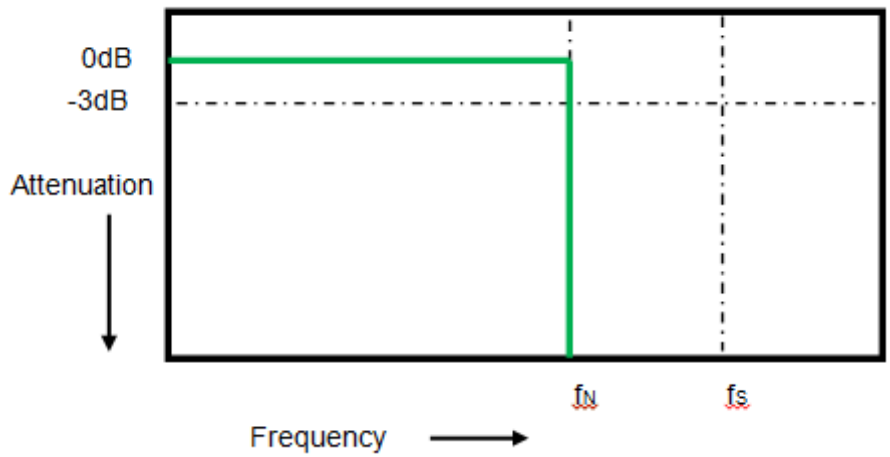
3. **Waveform Leakage:** when the sample rate is too low, the waveform reconstructed from the sample data cannot reflect all the actual signal information.



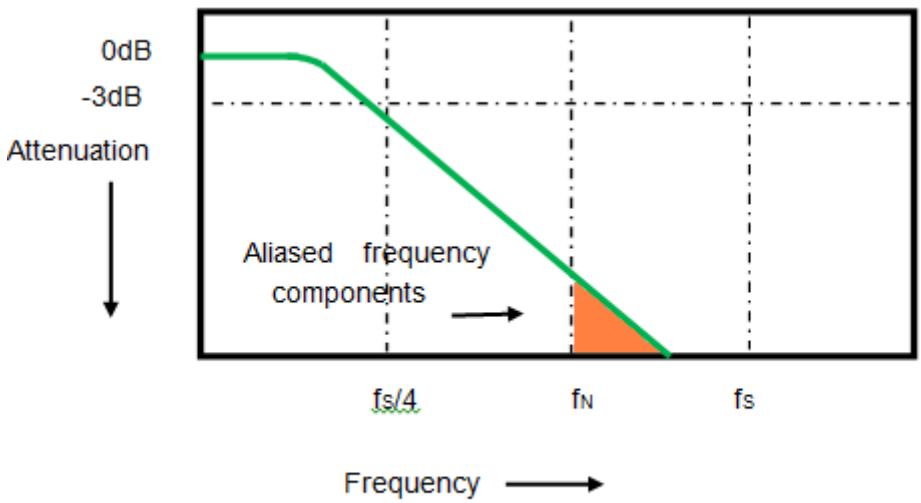
Bandwidth and Sample Rate

An oscilloscope's bandwidth is typically described as the lowest frequency at which the input sinusoidal signal is attenuated by 3 dB, i.e., the scope's reading of a frequency equal to its bandwidth is 30% lower than the true signal strength.

Below the bandwidth frequency, the required sampling rate is $f_S = 2f_{BW}$ according to sampling theory. However, the theory assumes that there are no frequency components above f_{MAX} (f_{BW} in this case) in the system, which means that it requires a system with an ideal brick wall frequency response.



In fact, digital signals have frequency components above the fundamental frequency (ex., a square wave is composed of a sine wave of the fundamental frequency and numerous odd harmonics), and generally, for oscilloscopes with a bandwidth of 500 MHz and below, the system has a Gaussian frequency response.



Limiting oscilloscope bandwidth (f_{BW}) to 1/4 the sample rate ($f_S/4$) reduces frequency components above the Nyquist frequency (f_N).

Therefore, the sampling frequency of the oscilloscope should be four or more times the bandwidth frequency: $f_S = 4fBW$. In this way, there is less aliasing, and the components in the aliasing area are significantly attenuated.

Memory Depth

Memory depth refers to the number of waveform points that the oscilloscope can store in a single trigger sample, and reflects the storage capacity of the sample memory.

Press **Acquire** on the front panel; press the **Mem Depth** F key and then turn the **Universal Knob** to select the desired value and push down the knob to confirm. You can also select the value by pressing the **Mem Depth** F key repeatedly.

The actual memory depth is displayed in the information area at the bottom right of the screen. Since the oscilloscope has two acquisition memories, when only one channel is on, the maximal memory depth is up to 12 Mpts.

The maximum storage depth in single channel mode is twice that in dual channel mode, as shown in the following table:

Single Channel Mode	Dual Channel Mode
12k	6k
120k	60k
1.2M	600k
12M	6M

The relation of memory depth, sample rate, and waveform length fulfills the equation below:
Memory depth = sample rate (Sa/s) × waveform length (s/div × div)

Sampling Mode

This oscilloscope only supports real-time sampling. In this mode, the oscilloscope samples and displays waveform around a trigger event. The maximum real-time sample rate is 1GSa/s.

Press **Run/Stop** to pause the sample, and the oscilloscope will hold the last display. At this point, you can still use the vertical control and horizontal control to pan and zoom.

Interpolation Method

Under real-time sampling, the oscilloscope acquires discrete sampled values of the waveform being displayed. In general, a waveform of discrete dots is difficult to interpret. In order to increase the clarity of the signal, digital oscilloscopes usually use interpolation to display a continuous waveform rather than disconnected sample points.

Interpolation is a processing method to "fill the gaps" between sampled points. This oscilloscope offers two algorithms for interpolation, which can be selected as follows.

Press **Acquire** on the front panel to enter the ACQUIRE Function menu; then press the **Interpolation** F key to select **Sinx/x** or **X**.

- **X** (Figure 16)**:** In the adjacent sample points are directly connected on a straight line. This method is confined to rebuilding on the edge of signals, such as a square wave.

- **Sinx/x** (Figure 17)**:** Connecting the sampling points with curves is more versatile. Sinx interpolation uses mathematical processing to calculate results in the actual sample interval. This method bends the signal waveform, and makes it produce a more realistic regular shape than pure square waves or pulses. When the sampling frequency is 3 to 5 times the bandwidth frequency of the system, the Sinx/x interpolation method is recommended.

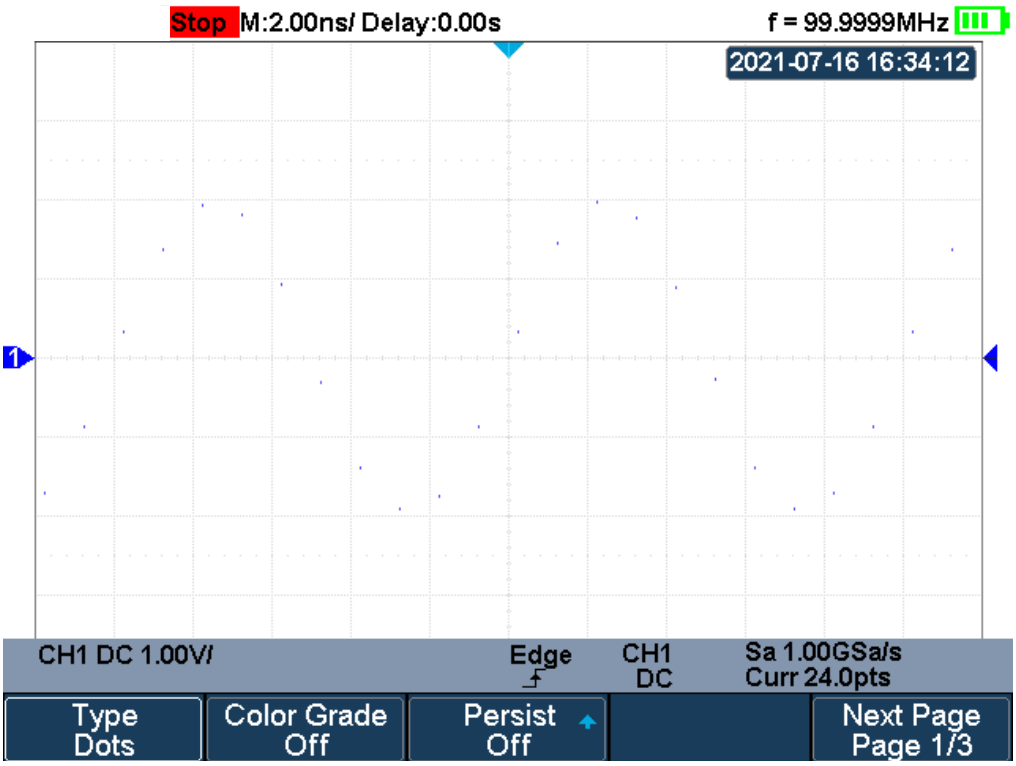


Figure 15 Display Type Set to Dots

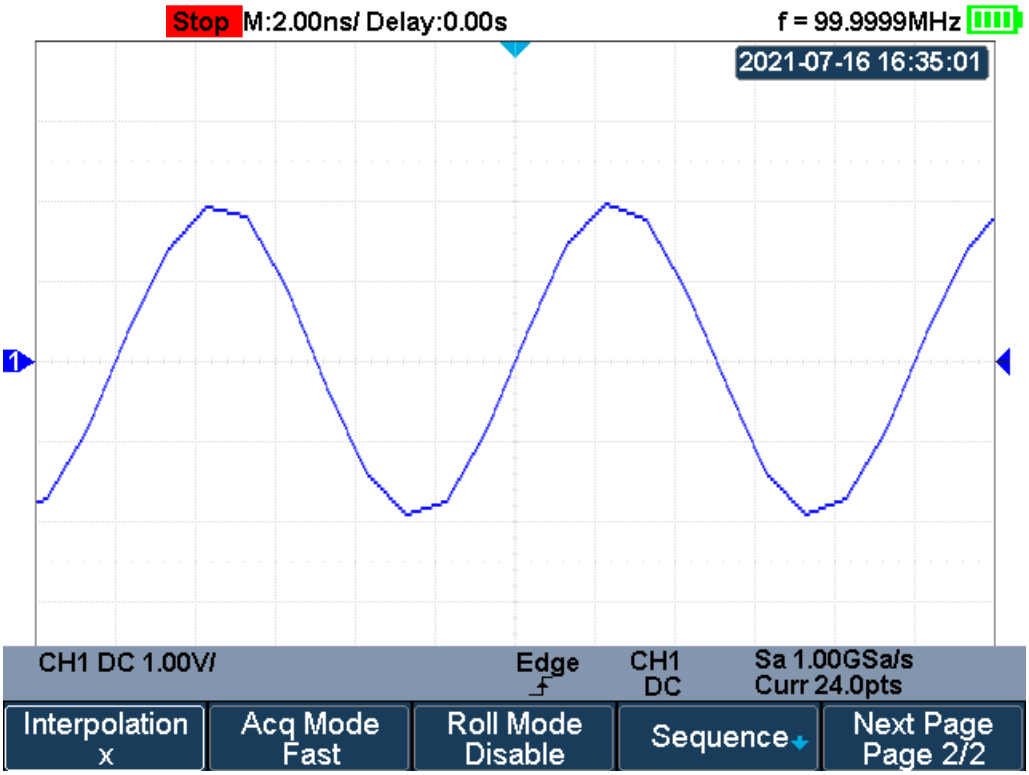


Figure 16 x Interpolation

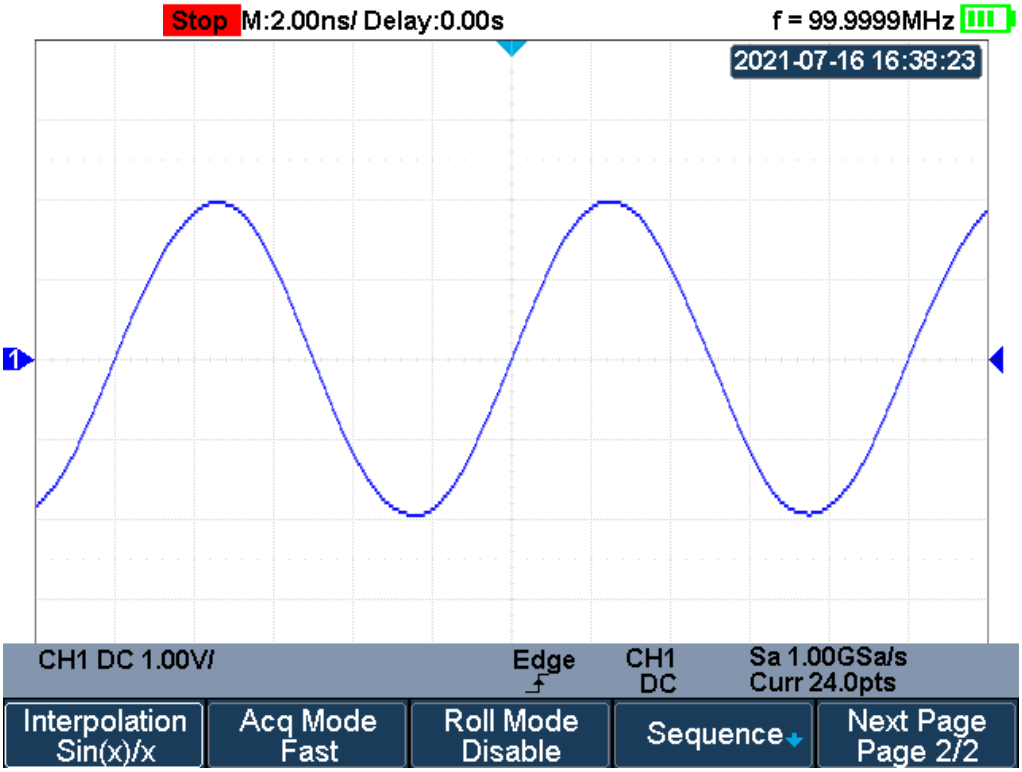


Figure 17 Sinx/x Interpolation

Acquisition Mode

The acquisition mode is used to control how waveform points are generated from sample points. The oscilloscope provides the following acquisition modes: Normal, Peak Detect, Average, and ERES.

1. Press **Acquire** on the front panel to enter the ACQUIRE function menu;
2. Press the **Acquisition** F key; then turn the **Universal Knob** to select the desired acquisition mode and push down the knob to confirm. The default setup is **Normal**.

Normal

In this mode, the oscilloscope samples the signal at equal time intervals to rebuild the waveform. For most waveforms, this mode gives the best result. It is the default acquisition mode.

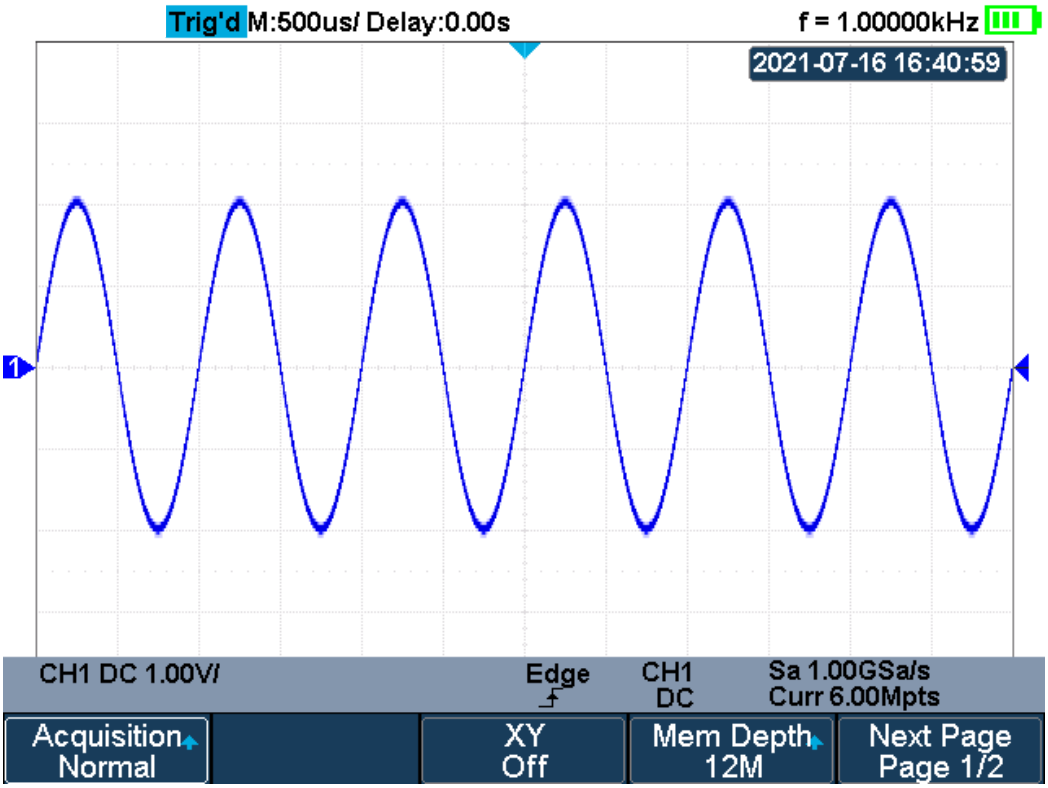


Figure 18 Acquisition System

Peak Detect

In this mode, the oscilloscope acquires the maximum and minimum values of the signal within the sample interval to get the signal's "envelope" or the narrow pulse of the signal that might be lost. In this mode, signal confusion can be prevented but more noise will be displayed.

In this mode, the oscilloscope can display all pulses with pulsewidths at least as wide as the sample period.

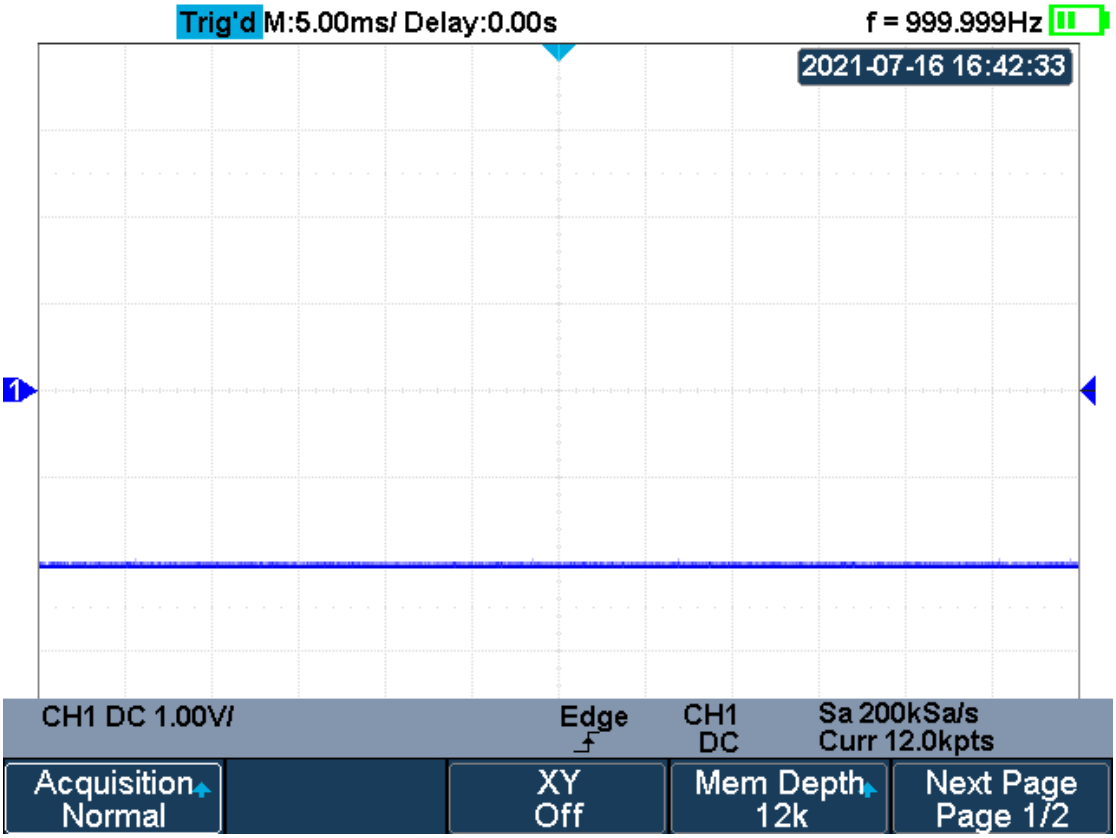


Figure 19 Pulse With 0.1% Duty, Normal Mode

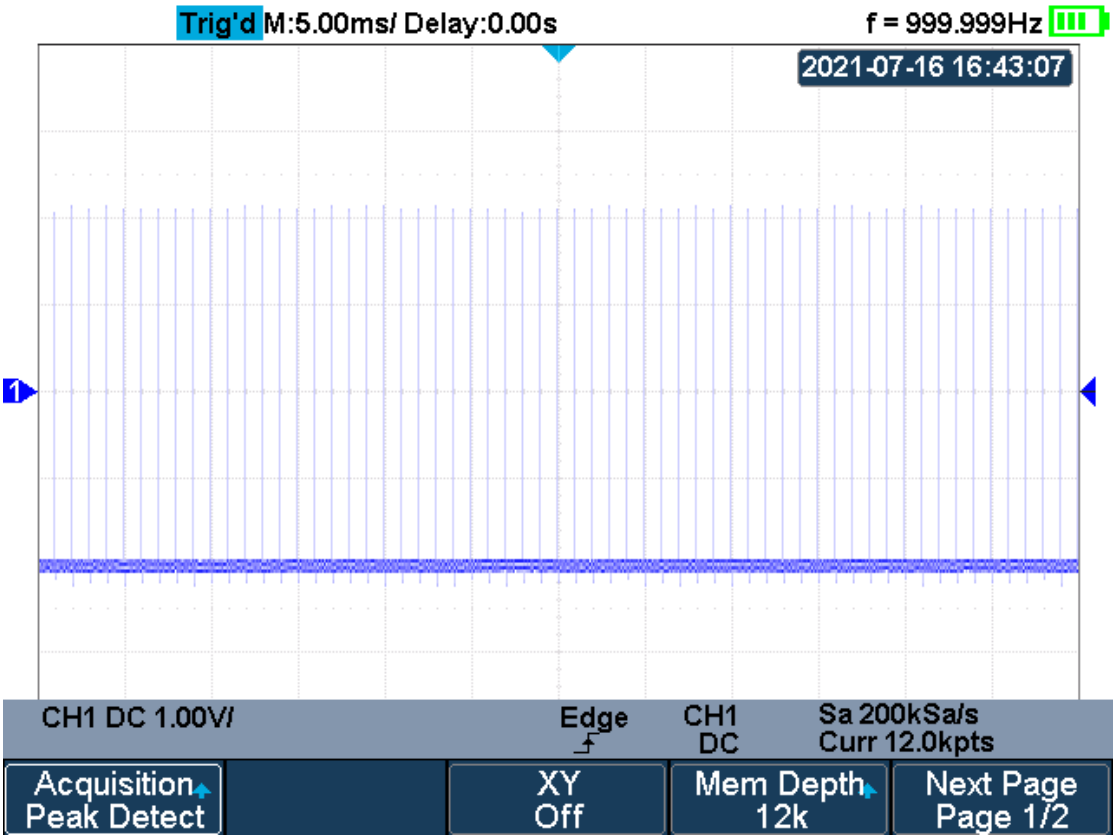


Figure 20 Pulse With 0.1% Duty, Peak Detect Mode

Average

In this mode, the oscilloscope averages the waveforms from multiple samples to reduce the random noise of the input signal and improve the vertical resolution. The greater the number of averages is, the lower the noise and the higher the vertical resolution will be. On the other hand, the greater the number of averages, the slower will be the response of the displayed waveform to real changes.

The available range of averages is from 4 to 1024 and the default is 16. When Average mode is selected, press **Averages** and turn the **Universal knob** or press the F key repeatedly to set the desired average time.

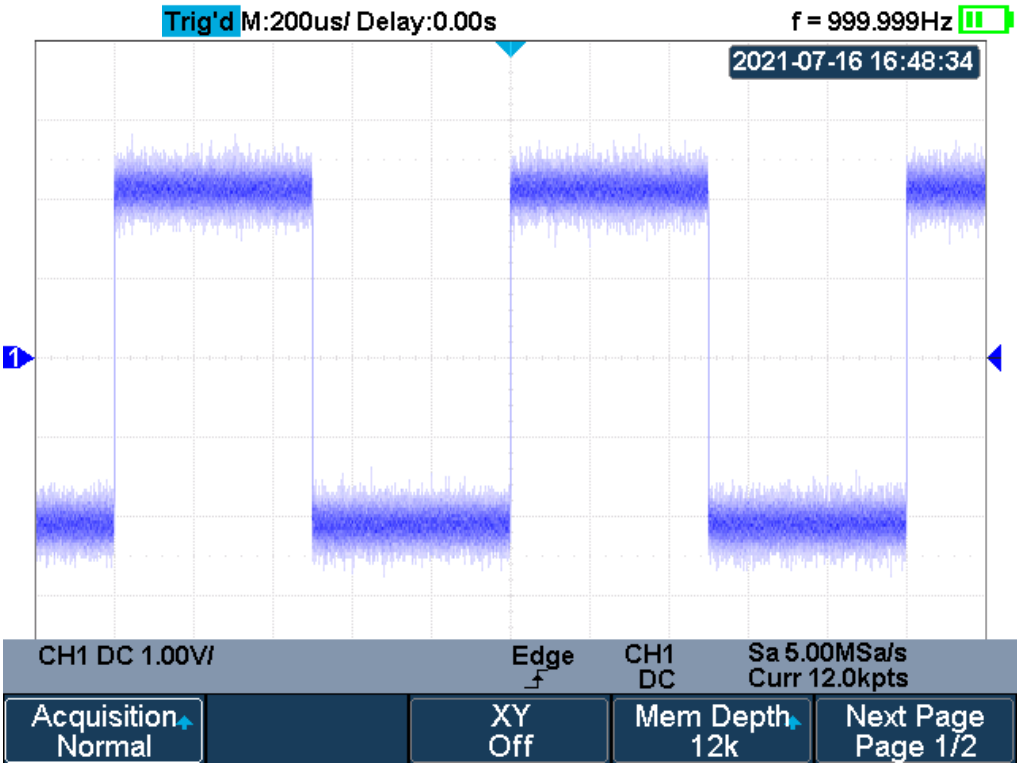


Figure 21 With Random Noise, Normal Mode

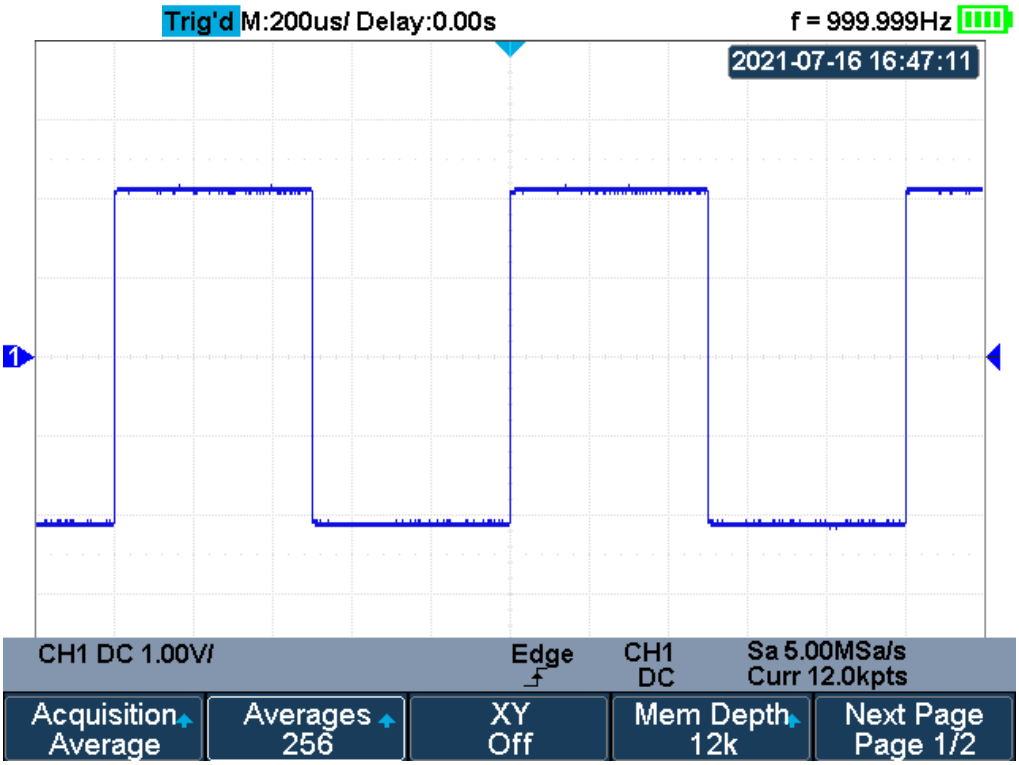


Figure 22 With Random Noise, Average Mode

ERES

This mode uses a kind of ultra-sample technique to average the neighboring points of the sample waveform to reduce random noise in the input signal and generate much smoother waveforms on the screen. ERES mode is generally used when the sample rate of the digital converter is higher than the storage rate of the acquisition memory.

ERES mode can be used on single-shot and repetitive signals and does not slow waveform updates. This mode does, however, limit the oscilloscope's real-time bandwidth because it effectively acts like a low- pass filter.

Note: “Average” and “ERES” mode use different averaging methods. The former uses “Waveform Average” and the latter uses “Dot Average”.

Horizontal Format

Press **Acquire** on the front panel; then press the **XY** F key to choose XY (On) or YT (Off) mode. The default setup is **YT**.

YT

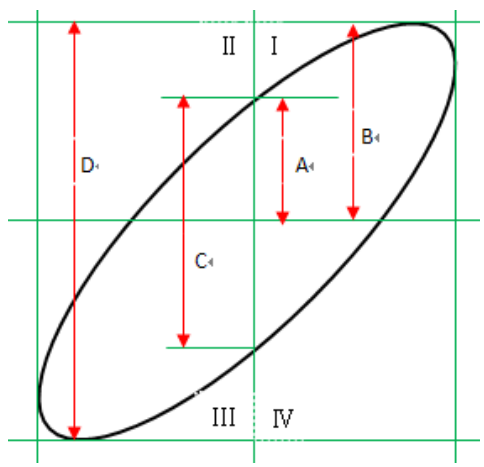
This is the normal viewing mode. In this mode, signal events occurring before the trigger are plotted to the left of the trigger point and signal events after the trigger plotted to the right of the trigger point.

XY

XY mode changes the display from volts vs time to a volts vs volts display, using signal from both channels. Channel 1 amplitude is plotted on the X- axis and Channel 2 amplitude is plotted on the Y-axis. The two channels will be turned on or off together.

You can use XY mode to compare frequency and phase relationships between two signals. XY mode can also be used with transducers to display strain versus displacement, flow versus pressure, volts versus current, or voltage versus frequency.

The phase deviation between two signals with the same frequency can be easily measured by the Lissajous method. The figure below is a schematic diagram of the phase deviation:



The phase deviation angle θ can be obtained using **sin**θ **= A/B** or **C/D** (where A, B, C, and D are as shown in the figure above): θ **= ±arcsin (A/B) or ±arcsin (C/D)**

If the principal axis of the ellipse is within quadrant I and III, the phase deviation angle obtained should be within quadrants I and IV, namely within (0 to π/2) or (3π/2 to 2π). If the principal axis of the is within quadrant II and IV, the phase deviation angle will be within quadrant II and III, namely within (π/2 to π) or (π to 3π/2).

XY mode can be used to measure the phase shift that occurs when the signal under test passes through a circuit network. Connect the oscilloscope to the circuit to monitor the input and output signals of the circuit.

Sequence Mode

Sequence is an acquisition mode that does not display the waveform during the sampling process. This improves the waveform capture rate, and the maximal capture rate is 400,000 wfs/s. Because of the high capture rate, sequence mode can capture small probability events effectively.

The oscilloscope runs and fills a memory segment for each trigger event. It repeats this process until all memory segments are filled, and then displays the waveforms on the screen.

To use sequence mode, the HORIZONTAL Format must be set to **YT**.

Do the following steps to use the sequence mode.

1. Press **Acquire** on the front panel to enter the ACQUIRE function menu;
2. Press the **Sequence** F key to enter the SEQUENCE function menu.

Figure 23 SEQUENCE Function Menu

3. Press the **Max. Segments** F key; and then turn the **Universal Knob** to select the desired value.

Do the following steps to replay the sequence waveform under history mode:

1. Press the **Shift** and **Meter** button to enable HISTORY function.

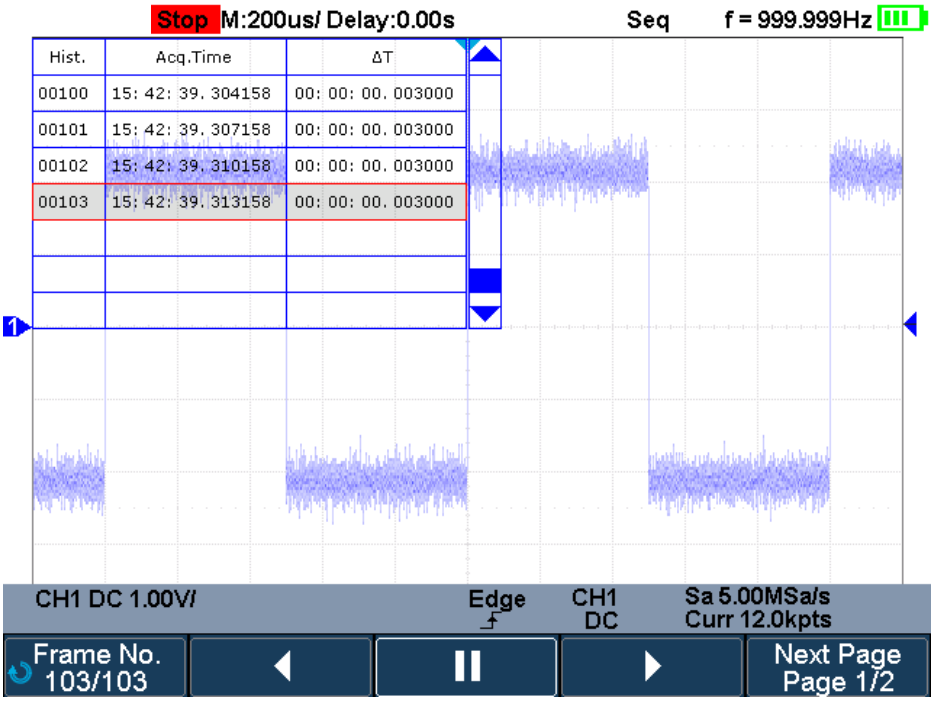


Figure 24 HISTORY Function Menu

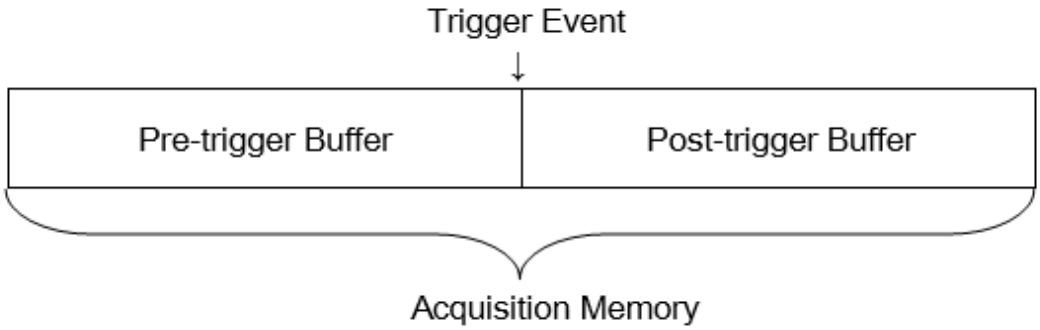
2. Press the **List** F key to turn on the list display. The list records the acquisition time of every frame and shows the frame number that displaying on the screen.

3. Press the **Frame No.** F key; and then turn the **Universal Knob** to select the frame to display.
4. Press the ◀ F key to replay the waveform from the current frame to 1.
5. Press the ❚❚ F key to stop replay.
6. Press the ▶ F key to replay the waveform from the current frame to the last frame.

Trigger

The oscilloscope can be set to trigger according to your requirements. When a waveform in the signal meets the trigger condition, the oscilloscope captures this waveform and the neighboring part and displays them on the screen. The oscilloscope displays a waveform whether or not it is stably triggered, but only a stable trigger can ensure a stable display. The trigger circuit ensures that every time base sweep or acquisition starts from the input signal and the user-defined trigger condition.

The following is a schematic diagram of the acquisition memory. The position of the trigger event is determined by the reference time point and the delay setting.



The trigger setting should be based on the features of the input signal. You should have some knowledge of the signal under test to quickly capture the desired waveform.

The oscilloscope provides abundant advanced trigger functions which can help you to focus on the desired waveform details. These trigger types are edge, slope, pulse, video, window, interval, dropout, runt, pattern and serial trigger. This chapter will describe these trigger types in detail and tell you how to set the trigger conditions to use them effectively.

Contents:

Trigger Source

Press **Trigger Setup** on the front panel to enter the TRIGGER function menu; press the **Source** F key and then turn the **Universal Knob** to select the desired trigger source.

The current trigger source is displayed at the bottom of the screen. Select the channel with signal input as the trigger source to obtain stable trigger.

Analog channel input:
Signals input from analog channels can all be used as the trigger source.

Trigger Mode

The oscilloscope's trigger mode includes auto, normal, single and Force. The mode affects how the oscilloscope searches for the trigger event.

After the oscilloscope starts running, it first fills the pre-trigger buffer. After the pre-trigger buffer is filled, it begins to search for a trigger. Data continues to flow through this buffer while the scope searches for the trigger. As the scope continues to search, overflow from the pre-trigger buffer is discarded on a first-in, first-out basis.

When a trigger is found, the oscilloscope fills the post- trigger buffer and displays the acquisition memory.

Press the **Trigger Setup** button and **Mode** F key to select the desired trigger mode.

- In the **Auto** trigger mode (the default setting), if the specified trigger conditions are not found, triggers are forced and acquisitions are made so that signal activity is displayed on the oscilloscope.

  The **Auto** trigger mode is appropriate when:
  - Checking DC signals or signals with unknown levels or activity.
  - Trigger events occur often enough that forced triggers are unnecessary.

- In the **Normal** trigger mode, triggers and acquisitions only occur when the specified trigger conditions are found. Otherwise, the oscilloscope holds the original waveform and waits for the next trigger.

  The **Normal** trigger mode is appropriate when:
  - You only want to acquire specific events specified by the trigger settings.
  - Triggering on an infrequent signal from a serial bus (for example, I2C, SPI, CAN, LIN, etc.) or another signal that arrives in bursts. The **Normal** trigger mode lets you stabilize the display by preventing the oscilloscope from auto-triggering.

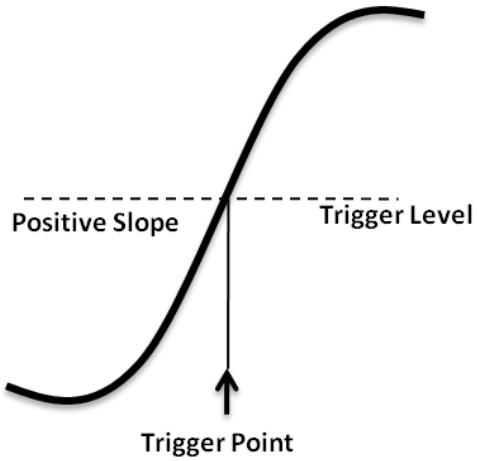
- In the **Single** trigger mode, the oscilloscope waits for a trigger, displays the waveform when the trigger condition is met, and stops.

  The **Single** trigger mode is appropriate when you want:
  - To capture a single event or a periodic signal.
  - To capture a burst or other unusual signals.

- In the **Force** trigger mode, when the trigger condition is not met, it will be force triggered after the frame is acquired. The trigger status in the upper left corner of the screen will be displayed as "FStop".

Trigger Level

The trigger level and slope define the trigger point,



You can adjust the trigger level for a selected analog channel by pressing **Trigger Level** and turning the **Universal Knob**.

You can push the **Universal Knob** to set the level to the waveform's 50% value immediately. If AC coupling is used, pushing the **Universal Knob** sets the trigger level to about 0 V.

The position of the trigger level is indicated by the trigger level icon (if the channel is on) at the left side of the display.

Trigger Coupling

Press **Trigger Setup** on the front panel to enter the TRIGGER function menu. Then, press the **Coupling** F key and turn the **Universal Knob** or press the **Coupling** F key repeatedly to select the desired coupling mode.

The oscilloscope provides 4 kinds of trigger coupling modes:

- **DC**: allow DC and AC components into the trigger path.
- **AC**: block all the DC components and attenuate signals lower than 8 Hz. Use AC coupling to get a stable edge trigger when your waveform has a large DC offset.
- **LF Reject**: block the DC components and reject the components lower than 2 MHz. Low frequency reject removes unwanted low frequency components from a trigger waveform, such as power line frequencies, etc. that can interfere with proper triggering. Use LF Reject coupling to get a stable edge trigger when your waveform has low frequency noise.
- **HF Reject**: reject the high frequency components (higher than 1.2 MHz)

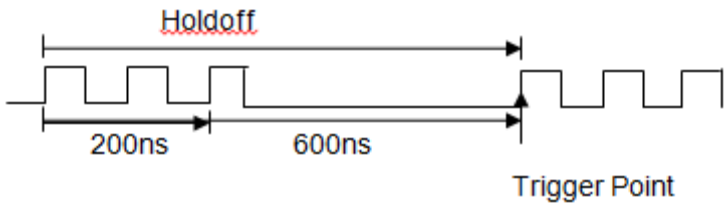
Note: trigger coupling has nothing to do with the channel coupling.

Trigger Holdoff

Trigger holdoff can be used to stably trigger complex waveforms (such as pulse series). The holdoff time is the amount of time that the oscilloscope waits before re-arming the trigger circuitry. The oscilloscope will not trigger until the holdoff time expires.

Use the holdoff to trigger repetitive waveforms with multiple edges (or other events) between waveform repetitions. You can also use holdoff to trigger on the first edge of a burst when you know the minimum time between bursts.

For example, to get a stable trigger on the repetitive pulse burst below, set the holdoff time to be >200 ns but <600 ns.



The correct holdoff setting is typically slightly less than one repetition of the waveform. Set the holdoff to this time to generate a unique trigger point for a repetitive waveform. Only edge trigger and serial trigger have the holdoff option. The holdoff time of the oscilloscope is adjustable from 80ns to 1.5s.

1. Press **Stop**, and then use the **Horizontal Position** button and the **Horizontal Scale** button to find where the waveform repeats. Measure this time using cursors; then, set the holdoff.
2. Press **Trigger Setup** to enter the TRIGGER function menu. The default trigger type is edge.
3. Press the **Holdoff Close** F key; and then turn the **Universal Knob** to set the desired holdoff time.

Noise Rejection

Noise Reject adds additional hysteresis to the trigger circuitry. By increasing the trigger hysteresis band, you reduce the possibility of triggering on noise. However, this also decreases the trigger sensitivity so that a slightly larger signal is required to trigger the oscilloscope.

Press **Trigger Setup** on the front panel, and then press the **Noise Reject** F key repeatedly to set the option to **On** or **Off** to turn on or off the noise rejection function.

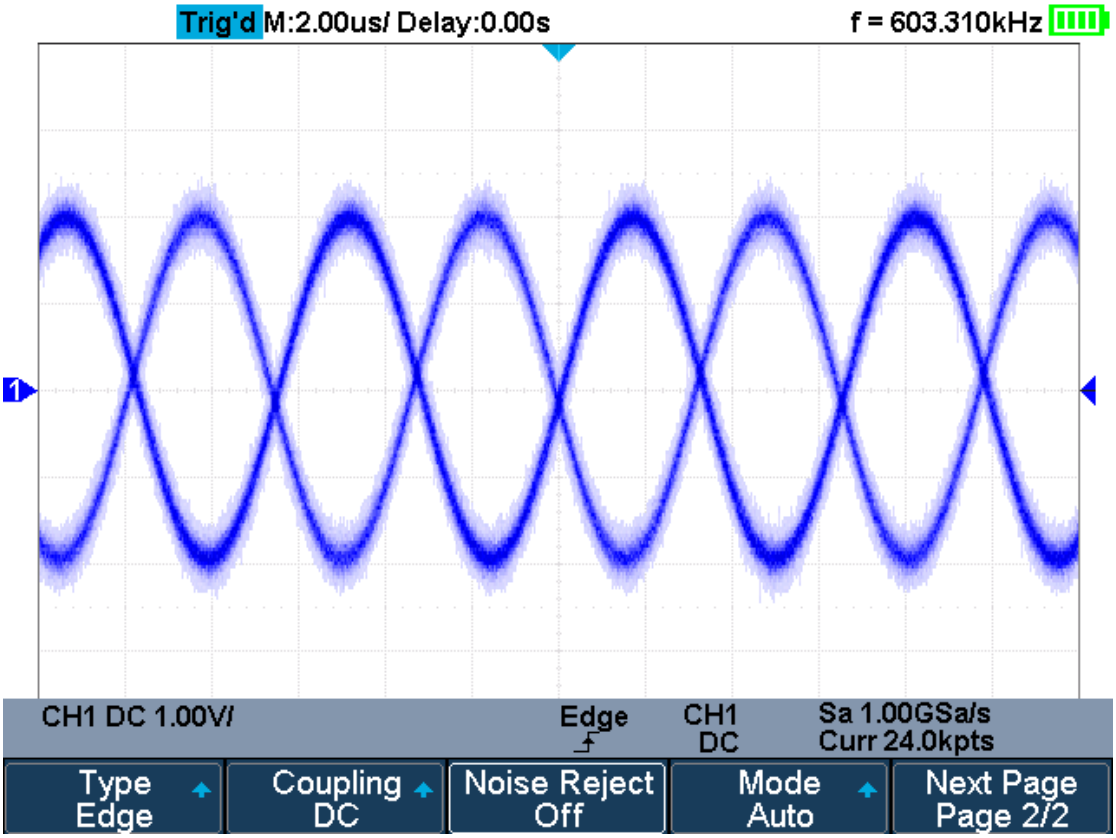


Figure 25 Turn off the Noise Reject

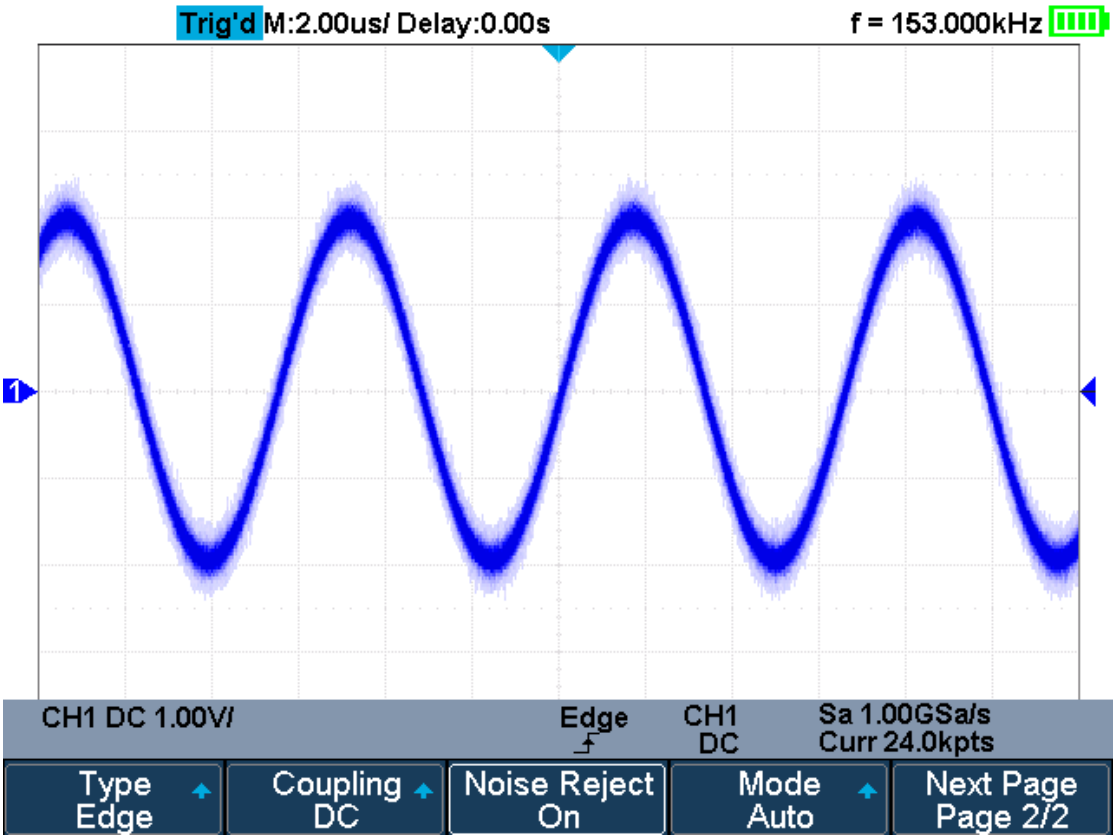


Figure 26 Turn on the Noise Reject

If the signal you are probing is noisy, you can set up the oscilloscope to reduce the noise in the trigger path and on the displayed waveform. First stabilize the displayed waveform by removing the noise from the trigger path. Second, reduce the noise on the displayed waveform.

1. Connect a signal to the oscilloscope and obtain a stable display.
2. Remove the noise from the trigger path by setting trigger coupling to **LF Reject** or **HF Reject**, or by turning on **Noise Reject**.
3. Set the **Acquisition** option to **Average** to reduce noise on the displayed waveform.

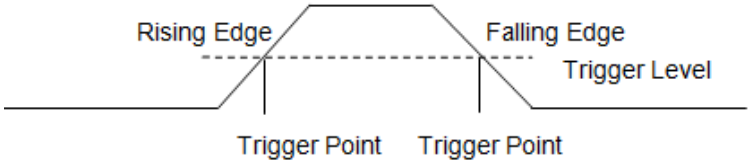
Trigger Types

The oscilloscope provides numerous advanced trigger functions, including various serial bus triggers.

- Edge trigger
- Slope trigger
- Pulse trigger
- Video trigger
- Window trigger
- Interval trigger
- Dropout trigger
- Runt trigger
- Pattern trigger

Edge Trigger

Edge triggering distinguishes the trigger points by seeking the specified edge (rising, falling, alternating) and trigger level.



1. Press **Trigger Setup** on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** F key; turn the **Universal Knob** to set select **Edge** and then push the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select the desired trigger source.
4. Press the **Slope** F key; turn the **Universal Knob** to select the desired trigger edge (rising, falling or alternating), and then press down the knob to confirm. The current trigger slope is displayed at the bottom of the screen.
5. Press the **Trigger Level** button and turn the **Universal Knob** to adjust the trigger level to obtain a stable trigger.

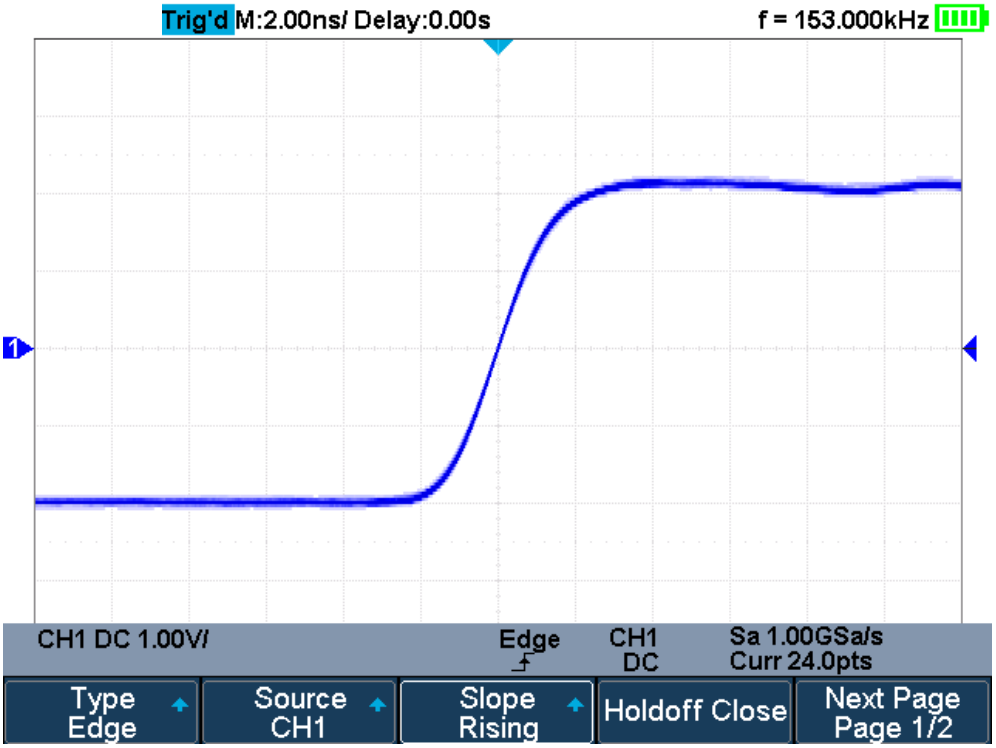


Figure 27 Edge Trigger

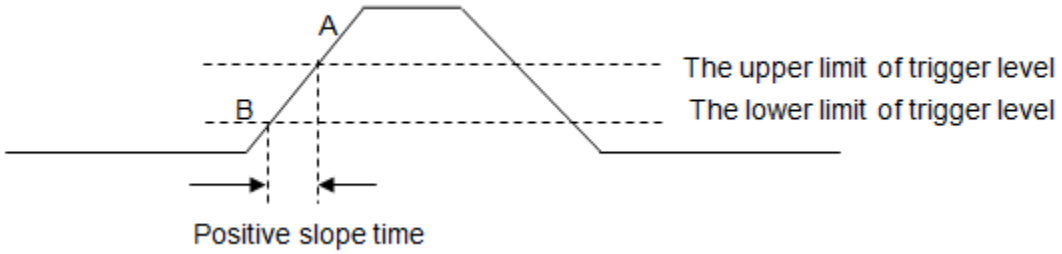
Holdoff, coupling, and noise rejection can be set in edge trigger mode. See the sections "Trigger Holdoff," "Trigger Coupling," and "Noise Rejection" for details.

Note: Press the Auto Setup button will set the trigger type to Edge and slope to rising.

Slope Trigger

The slope trigger looks for a rising or falling transition from one level to another level in greater than or less than a certain amount of time.

In the oscilloscope, positive slope time is defined as the time difference between the two crossing points of trigger level line A and B with the positive edge as shown in the figure below.



1. Press **Trigger Setup** on the front panel to enter the TRIGGER function menu.
2. Press the **Type** F key; turn the **Universal Knob** to set select **Slope** and then push the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select the trigger source.
4. Press the **Slope** F key; turn the **Universal Knob** to set select the desired trigger edge (rising or falling), and then push down the knob to confirm. The current trigger slope is displayed at the upper right corner of the screen.
5. Press **Lower Upper** F key to select the **Lower** or **Upper** trigger level; then press the **Trigger Level** button and turn the **Universal Knob** to adjust the position.
   The lower trigger level cannot be upper than the upper trigger level.

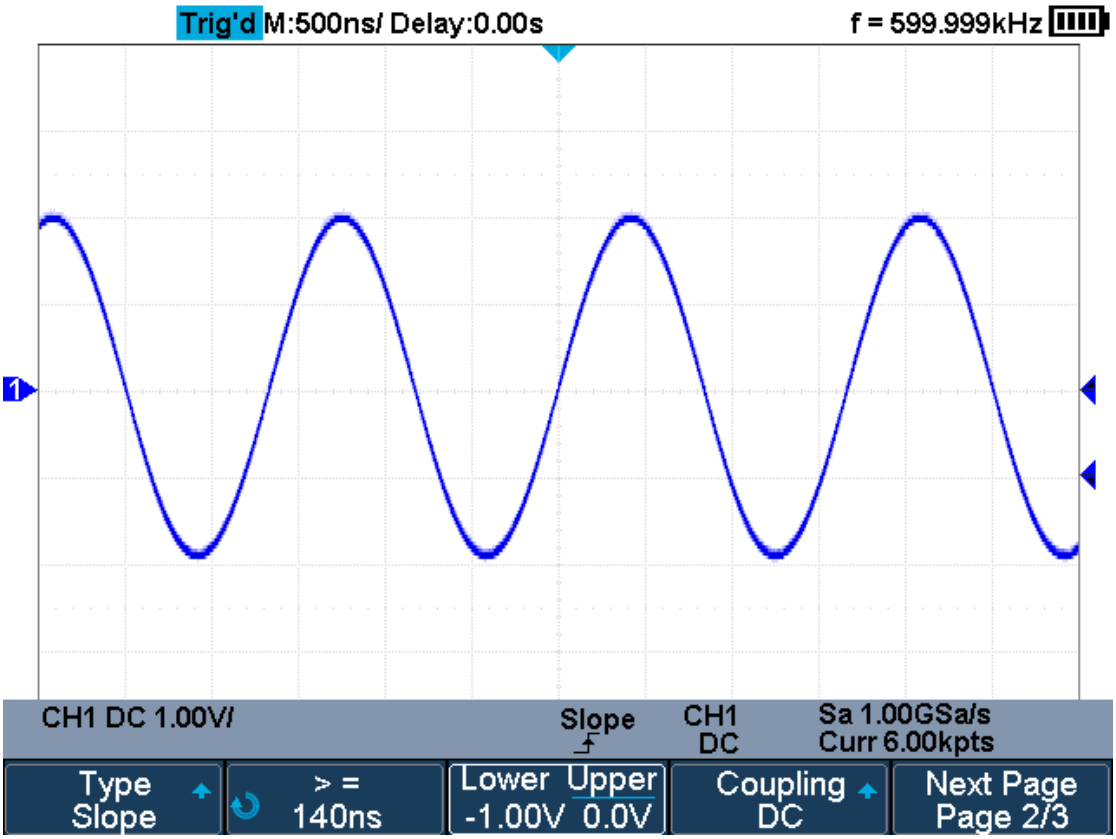


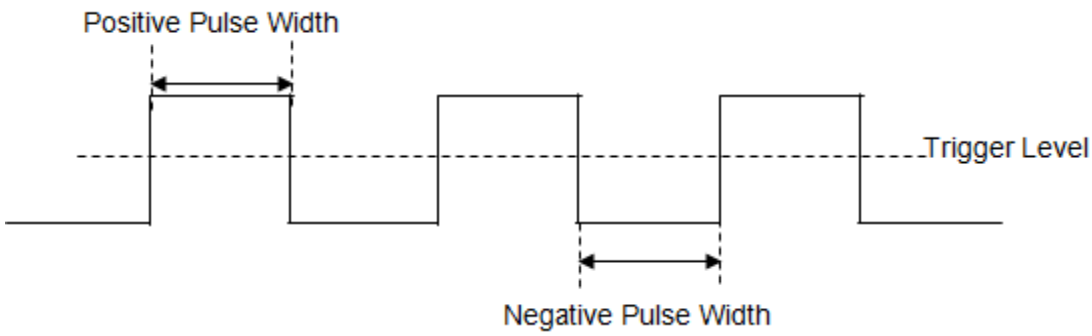
Figure 28 Slope Trigger

6. Press the **Limit Range** F key; turn the **Universal Knob** to select the desired slope condition, and push down the knob to confirm.
   - **<= (less than a time value):** trigger when the input signal's positive or negative slope time is lower than the specified time value.
   - **>= (greater than a time value):** trigger when the input signal's positive or negative slope time is greater than the specified time value.
   - **[--,--] (within a range of time value):**trigger when the input signal's positive or negative slope time is greater than the specified lower limit of time and lower than the specified upper limit of time value.
   - **--][-- (outside a range of time value):** trigger when the input signal's positive or negative slope time is greater than the specified upper limit of time and lower than the specified lower limit of time value.

Coupling and noise rejection can be set in slope trigger mode; see the sections "Trigger Coupling" and "Noise Rejection" for details.

Pulse Trigger

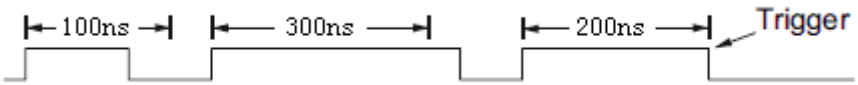
Trigger on the positive or negative pulse with a specified width.



1. Press **Trigger Setup** on the front panel to enter the TRIGGER function menu.
2. Press the **Type** F key; turn the **Universal Knob** to select **Pulse** and then push the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 or CH2 as the trigger source.
4. Press **Trigger Level** and turn the **Universal Knob** to adjust the trigger level to the desired place.
5. Press the **Polarity** F key to select **Positive** or **Negative** pulse to trigger on. The current trigger polarity is displayed at the upper right corner of the screen.
6. Press the **Limit Range** F key; turn the **Universal Knob** to select the desired condition.
   - **<= (less than a time value):** trigger when the input signal's positive or negative pulse time is lower than the specified time value.
     For example, for a positive pulse, if you set t (pulse real width) ≤ 100ns, then the first pulse below will not fire the trigger, but the second pulse will.

   - **>= (greater than a time value):** trigger when the input signal's positive or negative pulse time is greater than the specified time value.
     For example, for a positive pulse, if you set t (pulse real width) ≥ 100ns, the first pulse below will not satisfy the trigger, but the second one will.

   - **[--,--] (within a range of time value):** trigger when the input signal's positive or negative pulse time is greater than the specified lower limit of time and lower than the specified upper limit of time value.
     For example, for a positive pulse, if you set t (pulse real width) ≥100ns and t ≤ 300ns, the waveform will trigger.



- **--][-- (outside a range of time value):** trigger when the input signal's positive or negative pulse time is greater than the specified upper limit of time and lower than the specified lower limit of time.

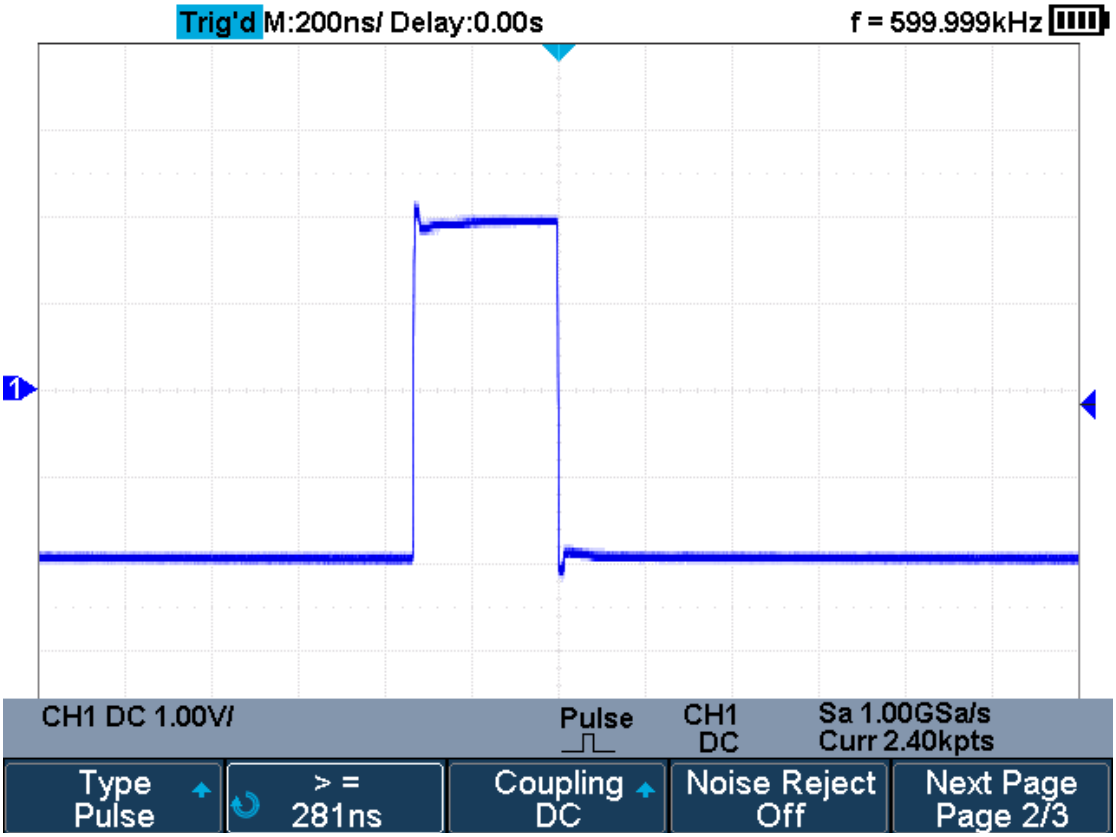


Figure 29 Pulse Trigger

Coupling and noise rejection can be set in pulse trigger mode. See the sections "Trigger Coupling" and "Noise Rejection" for details.

Video Trigger

Video triggering can be used to capture the complicated waveforms of most standard analog video signals. The trigger circuitry detects the vertical and horizontal interval of the waveform and produces triggers based on the video trigger settings you have selected.

The oscilloscope supports standard video signal field or line of NTSC (National Television Standards Committee), PAL (Phase Alternating Line) HDTV (High Definition Television), and custom video signal trigger.

1. Press **Trigger Setup** on the front panel to enter the TRIGGER function menu.
2. Press the **Type** F key; then turn the **Universal Knob** to select **Video** and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 or CH2 as the trigger source. The trigger level is automatically set to the sync pulse.
4. Press the **Standard** F key to select the desired video standard. The oscilloscope supports the following video standards:

Standard	Type	Sync Pulse
NTSC	Interlaced	BI-level
PAL	Interlaced	BI-level
HDTV 720P/50	Progressive	Tri-level
HDTV 720P/60	Progressive	Tri-level
HDTV 1080P/50	Progressive	Tri-level
HDTV 1080P/60	Progressive	Tri-level
HDTV 1080i/50	Progressive	Tri-level
HDTV 1080i/50	Progressive	Tri-level
Custom		

The table below shows the parameters of the Custom video trigger.

Frame Rate	25Hz, 30Hz, 50Hz, 60Hz	
Of Lines	300~2000	
Of Fields	1, 2, 3, 4	
Interlace	1:1, 2:1, 4:1, 8:1	
Trigger Position	**Line**	**Field**
	(line value)/1	1
	(line value)/2	2
	(line value)/3	3
	(line value)/4	4
	(line value)/5	5
	(line value)/6	6
	(line value)/7	7
	(line value)/8	8

The table below takes **Of Lines** as 800 as an example to explain the relation between **Of Lines**, **Of Fields**, **Interlace**, **Trigger Line**, and **Trigger Field**.

Of Lines	Of Fields	Interlace	Trigger Line	Trigger Field
800	1	1:1	800	1
800	1,2,4 or 8	2:1	400	1, 1~2, 1~4, 1~8
800	1,2,4 or 8	4:1	200	1, 1~2, 1~4, 1~8
800	1,2,4 or 8	8:1	100	1, 1~2, 1~4, 1~8

5. Press the **Sync** F key to select **Any** or **Select** trigger mode.
   - **Any**: trigger on any of the horizontal sync pulses
   - **Select**: trigger on the appointed line and field you have set. Press the **Line** or **Field** F key; then turn the **Universal Knob** to set the value.

   The following table lists the line numbers per field for each video standard.

Standard	Field 1	Field 2
NTSC	1 to 262	1 to 263
PAL	1 to 312	1 to 313
HDTV 720P/50, HDTV 720P/60	1 to 750	
HDTV 1080P/50, HDTV 1080P/60	1 to 1125	
HDTV 1080iP/50, HDTV 1080i/60	1 to 562	1 to 563

The following are examples to familiarize you with video triggering.

- To trigger on a specific line of video
- To use Custom video trigger

To Trigger on a Specific Line of Video

Video triggering requires greater than 1/2 division of sync amplitude with either channel as the trigger source.

The example below set to trigger on field 2, line 124 using the NTSC video standard.

1. Press **Trigger Setup** on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** F key; then use the **Universal Knob** to select Video and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 as the trigger source, and press the knob to confirm.
4. Press the **Standard** F key; turn the **Universal Knob** to select NTSC, and press the knob to confirm.
5. Press the **Sync** F key and set the option to **Select**; press the **Line** F key and then turn the universal to select **1** and push the knob to confirm; press the **Field** F key and then turn the **Universal Knob** to select **1** and push the knob to confirm

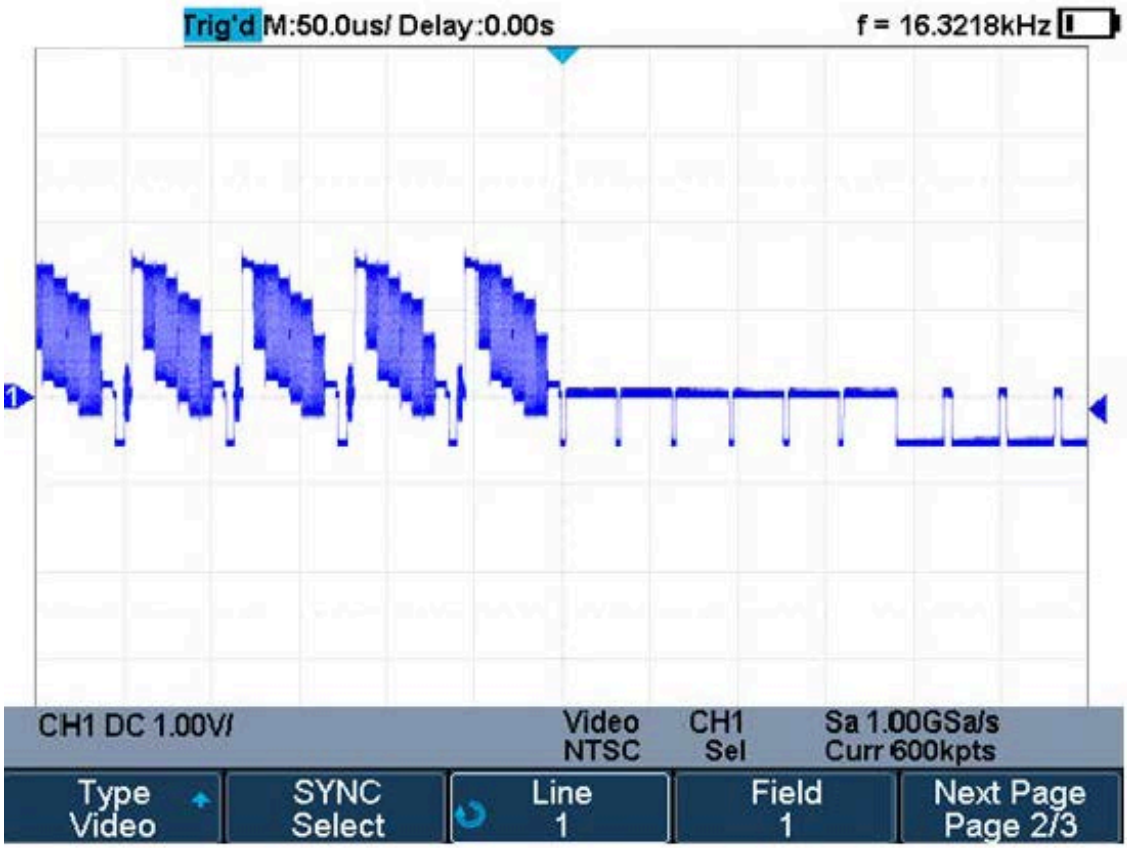


Figure 30 Video Trigger

To Use Custom Video Trigger

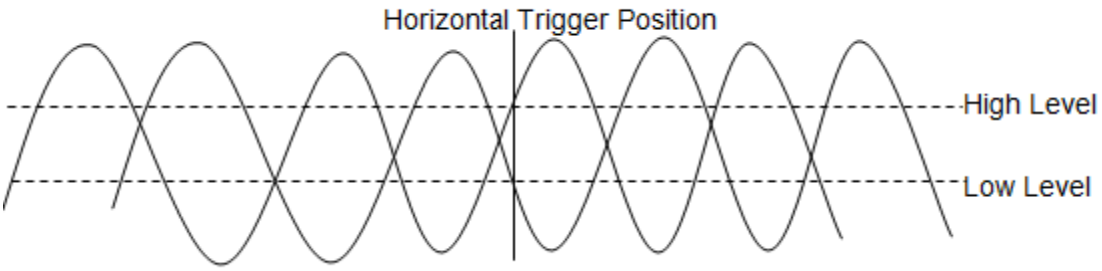
Custom video trigger supports frame rate of 25Hz, 30Hz, 50Hz and 60Hz, and the line range is available from 300 to 2000. The steps below show how to set custom trigger.

1. Press **Trigger Setup** on the front panel to enter the TRIGGER function menu.
2. Press the **Type** F key; then use the **Universal Knob** to select **Video** and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select **CH1** as the trigger source, and push down the knob to confirm.
4. Press the **Standard** F key; turn the **Universal Knob** to select **Custom**, and push down the knob to confirm.
5. Press the **Setting** F key to enter the custom setting function menu. Press the **Interlace** F key; turn the **Universal Knob** to select the desired value.
6. Press the **Of Field** F key; turn the **Universal Knob** to select the desired value.
7. Press the **Sync** F key to enter the TRIG ON menu to set the line and field.
   - Press the **Type** F key to select **Select** or **Any**.
   - If the **Type** option set to **Select**, press the **Line** F key; turn the **Universal Knob** to select the desired value. Press the **Field** F key; turn the **Universal Knob** to select the desired value.

Window Trigger

The window trigger mode has a high trigger level and a low trigger level. The instrument triggers when the input signal passes through either trigger level.

There are two kinds of window types: Absolute and Relative. They have different trigger level adjustment methods. Under Absolute window type, the lower and the upper trigger levels can be adjusted with the Level knob; under Relative window type, you instead set the center of the window, and then a "delta" value. The upper trigger level is the center plus delta; the lower trigger level is center minus delta.



- If the lower and the upper trigger levels are both within the triggering range, the oscilloscope will trigger on both rising and falling edges.
- Suppose the upper trigger level is within the triggering range while the lower trigger level is out of the triggering range. In that case, the oscilloscope will trigger on rising edge only.

To set absolute window triggers:

1. Press **Trigger Setup** on the front panel to enter the TRIGGER function menu.
2. Press the **Type** F key; then use the **Universal Knob** to select **Window** and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 or CH2 as the trigger source.
4. Press the Window **Type** F key to select Absolute.
5. Press the **Lower Upper** F key to select **Lower** or **Upper** trigger level; then press **Trigger Level** and turn the **Universal Knob** to adjust the position. The trigger level values are displayed at the upper right corner of the screen.
   The lower trigger level cannot be higher than the upper trigger level.

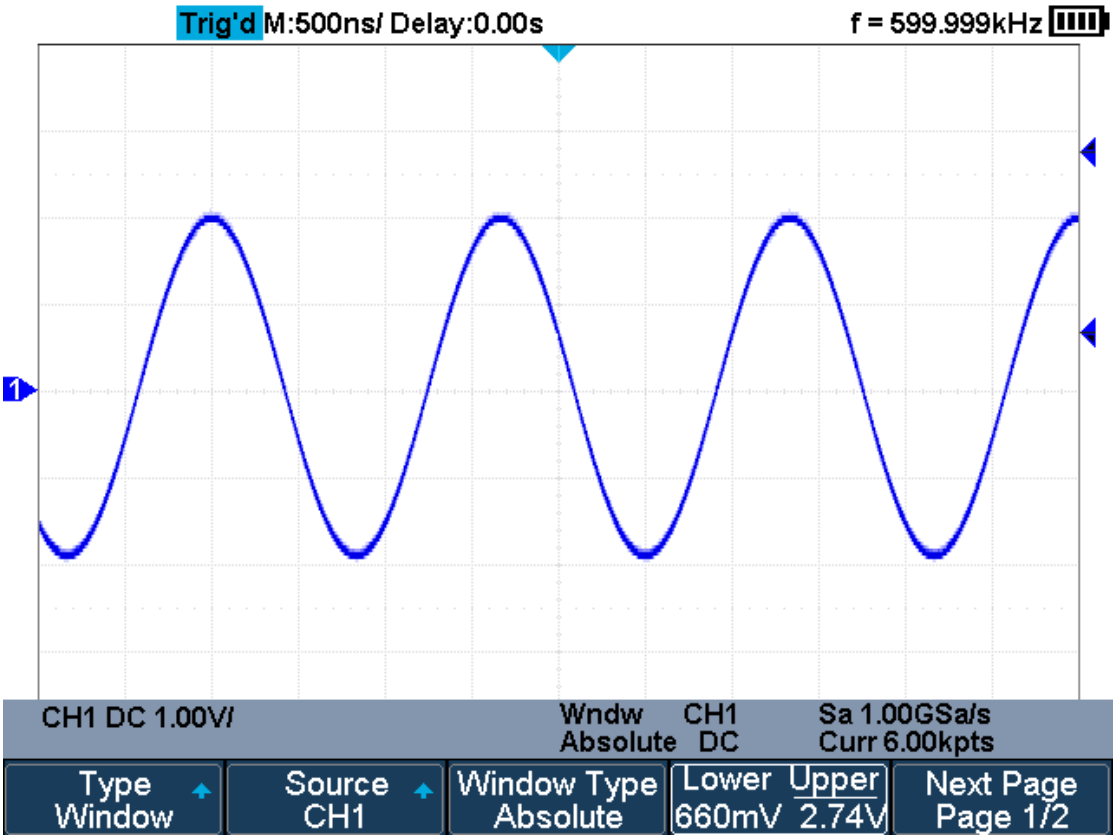


Figure 31 Absolute Window Trigger

To set relative window triggers:

1. Press **Trigger Setup** on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** F key; then use the **Universal Knob** to select **Window** and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 or CH2 as the trigger source.
4. Press the **Window Type** F key to select **Relative**.
5. Press the **Center Delta** F key to select **Center** or **Delta** trigger level mode; then press **Trigger Level** and turn the **Universal Knob** to adjust the position.

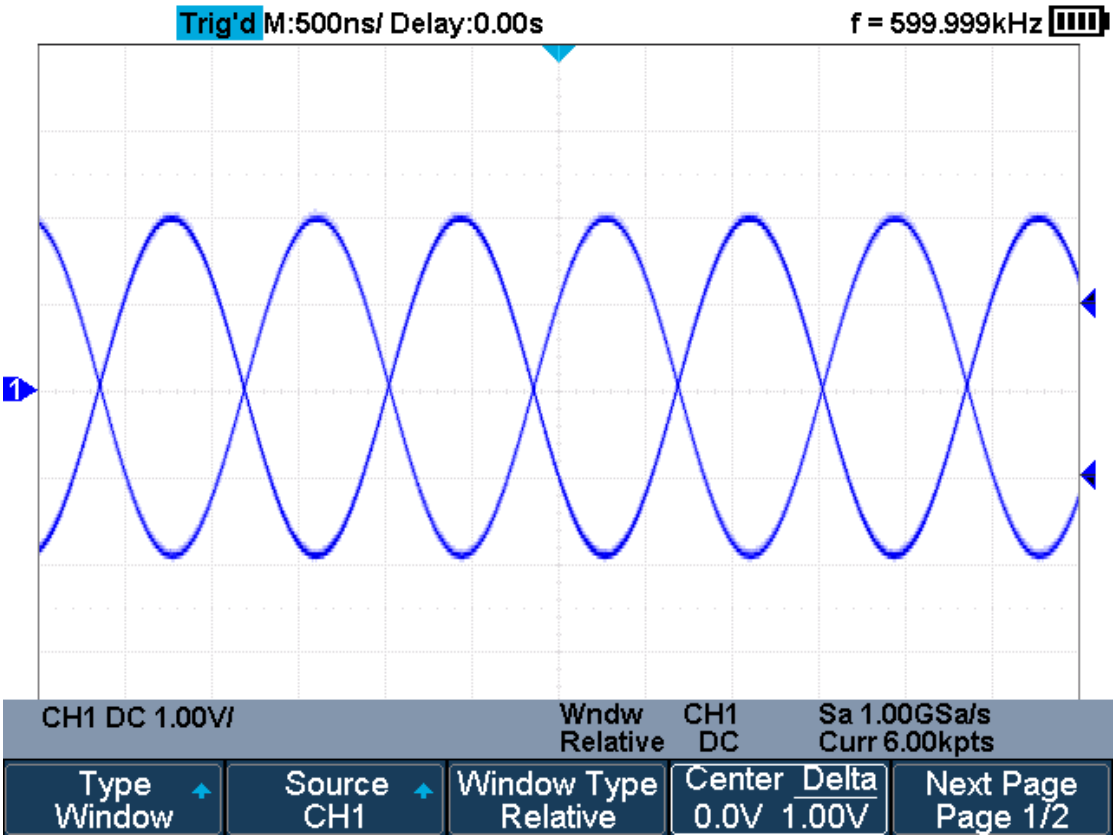
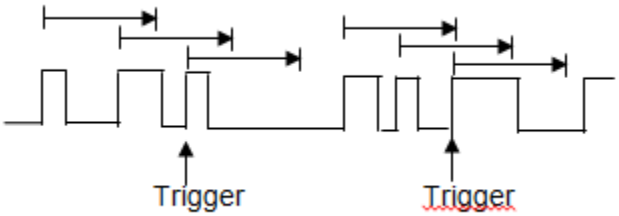


Figure 32 Relative Window Trigger

Coupling and noise rejection can be set in Window trigger mode. See the sections "Trigger Coupling" and "Noise Rejection" for details.

Interval Trigger

Trigger when the time difference between neighboring rising/falling edges meets the specified criteria (<=, >=, [--,--], --][--).



To set an interval trigger:

1. Press **Trigger Setup** on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** F key; then use the **Universal Knob** to select **Interval** and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 or CH2 as the trigger source.
4. Press the **Slope** F key to select rising or falling edge.
5. Press the **Limit Range** F key; turn the **Universal Knob** to select desired condition.
   - **<= (less than a time value):** trigger when the input signal's positive or negative pulse time is lower than the specified time value.
   - **>= (greater than a time value):** trigger when the input signal's positive or negative pulse time is greater than the specified time value.
   - **[--,--] (within a range of time value):**trigger when the input signal's positive or negative pulse time is greater than the specified lower limit of time and lower than the specified upper limit of time value.
   - **--][-- (outside a range of time value):** trigger when the input signal's positive or negative pulse time is greater than the specified upper limit of time and lower than the specified lower limit of time value.

6. Press the **Time Setting** F key (**<=, >=, [--,--],--][--**), turn the **Universal Knob** to select the desired value.

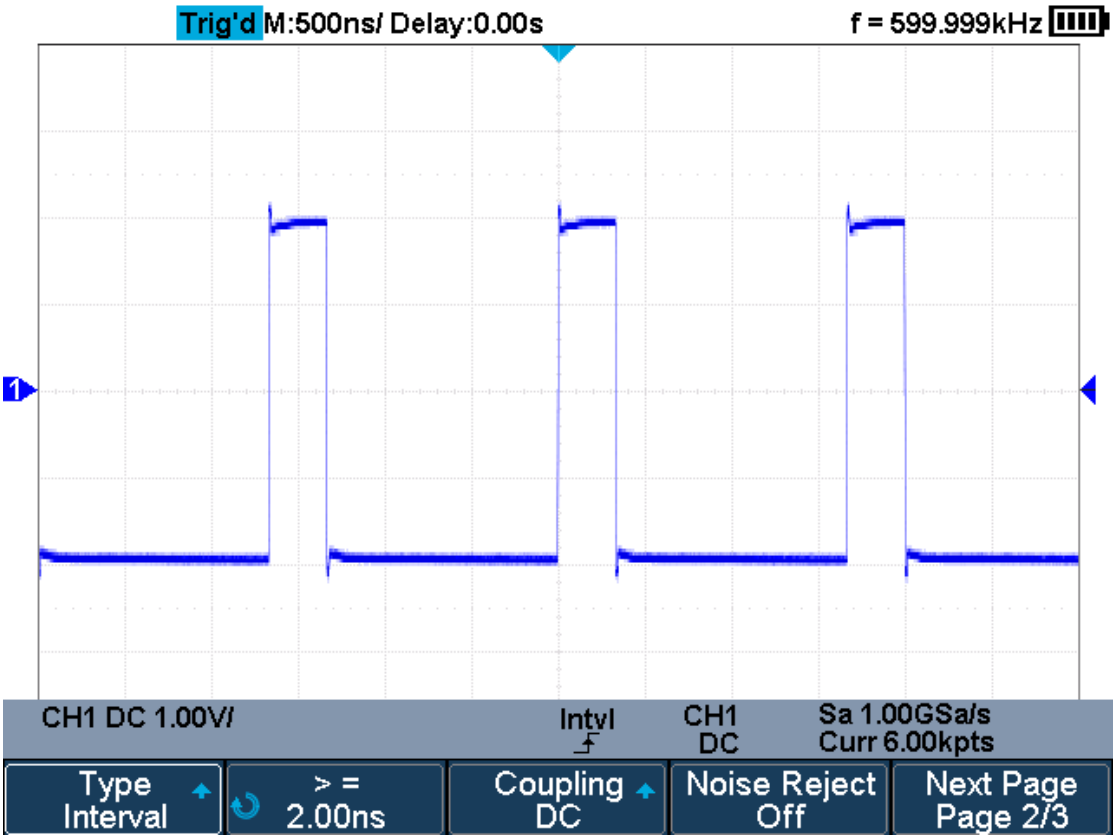


Figure 33 Interval Trigger

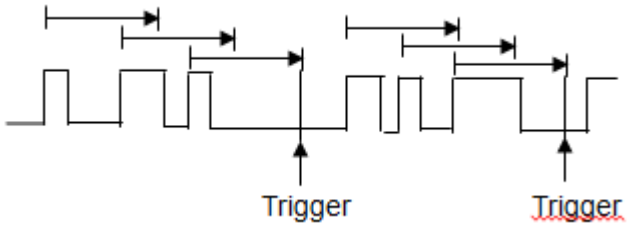
Coupling and noise rejection can be set in interval trigger mode. See the sections "Trigger Coupling" and "Noise Rejection" for details.

Dropout Trigger

Dropout trigger mode has two types: Edge and State.

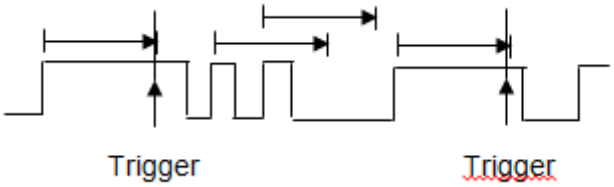
Edge

Trigger when the time interval ΔT - from when the rising edge (or falling edge) of the input signal passes through the trigger level to when the next rising or falling edge passes through the trigger level - is greater than the timeout time set, as shown in the figure below.



State

Trigger when the time interval ΔT - from when the rising edge (or falling edge) of the input signal passes through the trigger level to when the neighboring falling or rising edge passes through the trigger level - is greater than the timeout time set, as shown in the figure below.



To set edge Dropout trigger:

1. Press **Trigger Setup** on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** F key; then use the **Universal Knob** to select **Dropout** and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 or CH2 as the trigger source. The current trigger source is displayed at the upper right corner of the screen. Select channel with signal input as trigger source to obtain stable trigger.
4. Press the **Slope** F key to select rising or falling edge.
5. Press the **OverTime Type** F key to select **Edge**.
6. Press the **Time** F key; turn the **Universal Knob** to select the desired value.

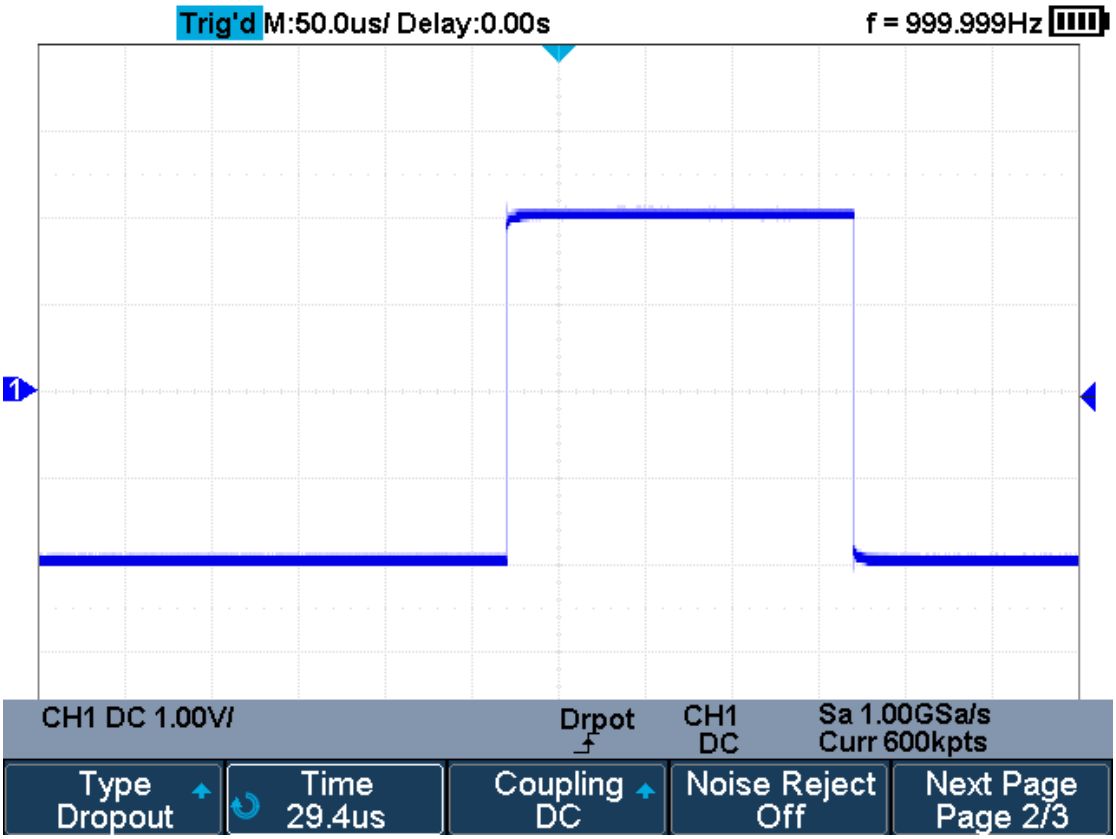


Figure 34 Edge Dropout Trigger

To set state Dropout trigger:

1. Press **Trigger Setup** to enter the TRIGGER system function menu.
2. Press the **Type** F key; then turn the **Universal Knob** to select **Dropout** and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 or CH2 as the trigger source.
4. Press the **Slope** F key to select rising or falling edge.
5. Press the **OverTime** Type F key to select State.
6. Press the **Time** F key; turn the **Universal Knob** to select the desired value.

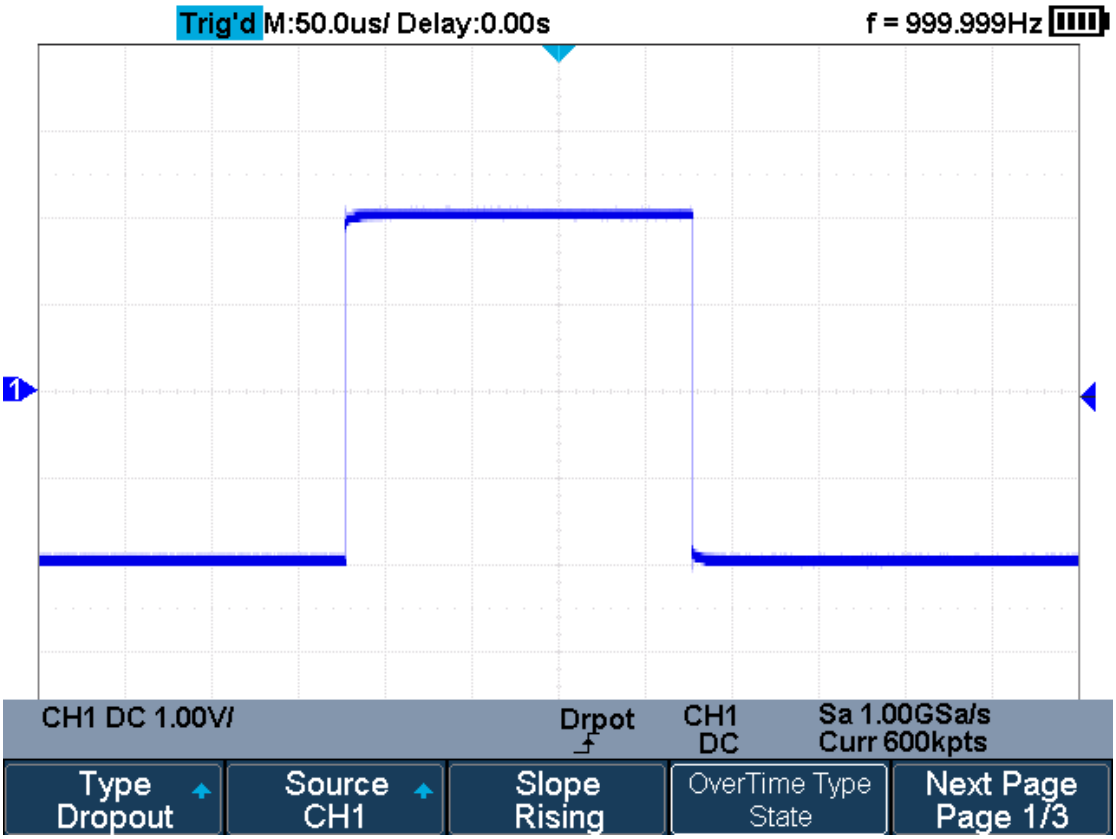
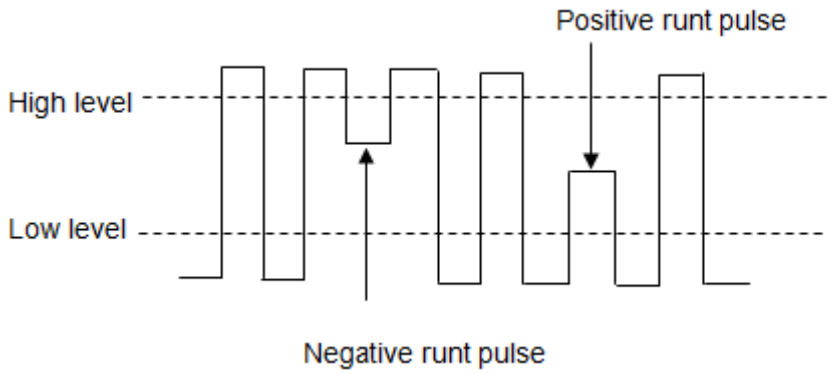


Figure 35 State Dropout Trigger

Coupling and noise reject can also be set in dropout trigger mode. See the sections "Trigger Coupling" and "Noise Rejection" for details.

Runt Trigger

Runt trigger mode has two trigger levels, high and low. It looks for pulses that cross one threshold but not the other, as shown below.



- A positive runt pulse across through a lower threshold but not an upper threshold.
- A negative runt pulse across through an upper threshold but not a lower threshold.

To trigger on runt pulse:

1. Press **Trigger Setup** on the front panel to enter the **TRIGGER** system function menu.
2. Press the **Type** F key; then turn the **Universal Knob** to select **Runt** and push down the knob to confirm.
3. Press the **Source** F key; turn the **Universal Knob** to select CH1 or CH2 as the trigger source.
4. Press the **Polarity** F key to select **Positive** or **Negative** pulse to trigger.
5. Press the **Limit Range** F key; turn the **Universal Knob** to select the desired condition (**<=**, **>=**, **[--,--]** or **--][--**).
6. Press the **Time Setting** F key, and then turn the **Universal Knob** to select the desired value.
7. Press the **Next Page** F key to enter the second page of the TRIGGER system function menu. Press the **Lower Upper** F key to select **Lower** or **Upper** trigger level and press the **Trigger Level** button and turn the **Universal Knob** to set the position.

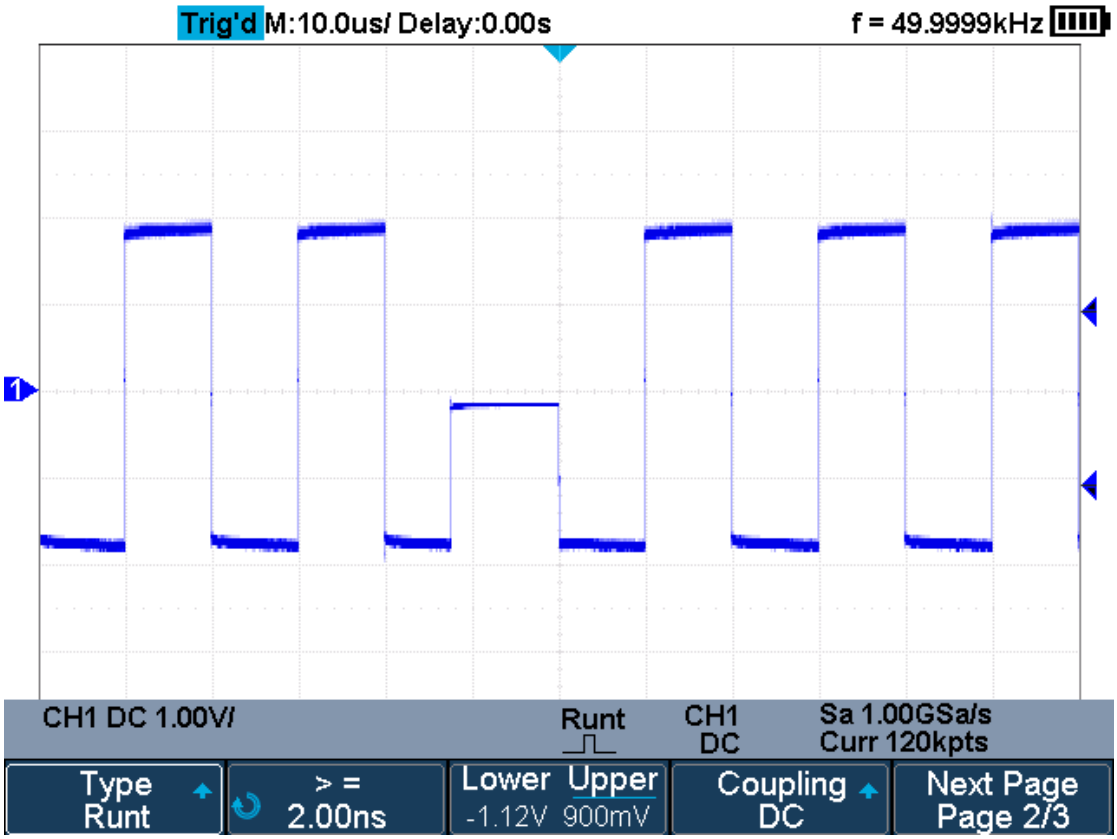
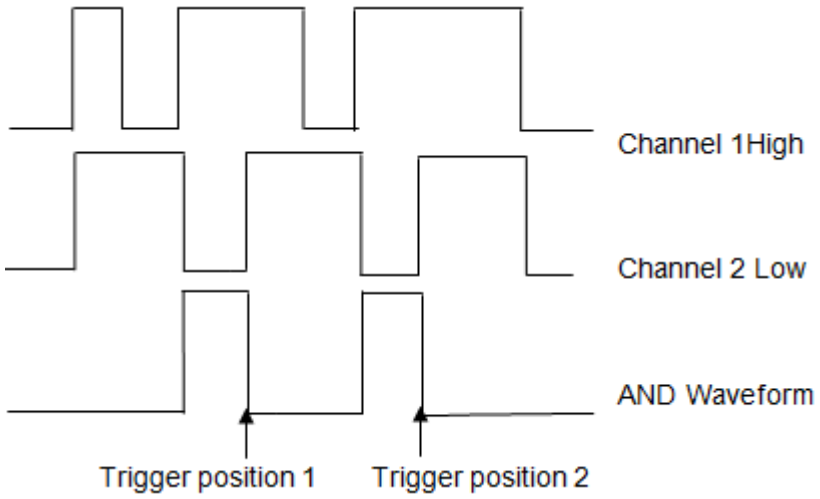


Figure 36 Runt Trigger

Coupling and noise rejection can be set in runt trigger mode. See the sections "Trigger Coupling" and "Noise Rejection" for details.

Pattern Trigger

Pattern trigger mode identifies a trigger condition by looking for a specified pattern. The pattern trigger can be expanded to incorporate delays, similar to other triggers. Pattern durations are evaluated using a timer. The timer starts on the last edge that makes the pattern "true". Potential triggers occur on the first edge that makes the pattern false, provided that the time qualifier criterion has been met. The oscilloscope provides 4 patterns: logical AND, OR, NAND and NOR combination of the channels. Each channel can set to low, high or invalid.



To trigger on a pattern:

1. Press **Trigger Setup** on the front panel to enter the TRIGGER function menu.
2. Press the **Type** F key; turn the **Universal Knob** to select **Pattern**, and then push down the knob to confirm.
3. Press the **Source** F key to select channel, and then press the level type F key to select **Don't care**, **High** or **Low**.
   - **Low** sets the pattern to low on the selected channel. ***Low*** means the channel's voltage is less than its trigger level.
   - **High** sets the pattern to high on the selected channel. A ***High*** voltage is greater than the channel's trigger level.
   - **Don't care** sets the pattern to don't care on the selected channel. Any channel set to don't care is ignored and is not used as part of the pattern.
   - Note: If all channels in the pattern are set to **Don't care**, the oscilloscope will not trigger. Adjust the trigger level for the selected analog channel by pressing **Trigger Level** and turning the **Universal Knob**. **Don't care** channels don't need their trigger levels set.
4. Press the **Next Page** F key to enter the second page of the pattern trigger menu.
5. Press the **Logic** F key and then turn the **Universal Knob** to select the desired logic combination **AND**, **OR**, **NAND** or **NOR**.
6. Press the **Time** F key; then turn the **Universal Knob** to select the desired time value.

7. Press the **Holdoff Close** F key to turn on the Holdoff function; then turn the **Universal Knob** to select the desired value.

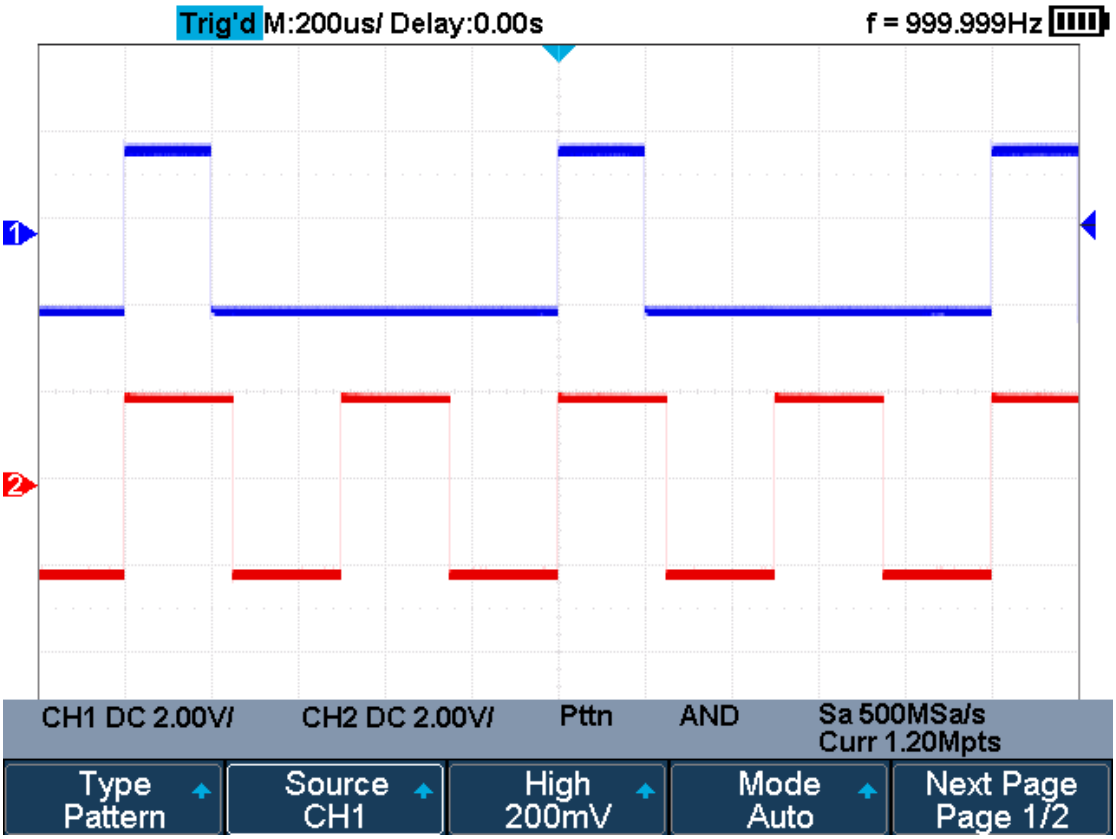


Figure 37 Pattern Trigger

Holdoff mode can be set using patterns. See the section "Trigger Holdoff" for details.

Serial Trigger and Decode

The oscilloscope provides I2C, SPI, UART, CAN and LIN serial trigger and decode functions.

Contents:

I2C Trigger and Serial Decode

Setup for I2C Signals

Setting the I2C (Inter-IC bus) signal entails two steps: connecting the serial data signal (SDA) and serial clock signal (SCK) to oscilloscope, and specifying the threshold voltage of each input signal.

1. Press **Shift** and **Print** to enter the **DECODE** function menu as Figure 38 shows.

Figure 38 I2C DECODE Menu

2. Press the **Decode** F key and select the desired slot (Decode1 or Decode2).
3. Press the **Protocol** F key and then select **I2C** by turning **Universal Knob**.
4. Press the **Signal** F key to enter the **SIGNAL** menu as Figure 39 shows.

Figure 39 I2C SIGNAL Menu

5. Set SCL (I2C's clock signal):
   a. Press the **SCL** F key to select the channel that is connected to the I2C clock signal.
   b. Press the first **Threshold** F key to set the I2C clock signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.
6. Set SDA (I2C's data signal):
   a. Press the **SDA** F key to select the channel that is connected to the I2C data signal.
   b. Press the second **Threshold** F key to set the I2C data signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.

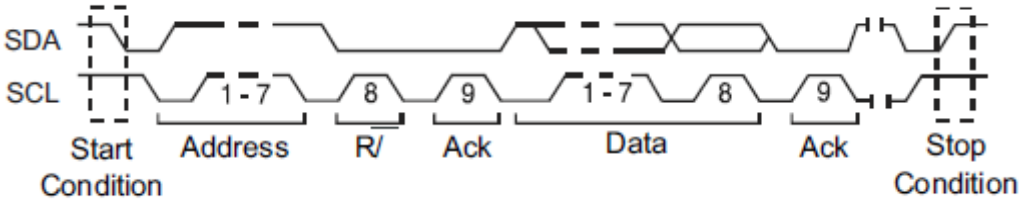
   (Note: SDA should keep stable during the whole high clock cycle, otherwise it will be interpreted as a start or stop condition (data transitioning while the clock is high))
7. Press the ⮌ F key to return previous menu.

I2C Trigger

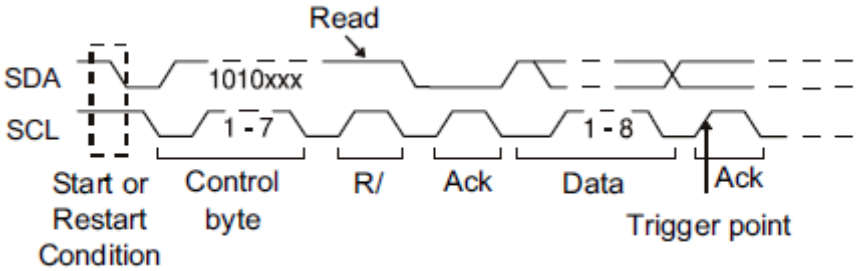
There are nine kinds of trigger conditions for I2C (Start, Stop, Restart, No Ack, EEPROM, 7 Addr&Data, 10 Addr&Data and Data Length).

Trigger Conditions

- **Start Condition**— the oscilloscope will be triggered when SDA signal transitions from high to low while the SCL clock is high. If it is chosen as the trigger condition (including frame triggers), a restart will be treated as a "Start condition".
- **Stop Condition**— the oscilloscope will be triggered when SDA transitions from low to high while the SCL is high.

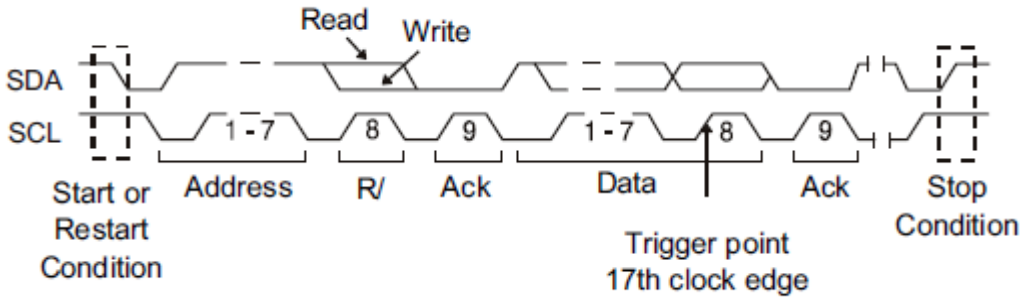


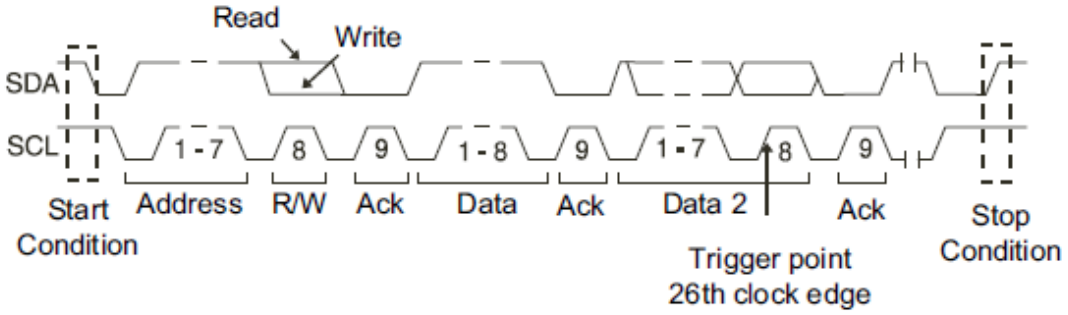
- **Restart**— the oscilloscope will be triggered when another "Start condition" occurs before a "Stop condition".
- **No Ack**— the oscilloscope will be triggered when SDA data is high during any SCL's ACK bit.
- **EEPROM** — the trigger searches for EEPROM control byte (the value is 1010xxx) on the SDA bus. And there is a Read bit and an ACK bit behind EEPROM. Use the **Limit Range** F key to set the qualifier and the **Data1** F key to set the data's value. If EEPROM's data is greater (less, equal) than Data1, the oscilloscope will be triggered at the edge of ACK bit behind Data byte. The Data bytes don't have to follow the EEPROM.



- **7 Address & Data** — the oscilloscope will be triggered when the following conditions are satisfied.
  - The address's length must be 7 bits and the address's value is the same as set value.
  - If you have set either Data1's or Data2's value, and the signal contains that value. If you have set both Data1's and Data2's value, the signal must have two consecutive data, with the first being Data1 and the second being Data2.

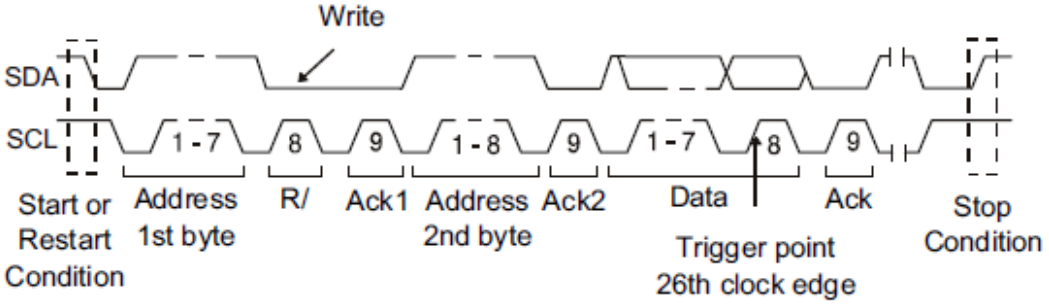
  (**Note**: If the data's value is 0xXX, any data value will be matched)





- **10 Address & Data** — the oscilloscope will be triggered when the following conditions are satisfied.
  - The address's length must be 10 bits and the address's value is the same as set value.
  - If you have set either Data1's or Data2's value, and the signal includes that value. If you have set both Data1's and Data2's value, the signal must have two consecutive data, the first having Data1's value, and the second matching Data2.

  (**Note**: If the set value is 0xXX, any data value will be matched)



- **Data Length** — When SDA data's length is equal to the value of Byte Length and address's length is the same as set value, the oscilloscope will be triggered. Byte length is in the range of 1 to 12 bits.

Operation Steps

1. Press **Trigger Setup** to enter the **TRIGGER** function menu.
2. Press the **Type** F keyand select **Serial**.
3. Press the **Protocol** F key and select **I2C**.
4. Press the **Trigger Setting** F key.

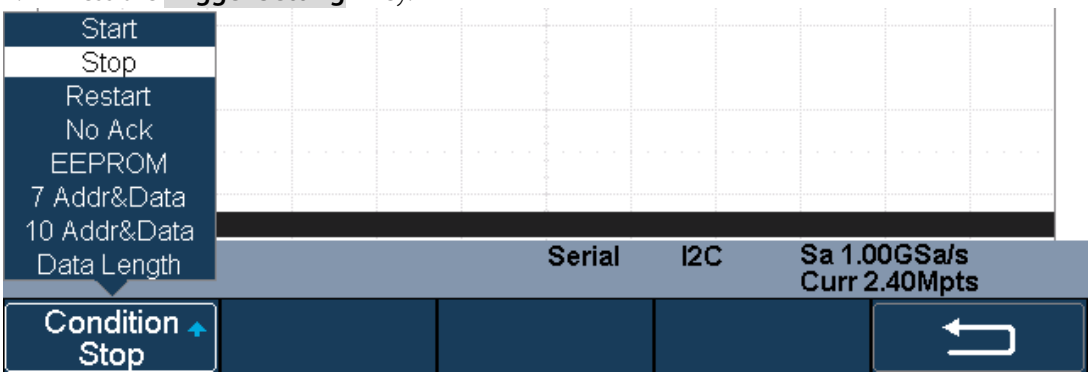


Figure 40 I2C TRIGGER Menu

5. Press the **Condition** F key and turn the **Universal Knob** to select the trigger condition:

- If you select the **EEPROM** condition:
  a. Press the **Limit Range** F key to set the qualifier (=, < or **>**).
  b. Press the **Data1** F key and set its value by turning the **Universal Knob**.
- If you select **7 Addr & Data** or **10 Addr & Data** condition:
  a. Press the **Addr** F key and turn the **Universal Knob** to select the 7- bit or 10- bit device address.
  b. Press the **Data1** or **Data2** F key and set their value(s).
  c. Press the **R/W bit** F key and select write-frame or read-frame to trigger the oscilloscope.

  (**Tips**: If device address is 7-bit, the value of address is in range of 0x00 to 0x7F. If device's address is 10-bit, the value of address is in range of 0x00 to 0x3FF.)
- If you select the **Data Length** condition:
  a. Press the **Address** to set the SDA address length 7bit or 10 bit.
  b. Press the **Byte Length** F key and set the byte length by **Universal Knob**. The range of the Byte Length is 1 to 12.

I2C Serial Decode

After completing the setup of I2C signal and trigger, we will decode I2C signals as follows:

1. Press **Shift** and **Print**→ **Decode**. Select one of the options from the **Decode1** and **Decode2.**
2. Press **Display** and select **On** to display the result of decoding.
3. Press **Configure** to turn on or off the read or write bit.
4. Press **List** to enter the **LIST** function menu.
   - Press **Display** and choose the same options as the first step.
   - Press **Scroll** and turn the **Universal Knob** to view all frames.
   - Press **Lines** and set the number of lines by **Universal Knob**. The range of the lines is 1 to 7.
   - When a packet of data is long, not all decoded data are displayed. Turn on the **Long Data** to display the complete data.
   - Users can export the result list of the package to a USB storage device in csv format. Saving the data (*.csv) is similar to the operation of setup files; see the section "External save and recall" for details.
5. Press **Format** to change the character encoding format of the decoding's result.
6. Press **Copy Setting** to enter the **COPY** function menu to synchronize the corresponding bus and trigger configurations.
7. Press **Tips Info** to turn on or off the decoding limit prompt. When the decoding frame number reaches the maximum, it will give an alert: "Decoding to maximum frame number limitation!"

Interpreting I2C Decode

Decode Result Frames:

- The address values are displayed at the beginning of a frame. The write address is displayed in green, and the read address in yellow.
- W/R bit is represented by (W) and (R), following the address value.
- The data value is displayed in white.
- "~A" after a data or address bits indicates no acknowledgement. For example, DB~A.
- Indicates there is not enough space on the display to show the complete content of a frame, and some content is hidden.

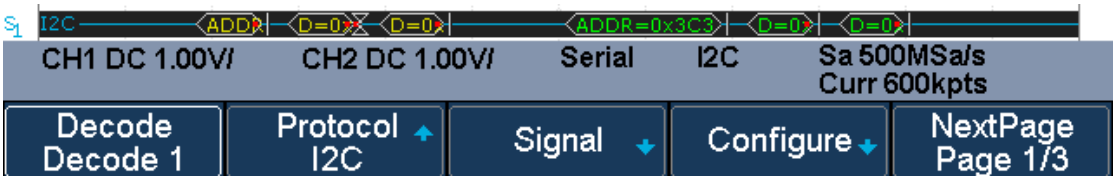


Figure 41 I2C Decode Bus Display

Decode Result Lists:

- Time — the horizontal displacement between current frame and trigger position.
- Address — the address of a frame.

- R/W — the type of a frame (write or read).
- Data — the value of the data.

I2C	Time	Address	R/W	Data(~A: no ack)
1	-2.52629ms	0x50	R	0xB0 C1~A
2	-2.11012ms	0x3C3	W	0xD2 E3
3	-1.52629ms	0x50	R	0xB0 C1~A
4	-1.11012ms	0x3C3	W	0xD2 E3
5	-526.288us	0x50	R	0xB0 C1~A
6	-110.118us	0x3C3	W	0xD2 E3
7	473.714us	0x50	R	0xB0 C1~A

Figure 42 I2C Decode List Display

SPI Trigger and Serial Decode

Setup for SPI Signals

Setting the SPI (Serial Peripheral Interface) signal includes two steps: connecting the CLK, MISO, MOSI and CS signals to the oscilloscope, and specifying the parameters of each input signal.

1. Press the **Shift** and **Print** to enter the **DECODE** function menu.
2. Press the **Decode** F key and select the desired slot (Decode1 or Decode2).
3. Press the **Protocol** F key and then select **SPI** by turning **Universal Knob**.
4. Press the **Signal** F key to enter the **SIGNAL** menu as Figure 43 shows.

Figure 43 SPI SIGNAL Menu

5. Set CLK (clock signal):
   a. Press the **CLK** F key to enter **CLK** menu.
   b. Press the **CLK** F key to select the channel that is connected to the SPI clock signal.
   c. Press the **Threshold** F key to set the SPI clock signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.
   d. Press the **Edge Select** F key to set the oscilloscope will samples at clock signal's rising edge or falling edge.
   e. Press the F key to return to the previous menu.

Figure 44 SPI CLK Menu

6. Set MISO:
   a. Press the **MISO** F key to enter the MISO menu.
   b. Press the **MISO** F key to select the channel that is connected to the SPI MISO signal.
   c. Press the **Threshold** F key to set the SPI MISO signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.
   d. Press the F key to return previous menu.

Figure 45 MISO Menu

7. Set MOSI:
   a. Press the **MOSI** F key to enter the MOSI menu.
   b. Press the **MOSI** F key to select the channel that is connected to the SPI MOSI signal.
   c. Press the **Threshold** F key to set the SPI MOSI signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.
   d. Press the F key to return previous menu.

Figure 46 MOSI Menu

8. Set CS:
   a. Press the **CS** F key to enter the MOSI menu.
   b. Press the **CS Type** F key to select the chip select type.
   c. Modify the Cs type's value.
   d. Press the F key to return previous menu.

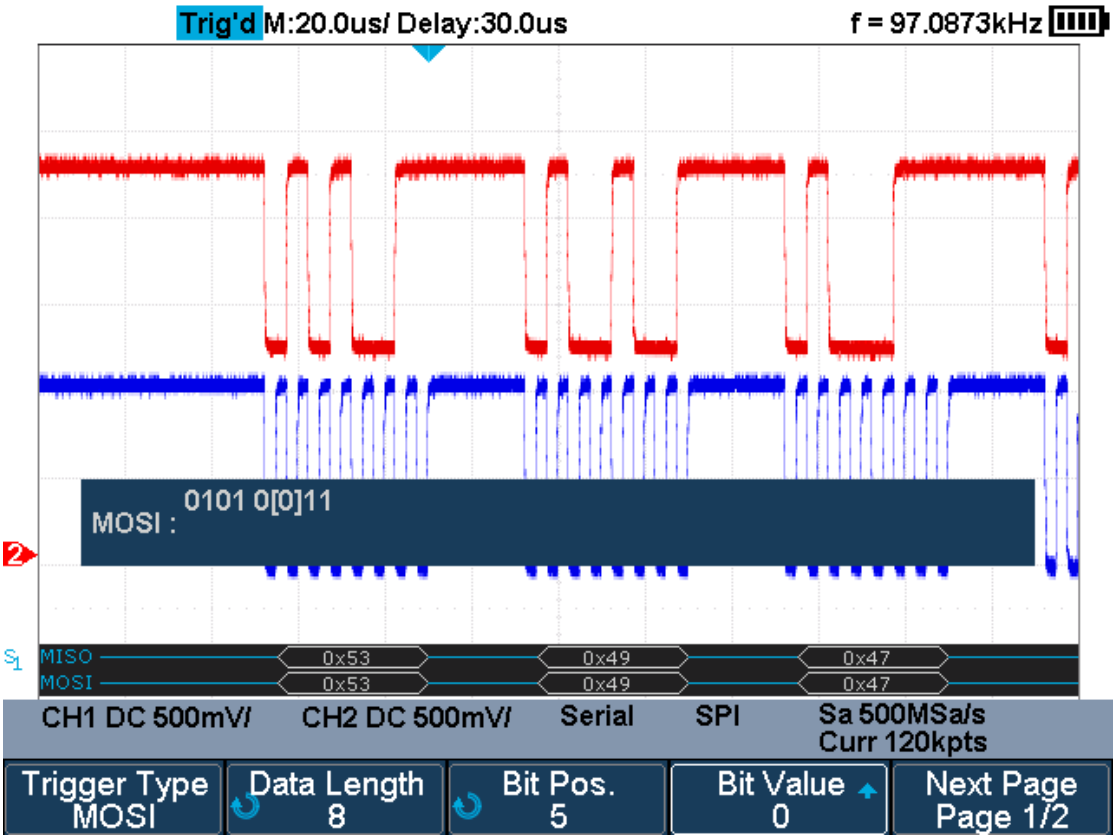
Function Menu	Settings	Explanation
CS Type	~CS	low voltage level of CS signal is available
	CS	high voltage level of CS signal is available
	CLK Timeout	If the time between two edges of clock signal is less than (or equal to) the timeout value, the signal between the two edges is treated as a frame. The range of clock timeout is 100ns-5ms. This setting is suitable for case where CS signal is not connected, or the number of oscilloscope channels is insufficient.

Table 1 Menu Explanations of the CS Type Parameters

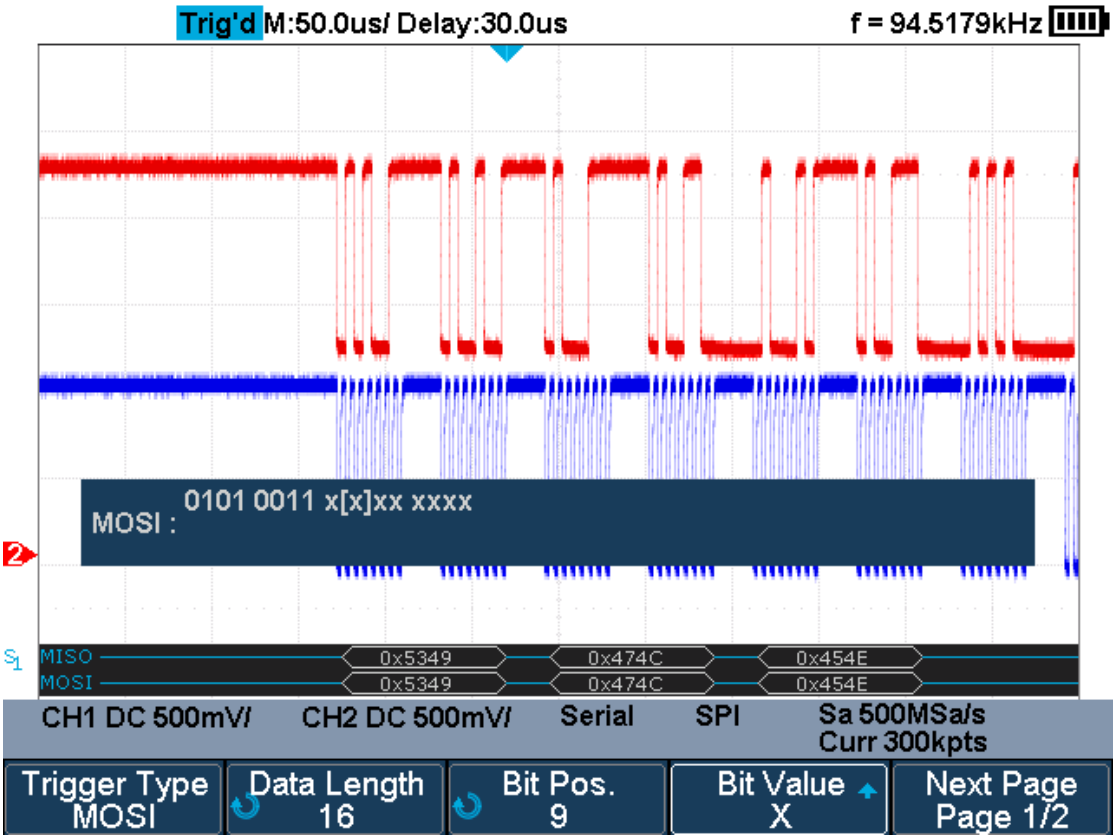
Example:

Connect the data, CLK signals of a SPI bus respectively to C1, C2. Data width = 8-bit, Bit order = MSB, and 12 bytes are transmitted in one frame.

In the SPI trigger signal menu, set the source and threshold of CLK, MISO signals. Set the CS type to Clock Timeout, the clock idle time between frames is T3, the clock period is T1, then set the timeout to a value between T1 and T3：



If the data width is set to be greater than 8 bits, the clock idle time between 8-bit data packets T2, then set the timeout time to a value between T1/2+T2 and T3.


Trig'd M:50.0us/ Delay:30.0us
f = 94.5179kHz
MOSI : 0101 0011 x[x]xx xxxx
MISO
MOSI
0x5349
0x474C
0x454E
CH1 DC 500mV/
CH2 DC 500mV/
Serial
SPI
Sa 500MSa/s
Curr 300kpts
Trigger Type
MOSI
Data Length
16
Bit Pos.
9
Bit Value
X
Next Page
Page 1/2

SPI Trigger

1. Press the **Trigger Setup** button to enter the **TRIGGER** function menu.
2. Press the **Type** F key and select **Serial**.
3. Press the **Protocol** F key and select **SPI**.
4. Press the **Trigger Setting** F key.

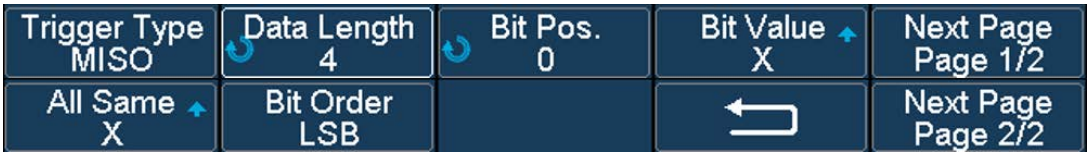


Figure 47 SPI TRIG SET Menu

5. Press the **Trigger Type** F key to select the trigger condition.

Function Menu	Settings	Explanation
Trigger Type	MISO	Master-In, Slave-Out
	MOSI	Master-Out, Slave-In

Table 2 Menu Explanations of the SPI trigger type

6. Press the Data Length F key, and turn the **Universal Knob** to set the length of a data. The range of data length is 4 to 96 bits.
7. Set the value of the trigger data.
   - Set the value of a bit:
     a. Press the **Bit Roll** F key to select a bit in the data.
     b. Press the **Bit Value** F key to set the value of the selected bit.
   - Set the value of all bits:
     a. Press the **All Same** F key to set the value of all bits.

Function Menu	Settings	Explanation
Bit Value	0	High voltage level
	1	Low voltage level
	X	Voltage level unimportant

Table 3 Menu Explanations of the SPI Bit value

8. Press the **Next Page** F key.
9. Press the **Bit Order** F key to set the bit order (MSB or LSB).

SPI Serial Decode

After completing the setup of SPI signal and trigger, we will decode SPI signals as follows.

1. Press **Shift** and **Print** → **Decode**. Select one of the options from the **Decode1** and **Decode2.**
2. Press the Configure F key to set the bit stream format and bits (4-32 bits)
3. Press the **Display** F key and select **On** to display the result of decoding.
4. Press the **Configure** F key to set bit order and data length.
5. Press the **List** F key to enter the **LIST** function menu.
   - Press the **Display** F key and choose the same options as the first step.
   - Press the **Scroll** F key and turn the **Universal Knob** to view all frames.
   - Press the **Lines** F key and set the number of lines by **Universal Knob**. The range of the lines is 1 to 7.
   - Users can export the result list of the package to a USB storage device in csv format. Saving the data (*.csv) is similar to the section "External save and recall".
6. Press the **Format** F key to change the character encoding format of the decoding's result.
7. Press the **Copy Setting** F key to enter the **COPY** function menu, to synchronize the corresponding bus configuration and trigger configuration.
8. Press the **Tips Info** F key to turn on or off the decoding limit prompt. When the decoding frame number reaches the maximum, it will give an alert: "Decoding to maximum frame number limitation!"

Interpreting SPI Decode

Decode Result Frames:

- The data values are displayed in frames and are shown in white. The scope supports 4~96 bit data display.
- MISO — the decoding result of "Master-In, Slave-Out" line.
- MOSI —the decoding result of "Master-Out, Slave-In" line.
- Indicates there is not enough space on the display to show the complete content of a frame, and some content is hidden.

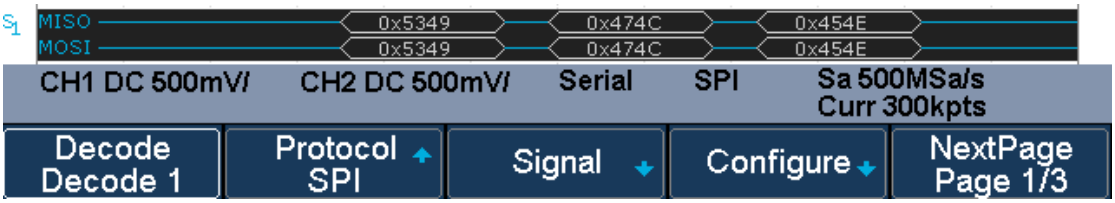


Figure 48 SPI Decode Bus Display

Decode Result Lists:

- Time — the horizontal displacement between current frame and trigger position.
- MISO — the decoding result of "Master-In, Slave-Out" line.
- MOSI —the decoding result of "Master-Out, Slave-In" line.

SPI	Time	MISO	MOSI
1	-35.0420us	0x5349	0x5349
2	84.9840us	0x474C	0x474C
3	204.950us	0x454E	0x454E
4	324.944us	0x545F	0x545F
5	444.910us	0x0000	0x0000
6	564.964us	0x0250	0x0250

Figure 49 SPI Decode List Display

UART Trigger and Serial Decode

Setup for UART Signals

1. Press the **Shift** and **Print** buttons to enter the **DECODE** function menu.
2. Press the **Decode** F key and select the desired slot (Decode1 or Decode2).
3. Press the **Protocol** F key and then select **UART** by turning **Universal Knob**.
4. Press the **Signal** F key to enter the **SIGNAL** menu as below shows.

Figure 50 UART SIGNAL Menu

5. Set RX:
   a. Press the **RX** to select the channel that is connected to the RX signal.
   b. Press the first **Threshold** key to set the RX signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.
6. Set TX:
   a. Press the **TX** to select the channel that is connected to the TX signal.
   c. Press the second **Threshold** key to set the TX signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.
7. Press the F key to return to the previous menu.
8. Press the **Configure** F key to enter the **BUS CONFIG** menu.

Figure 51 BUS CONFIG Menu

9. Press the **Baud** F key to set baud rate.
   - The baud rate can be set as predefined value.
   - If the desired baud rate is not listed, press the **Baud** and select **custom** option, press the **Custom** and turn the **Universal Knob** to set the desired baud rate.
10. Press the **Data Length** F key and set byte bits (5-8) by **Universal Knob**.
11. Press the **Parity Check** F key to set the type of parity check (**Even, Odd, Mark, Space** or **None**).
12. Press the **Stop Bit** F key to set the length of stop bit (**1, 1.5** or **2 bits**).
13. Press the **Next Page** F key.
14. Press the **Bit Order** F key to select the bit order (**LSB** or **MSB**).
15. Press the **Idle Level** F key to set the idle level (**LOW** or **HIGH**).

UART Trigger

1. Press **Trigger Setup** to enter the **TRIGGER** function menu.
2. Press **Type** and select **Serial**.
3. Press **Protocol** and select **UART**.
4. Press **Trigger Setting** to enter **UART TRIG SET** menu.

Figure 52 UART TRIG SET Menu

5. Press **Source Type** to select the source of trigger (RX or TX).
6. Press **Condition** and set up the desired trigger condition:
   - **Start** — the oscilloscope will be triggered at the position of start bit.
   - **Stop** —the oscilloscope will be triggered at the position of stop bits.
   - **Data** — the oscilloscope will be triggered when a byte is found equal to (greater or less than) the specified data.
     a. Press the **Compare Type** F key and choose an equality qualifier (**>**, **<** or **=**).
     b. Press the **Value** F key to set data's value. Data's value is in range of 0x00 to 0xff.
   - **ERROR** — if the parity check has been set, and the bit of parity check is error, the oscilloscope will be triggered.

UART Serial Decode

After completing the setup of UART signal and trigger, we will decode UART signals.

1. Press **Shift** and **Print** → **Decode**. Select one of the options from the **Decode1** and **Decode2.**
2. Press the **Display** and select **On** to display the result of decoding.
3. Press the **Configure** to set the baud rate, data length, parity check, stop bit, idle level and bit order. Refer to "Setup for UART Signals" to set parameters.
4. Press the **List** to enter the **LIST** function menu.
   - Press the **Display** and choose the same options as the first step.
   - Press the **Scroll** and turn the **Universal Knob** to view all frames.
   - Press the **Lines** and set the number of lines by **Universal Knob**. The range of the lines is 1 to 7.
   - Users can export the result list of the decode to a USB storage device in csv format. See the section "External save and recall" for details.
5. Press the **Format** to change the character encoding format of the decoding's result.
6. Press the **Copy Setting** to enter the **COPY** function menu to synchronize the corresponding bus configuration and trigger configuration.
7. Press the **Tips Info** to turn on or off the decoding limit prompt. When the decoding frame number reaches the maximum, it will pop up "Decoding to maximum frame number limitation!"

Interpreting UART Decode

Decoding Result Frames:

- RX — the decoding result of the data received.
- TX — the decoding result of the data transmitted.
- Indicates there is not enough space on the display to show the complete content of a frame, and some content is hidden.

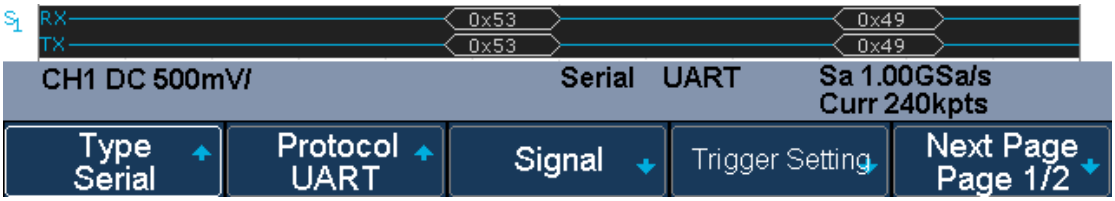


Figure 53 UART Decode Bus Display

Decoding Result Lists:

- Time— the horizontal displacement between current frame and trigger position.
- RX — the receiving channel.
- TX — the transmitting channel.

- RX Err— Parity error or unknown error in the data received.
- TX Err— Parity error or unknown error in the data transmitted.

UART	Time	RX	RX Err	TX	TX Err
1	-24.1920us	0x53			
2	-24.1920us			0x53	
3	65.4360us	0x49			
4	65.4360us			0x49	
5	154.975us	0x47			
6	154.975us			0x47	
7	244.575us	0x4C	Parity Err		

Figure 54 UART Decode List Display

CAN Trigger and Serial Decode

Setup for CAN Signals

1. Press **Shift** and **Print** to enter the **DECODE** function menu.
2. Press the **Decode** F key and select the desired slot (Decode1 or Decode2).
3. Press the **Protocol** F key and then select **CAN** by turning **Universal Knob**.
4. Press the **Signal** F key to enter the **SIGNAL** menu as below shows.

Figure 55CAN SIGNAL Menu

   a. Press the **Source** F key to select the channel that is connected to the CAN signal.
   d. Press the **Threshold** key to set the CAN signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.
5. Press the **Configure** F key to enter the **BUS CONFIG** menu.
6. Press the **Baud** to set baud rate by **Universal Knob**.
   - The baud rate can be set as predefined value (from 5kb/s to 1Mb/s) or custom value (from 5kb/s to 1Mb/s).
   - If the desired baud rate is not listed, press **Baud** and select **custom** option, press the **Custom** and turn the **Universal Knob** to set the desired baud rate.

CAN Trigger

Trigger Conditions

- **Start**— the oscilloscope will be triggered at the start bit of a frame.
- **Remote** — the oscilloscope will be triggered by a remote frame with specified ID.
- **ID** — the oscilloscope will be triggered by a frame with specified ID.
- **ID+DATA**— the oscilloscope will be triggered by data frame that have specified ID and data.
- **Error** — the oscilloscope will be triggered by an error frame.

Operation Steps

1. Press the **Trigger Setup** to enter the **TRIGGER** function menu.
2. Press the **Type** and select **Serial**.
3. Press **Protocol** and select **CAN**.
4. Press **Trigger Setting** to enter the **CAN TRIG SET** menu.
5. Press **Condition** and select the trigger condition by **Universal Knob**:
   - If you select the **REMOTE** and **ID** condition:
     a. Press **ID Bits** to set the length of ID (11bits or 29 bits).
     b. Press **Curr ID Byte** and use the **Universal Knob** to select the byte that you want to set.
     c. Press **ID** and set the ID's value by the **Universal Knob**.

     (**Tips**: For convenience, the ID is split into several bytes. For example, if the ID's length is 11 bits, it will be split into two bytes, a byte includes 8 bits. If "1st byte" is selected, only the 8 least significant bits can be changed.)
   - If you select the **ID+DATA** condition:
     a. Press the **ID bits** F key to select the ID's length (11 or 29 bits).
     b. Press the **Curr ID Byte** F key and use **Universal Knob** to select the byte that you want to modify.
     c. Press the **ID** F key and set the ID's value by **Universal Knob**.
     d. Press the **Data** F key and set the value of the first byte by **Universal Knob**.

CAN Serial Decode

1. Press **Shift** and **Print** → **Decode**. Select one of the options from the **Decode1** and **Decode2.**
2. Press the **Display** and select **On** to display the result of decoding.
3. Press the **Configure** to set the baud rate.
4. Press the **List** to enter the **LIST** function menu.
   - Press the **Display** and choose the same options as the first step.
   - Press the **Scroll** and turn the **Universal Knob** to view all frames.
   - Press the **Lines** and set the number of lines by **Universal Knob**. The range of the lines is 1 to 7.
   - Users can export the result list of the package to a USB storage device in csv format. See the section "External save and recall" for details.
5. Press the **Format** to change the character encoding format of the decoding's result.
6. Press the **Copy Setting** to enter the **COPY** function menu to synchronize the corresponding bus configuration and trigger configuration.
7. Press the **Tips Info** to turn on or off the decoding limit prompt. When the decoding frame number reaches the maximum, it will give an alert: "Decoding to maximum frame number limitation!"

Interpreting CAN Decode.

Decoding Result Frames:

- Arbitration field is displayed in frame
- Control field is displayed in frame
- Data field is displayed in frame
- CRC field is displayed in frame
- Indicates there is not enough space on the display to show complete content of a frame and some content is hidden.

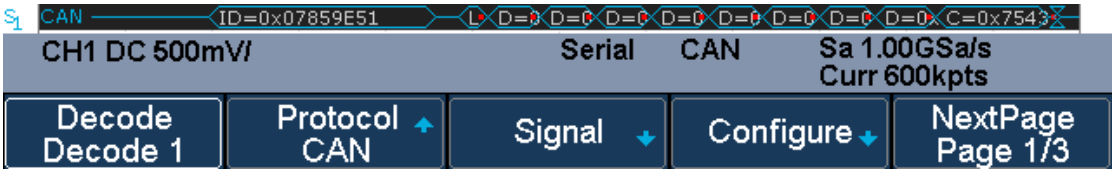


Figure 56 CAN Decode Bus Display

Decoding Result List:

- Time— the horizontal displacement between current frame and trigger position.
- Type — the type of frames, "D" represents data frame, "R" represents remote frame.
- ID — the id of frames, the oscilloscope can automatically detect the length of frame's id (11 bits or 27 bits).
- Length — the length of data field.
- Data — the value of data field.

- CRC — the value of CRC (Cyclic Redundancy Check) field.
- ACK — Acknowledgment bit.

CAN	Time	Type	ID	Length	Data	CRC	ACK
1	-1.00906ms	Ext	0x07819F51	0x08	0x53 49 47 4C 45 4E 54 5F	0x7541	yes
2	-9.05500us	Ext	0x07819F51	0x08	0x53 49 47 4C 45 4E 54 5F	0x7541	yes
3	990.946us	Ext	0x07819F51	0x08	0x53		

Figure 57 CAN Decode List Displa

LIN Trigger and Serial Decode

Setup for LIN Signals

There are two steps of setting the LIN signal, connecting the signal to oscilloscope, and specifying each input signal's parameters.

1. Press the **Shift** and **Print** to enter the DECODE function menu.
2. Press the **Decode** F key and select the desired slot (Decode1 or Decode2).
3. Press the **Protocol** F key and then select **LIN** by turning **Universal Knob**.
4. Press the **Signal** F key to enter the **SIGNAL** menu as below shows.

Figure 58 LIN SIGNAL Menu

5. Press the **Source** F key to select the channel that is connected to the LIN signal.
   e. Press the **Threshold** F key and set the LIN signal's threshold voltage level by **Universal Knob**. The threshold voltage level is for decoding, and it will be treated as the trigger voltage level when you set the trigger type to serial.
6. Press the F key to return previous menu.
7. Press the **Configure** F key to enter the **BUS CONFIG** menu.
8. Press the **Baud** F key to set baud rate.
   - The baud rate can be set as predefined value.
   - If the desired baud rate is not listed, select **Custom** option, press the **Custom** and turn the **Universal Knob** to set the desired baud rate.

LIN Trigger

Trigger Conditions

- **Break** — the oscilloscope will be triggered at the position of break field's break delimiter.
- **ID** (Frame ID) — the oscilloscope will be triggered at the position of identifier field's stop bit, if the value of a frame's ID is equal to specified value.
  (**Note**: If the data's value is 0xXX, any data value will be matched)
- **ID + Data** (Frame ID and Data) — the oscilloscope triggers when a frame with an ID and data equal to the selected values is detected. Use the **Universal Knob** to select the value for the ID, Data1 and Data2.
  a. The ID's value is the same as the set value.
  b. If you have set either Data1's or Data2's value, and the signal contains that value. If you have set both Data1's and Data2's value, the signal must include both data, the first being Data1, and the second Data2.
  (**Note**: If the data's value is 0xXX, any data value will be matched)
  - **Data Error** —the oscilloscope will be triggered when errors (ID check, checksum, and sync byte field errors) are detected.

Operation

1. Press **Trigger Setup** to enter the **TRIGGER** function menu.
2. Press **Type** and select **Serial**.
3. Press **Protocol** and select **I2C**.
4. Press **Trigger Setting** to enter **LIN TRIG SET** menu.
5. Press **Condition** and select the trigger condition by **Universal Knob**:
   - If you select **ID** condition**:**
     a. Press the **ID** F key and set its value by turning the **Universal Knob**.
   - If you select **ID+DATA** condition**:**
     a. Press the **ID** F key and set its value by turning the **Universal Knob**.
     b. Press the **DATA1** F key and set its value by turning the **Universal Knob**.
     c. Press the **DATA2** F key and set its value by turning the **Universal Knob**.

LIN Serial Decode

After completing the setup of LIN signal and trigger, we will decode LIN signals.

1. Press **Shift** and **Print** → **Decode**. Select one of the options from the **Decode1** and **Decode2.**
2. Press **Display** and select **On** to display the result of decoding.
3. Press **Configure** to set the baud rate.
4. Press **List** to enter the **LIST** function menu.

- Press **Display** and choose the same options as the first step.
- Press **Scroll** and turn the **Universal Knob** to view all frames.
- Press **Lines** and set the number of lines by **Universal Knob**. The range of the lines is 1 to 7.
- Users can export the result list of the package to the external USB storage device in csv format. See the section "External save and recall" for details.

5. Press **Format** to change the character encoding format of the decoding's result.
6. Press **Copy Setting** to enter the **COPY** function menu to synchronize the corresponding bus configuration and trigger configuration.
7. Press **Tips Info** to turn on or off the decoding limit prompt. When the decoding frame number reaches the maximum, it will give an alert: "Decoding to maximum frame number limitation!"

Interpreting LIN Decode

Decoding Result Frame:

- Protected Identifier Field is displayed in frame
- Data Length is displayed in frame
- Data Field is displayed in frame.
- Checksum Field is displayed in frame.
- Indicates there is not enough space on the display to show complete content of a frame and some content is hidden.

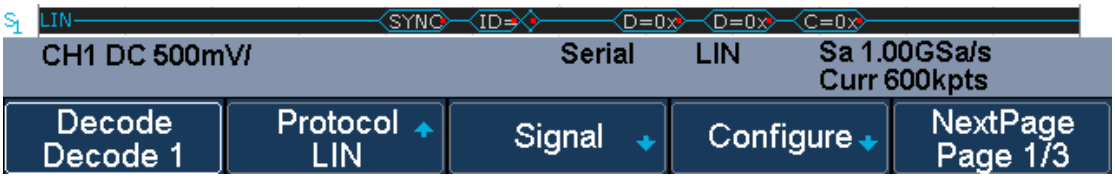


Figure 59 LIN Decode Bus Display

Decoding Result List:

- Time— the horizontal displacement between current frame and trigger position.
- ID — the value of frame's Protected Identifier Field.
- Data Length — the length of Data Field.
- ID Parity — the two check bits of Protected Identifier Field.
- Data — the value of Data Field.
- Checksum— the value of Checksum Field.

LIN	Time	ID	Data Length	ID Parity	Data	Checksum
1	-2.07519ms	0x06	2	0	0x54 5F	0x46
2	-1.07519ms	0x06	2	0	0x54 5F	0x46
3	-75.1870us	0x06	2	0	0x54 5F	0x46
4	924.814us	0x06	2	0	0x54 5F	0x46
5	1.92481ms	0x06	2	0	0x54 5F	0x46

Figure 60 LIN Decode List Display

Reference Waveform

The oscilloscope can save a channel waveform or math waveform to a reference (REF) waveform position in the oscilloscope. Then, a REF waveform can be displayed and compared against other waveforms. All REF waveforms can be displayed at a time.

Contents:

- Save REF Waveform to Internal Memory
- Display REF Waveform
- Adjust REF Waveform
- Clear REF Waveform

Save REF Waveform to Internal Memory

To save a REF waveform to internal memory:

1. Press **Shift** and **Hide Menu** on the front panel to enter the REF WAVE function menu. Note that when the horizontal format is in X-Y mode, REF cannot be enabled.
2. Press the **Source** F key; then, turn the **Universal Knob** to select the source of reference channel. The source includes analog channel and math waveforms.
3. Press the **Location** F key; then, turn the **Universal Knob** to select the position to save the REF waveform. The source includes analog channel and math waveforms.
4. Press the **Save** F key to save the channel or math waveform to the chosen location. The vertical scale information and the vertical offset of the waveform will be saved as well. You will get an alert: "**Saved to internal file REF***" when the waveform has been saved successfully.

Note: The REF waveforms are non-volatile. The REF waveform can still be saved after restarts or default operation.

Display REF Waveform

To display a REF waveform:

1. Press **Shift** and **Hide Menu** on the front panel to enter the REF WAVE function menu.
2. Press the **Location** F key; then, turn the **Universal Knob** to select the REF waveform that you want to display.
3. Press the **Display** F key to select **On** and display the REF waveform on the screen. The oscilloscope can display all reference waveforms at the same time.

Adjust REF Waveform

1. Press the **Scale** and **Position** F key and turn the **Universal Knob** to adjust the vertical scale and position of the reference waveform. The vertical scale and position information display at the middle of the screen.

The initial values display at the middle of the screen is the setup when the reference waveform is saved.

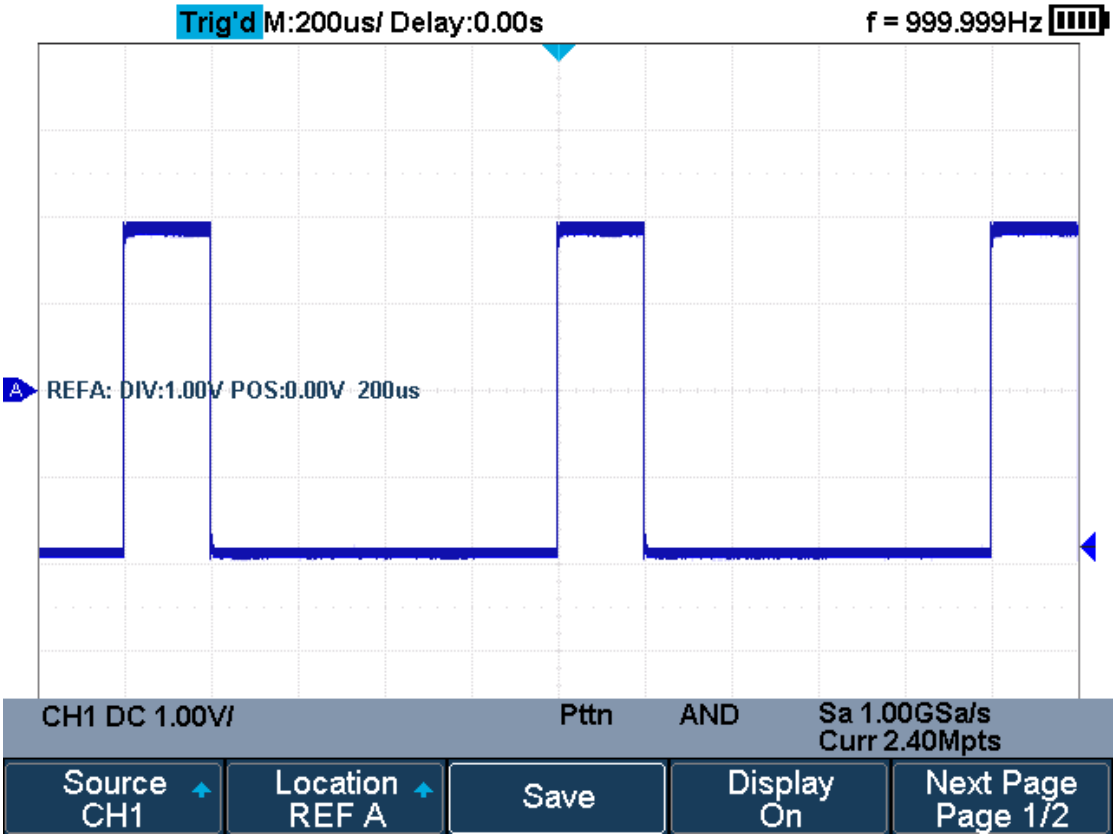


Figure 61 Reference Waveform

Clear REF Waveform

The oscilloscope does not have a "Clear" option under the REF WAVE function menu. To clear the appointed reference waveform, you can save a new reference waveform to the same location to cover it. Or press **Shift** + **Cursors** → **Recall** → **Type** and select **Security Erase** to clear the stored waveform.

Math

The oscilloscope supports many math operations between analog channels, including addition (+), subtraction (-), multiplication (*), division (/), FFT, differentiation (d/dt), integration (∫dt), and square root (√). The resulting math waveform is displayed in white and labeled with "M". You can use cursors to measure it.

The contents of this chapter:

- Units for Math Waveforms
- Math Operators

Note: if the analog channel or the math function is cut off (waveforms do not display on the screen completely), the resulting math will also be cut off.

Units for Math Waveforms

Use the channel function menu to set the unit of each channel to "**V**" or "**A**". The oscilloscope math operation includes units as below:

Math Operation	**Unit**
Addition (+) or subtraction (-)	V, A
multiplication (*)	V^2, A^2 or W
division (/)	None, V*A^-1 or V^-1*A
FFT	dBVrms, Vrms, dBm, dBArms, Arms
derivative (d/dt)	V*S^-1 or A*S^-1
integral (∫dt)	Wb, C
square root (√)	V^1/2 or A^1/2

Math Operators

The oscilloscope supports math count operations (addition, subtraction, multiplication, division), the FFT (Fourier transform) operation, and differentiation, integration, and square root.

Addition or Subtraction

Math operators perform arithmetic operations add or subtract operation on any two analog input channels. When you select addition or subtraction, the **Source A** and **Source B** values are added or subtracted point by point, and the result is displayed.

1. Press **Shift** and **Measure** on the front panel to enter the MATH function menu.
2. Press the **Source A** and **Source B** F keys respectively, and then turn the **Universal Knob** to select the source for math operation. Addition or Subtraction can be applied between analog channels.
3. Press the **Operator** F key and then turn the **Universal Knob** to select **+** or **-** to make addition or subtraction operation. The resulting math waveform is displayed in pink and labeled with "M".
4.

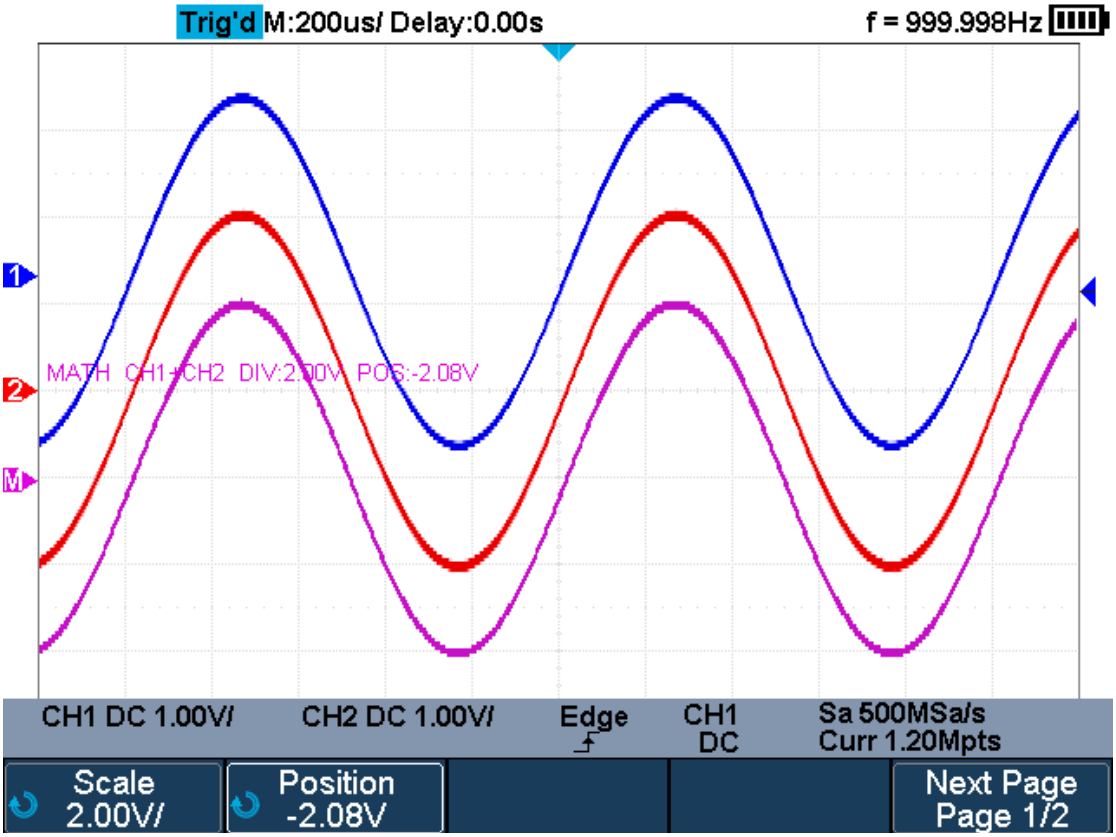


Figure 62 C1+C2 Waveform

If you want to invert the math waveform, press the **Invert** button and set the option to **On** to invert the display of the math waveform.

Multiplication and Division

When you select multiplication or division, the **Source A** and **Source B** values are multiplied or divided point by point and the result is displayed.

1. Press **Shift** and **Measure** on the front panel to enter the MATH function menu.
2. Press the **Source A** and **Source B** F keys, respectively, and then turn the **Universal Knob** to select the source for math operation. Multiplication or Division can be applied between analog channels.
3. Press the **Operator** F key and then turn the **Universal Knob** to select ***** or **/** to make multiplication or division operation. The resulting math waveform is displayed in pink and labeled with "M".

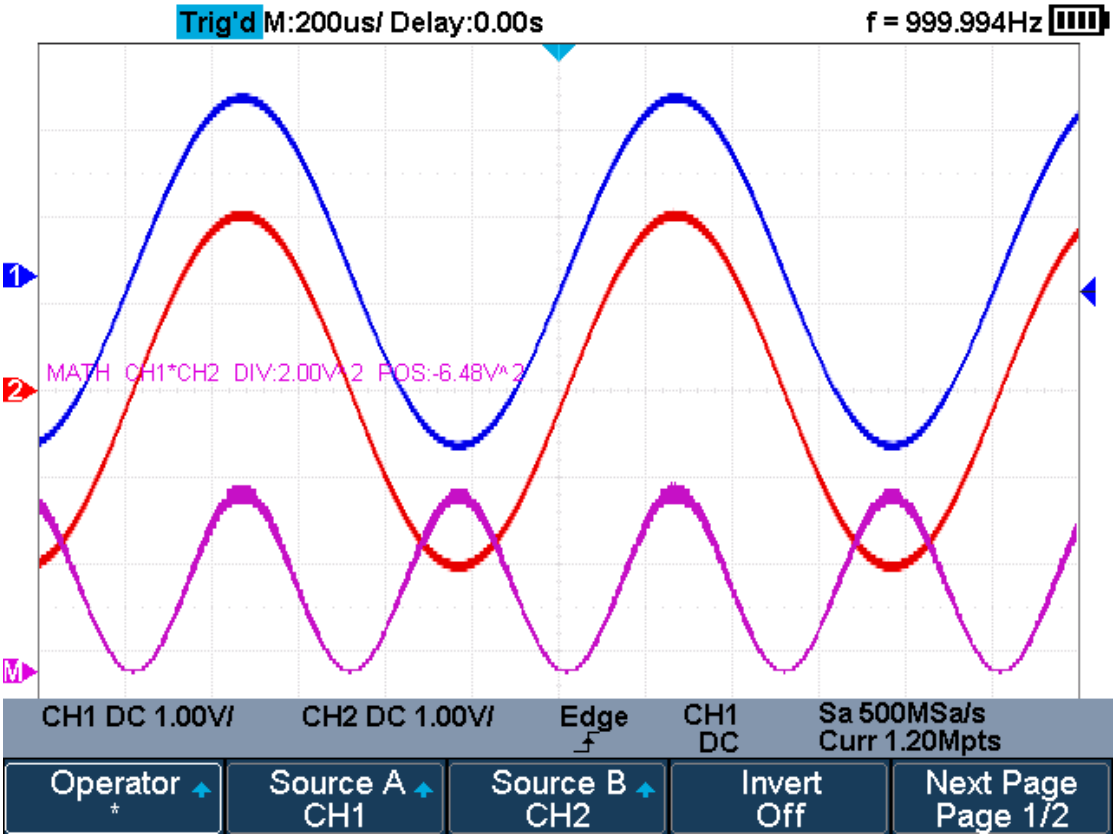


Figure 63 C1*C2 Waveform

4. If you want to invert the math waveform, press the **Invert** button and set the option to **On** to invert the display of the math waveform.

FFT

FFT is used to compute the fast Fourier transform of channel input. FFT takes the digitized time record of the specified source and transforms it to the frequency domain. When the FFT function is selected, the FFT spectrum is plotted on the oscilloscope display as magnitude in dBV versus frequency. The readout for the horizontal axis changes from time to frequency (Hertz) and the vertical readout changes from V to dB.

FFT operation is useful when:

- Measuring harmonic components and distortion in the device under test
- Measuring the characteristics of the noise in DC power
- Analyzing vibration

To display a FFT waveform:

1. Press **Shift** and **Measure** on the front panel to open the MATH function menu.
2. Press the **Operator** F key and then turn the **Universal Knob** to select **FFT**. The resulting math waveform is displayed in pink and labeled with "M".

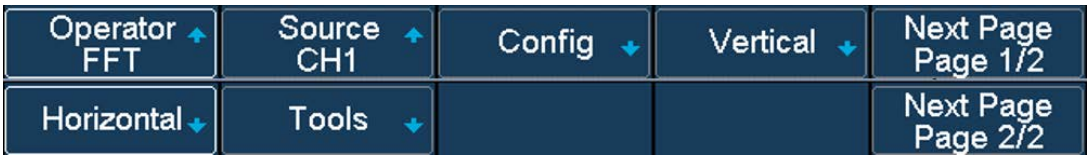


Figure 64 FFT Menu

3. Press the **Source** F key, and then turn the **Universal Knob** to select the source to do FFT operation. Analog channels can be used as the source.
4. Press the **Config** F key to enter CONFIG menu.

Figure 65 FFT CONFIG Menu

- Press the **Maximum points** F key, and then turn the **Universal Knob** to select the Maximum points.
- Press the **Window** F key, and then turn the **Universal Knob** to select an appropriate window. Spectral leakage can be considerably decreased when a window function is used. The oscilloscope provides five windows (Rectangle, Blackman, Hanning, Hamming and Flattop) which have different characteristics and are applicable to measure different waveforms. You need to select the window function according to different waveforms and their characteristics. Please read the table below carefully to choose an appropriate option according to the input signal.

Window	Applications and Characteristics
Rectangle	These are normally used when the signal is transient (completely contained in the time-domain window) or periodic with a fundamental frequency that is an integer multiple of the frequency of the window. Signals other than these types will show varying amounts of spectral leakage and scallop loss, which can be corrected by selecting another type of window.
Hanning	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Hamming	These reduce leakage and improve amplitude accuracy. However, frequency resolution is also reduced.
Flat Top	This window provides excellent amplitude accuracy with moderate reduction of leakage, but with reduced frequency resolution.
Blackman	It reduces the leakage to a minimum, but with reduced frequency resolution.

- Press the **Auto Set** F key to automatically set the appropriate parameters for the FFT measurement.
- Press the **Display** F key to select **Split Screen, Full Screen,** or **Exclusive** display mode.

  Split Screen: Time domain waveform and frequency domain waveform are displayed separately. The time domain waveform is on the upper half screen, while the frequency domain waveform is located within the lower half of the display. In Split mode, if Zoom is enabled, the zoom waveform and the frequency domain waveform are displayed on the lower half screen together.

  Full Screen: Time-domain waveform and frequency-domain waveform are displayed together.

  Exclusive: Only the frequency-domain waveform is displayed.
- Press the **Mode** F key to select **Normal**, **Max-Hold** or **Average**. When you select **Average**, it is necessary to set the average times.

5. Press the **Vertical** F key to enter VERTICAL menu.

Figure 66 VERTICAL Menu

- Press the **Scale** F key, and then turn the **Universal Knob** to select the desired vertical FFT scale
- Press the **Ref Level** F key, and then turn the **Universal Knob** to select the desired vertical FFT offset.
- Press the **Unit** F key to select the unit of vertical axis. The vertical axis units can be dBVrms, dBm, Vrms or dBArms, Arms, which use a logarithmic scale or a linear scale to display vertical amplitude respectively.
- Press the **Ext Lode** F key and then turn the **Universal Knob** to select the external load value.

6. Press the **Horizontal** F key to enter HORIZONTAL menu.

Figure 67 HORIZONTAL Menu

- Press the **Center** F key, and then turn the **Universal Knob** to select the desired center frequency.
- Press the **Span** F key, and then turn the **Universal Knob** to select the desired span frequency.

7. Press the **Tools** F key to enter TOOLS menu. Press **Type** F key to select the type of tools. The available tools are **Peaks**, **Markers**, and **Off**.

Peaks: Automatically mark the peak of the current FFT waveform according to the search configuration.

Figure 68 FFT Peaks Menu

- Press the **Show Table** F key to display a table of peaks, and press **Show Frequency** F key to display the peaks' frequencies. Press the **Sort By** F key repeatedly to set the table sorting by Amplitude or Frequency.
- Press the **Search Config** F key to set a search configuration.

Figure 69 SEARCH Menu

a. Press the **Threshold** F key and turn the **Universal knob** to set the minimum peak amplitude. Features can only be considered peaks if they are larger than the peak limit.
b. Press the **Excursion** F key and turn the **Universal knob** to set the difference between the peak value and the minimum Amplitude on both sides. Features can only be considered peaks if the difference peak-minimum differences are greater than the excursion setting.

Markers: Customize the marker locations on the FFT waveform based on the search configuration.

Figure 70 FFT Markers Menu

- Press **Markers Control** F key to enter MARKER Menu.

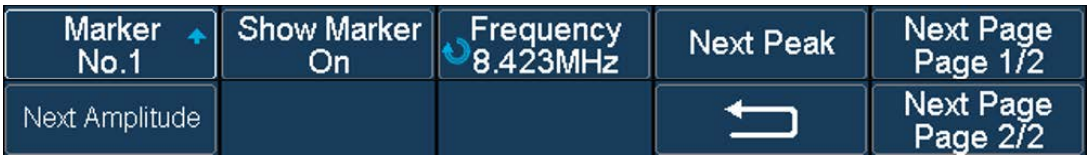


Figure 71 Marker Control Menu

a. Press the **Marker** F key repeatedly to select the marker number from NO.1~NO.8
b. Press the **Show Marker** F key repeatedly to turn on or off the selected mark.
c. Press the **Frequency** F key to set the frequency value of the selected marker.
d. Press the **Next Peak** F key to move the selected mark to the next peak and press the **Next Amplitude** F key to move the selected marker to the next peak with lower amplitude. (Up to 20 peaks are supported)

- Press the **Search Config** F key to enter the SEARCH menu. Similar to the **Search Config** for peaks.
- Press the **Markers on Peaks** F key to set the markers on peaks, and press the **Markers on Harmonics** F key to set the markers on harmonics.
- Press the **Show Table** F key to show the amplitude of the markers; press the **Show Frequency** F key to show the frequency value of the markers, and press the **Show Delta** F key to show the delta amplitude between markers.

The following figure shows the FFT waveform displayed on the split screen and the peak tool is turned on:

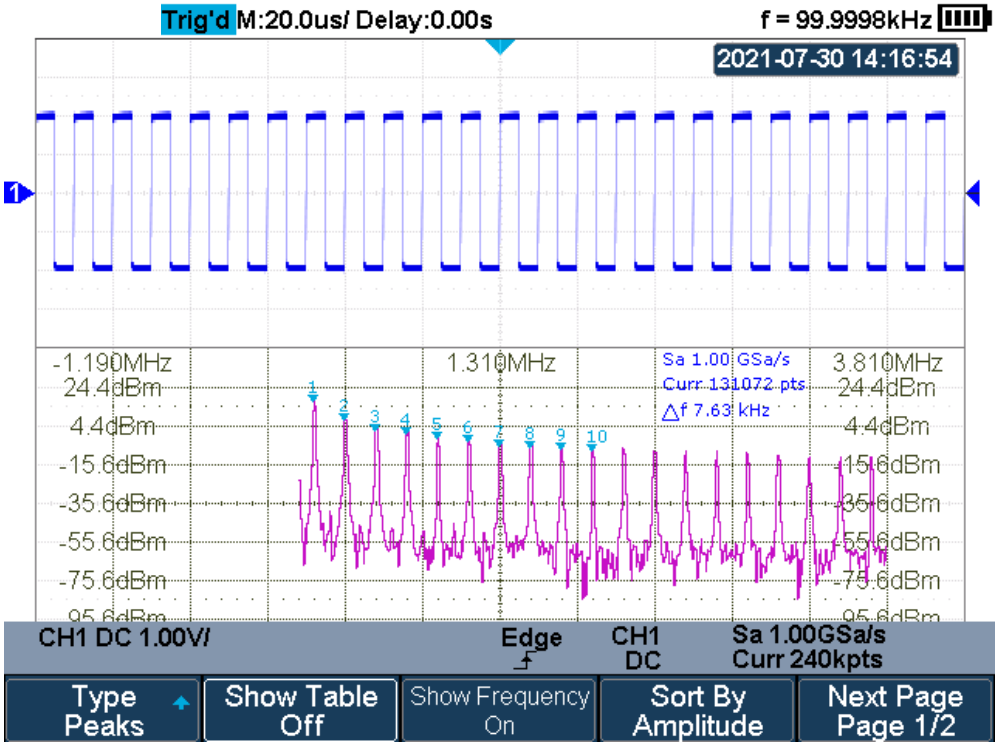


Figure 72 FFT Waveform

Note:

- Signals with DC components or deviation would cause error or deviation of the FFT waveform components. To reduce the DC components, set the Channel **Coupling** to AC.
- To reduce the random noise and aliasing frequency components of repetitive or single pulse, set the **Acquisition** of the oscilloscope to Average.

Measurement of an FFT:

To make cursor measurements, press the **Cursors** button, and then press the **Mode** F key to select **On** to turn the cursors. Use the X1 and X2 cursors to measure frequency values and the difference between two frequency values (ΔX). Use the Y1 and Y2 cursors to measure amplitude in dB and difference in amplitude (ΔY).
You can find the frequency value at the first occurrence of the waveform maximum by using the X at Max Y measurement.

Note: please refer to the cursors chapter to obtain the method of using cursors.

Math Function Operation

The oscilloscope can also calculate the signal's derivative (d/dt), integral (∫dt) and square root ($\sqrt{\ }$).

Differentiate

d/dt (differentiate) calculates the discrete time derivative (slope) of the selected source.

$$di = \frac{y(i+dx) - y(i)}{dx}$$

Where:

- d = differential waveform
- y = voltage value of data point
- i = data point index
- dx = point-to-point time difference

The **dx** option under d/dt math function operation menu indicates the number of data points to take the difference over. The minimum value is 4, and the unit is points.

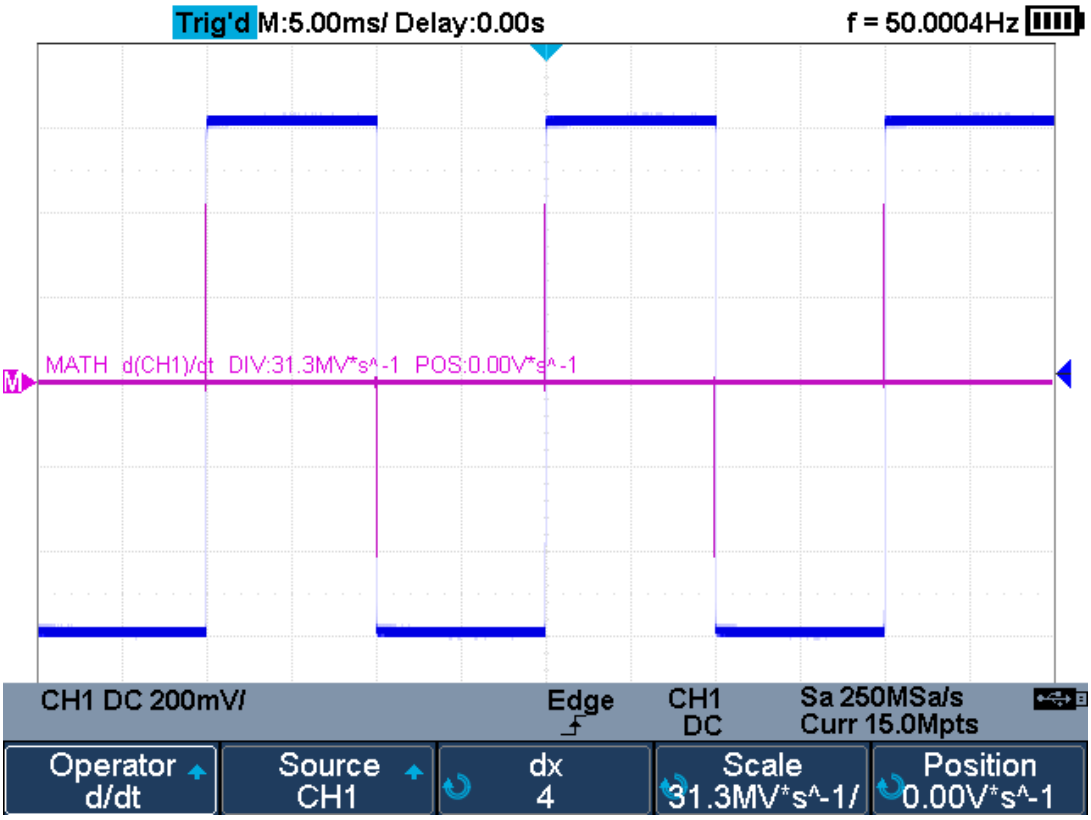


Figure 73 Square Wave and Derivative

You can use d/dt to measure the instantaneous slope of a waveform. For example, the slew rate of an operational amplifier may be measured using the d/dt function

Note: Because d/dt is very sensitive to noise, it is helpful to set acquisition mode to **Average.**

Integrate

dt (integrate) calculates the integral (area under the curve) of the selected channel. You can use integrate to calculate the energy of a pulse in volt-seconds or measure the area under a waveform.

dt plots the integral of the source using the Trapezoidal Rule. The equation is:

$$I_n = c_0 + \Delta t \sum_{i=0}^{n} y_i$$

Where:

- I = integrated waveform
- Δt = point-to-point time difference
- y = channel 1, 2, 3, or 4 data points
- c_o = arbitrary constant
- i = data point index

The integrate operator provides an **Offset** F key that lets you enter a DC offset correction factor for the input signal. A small DC offset in the integrate function input (or even small oscilloscope calibration errors) can cause the integral to "ramp" up or down. The DC offset correction lets you level the integral waveform.

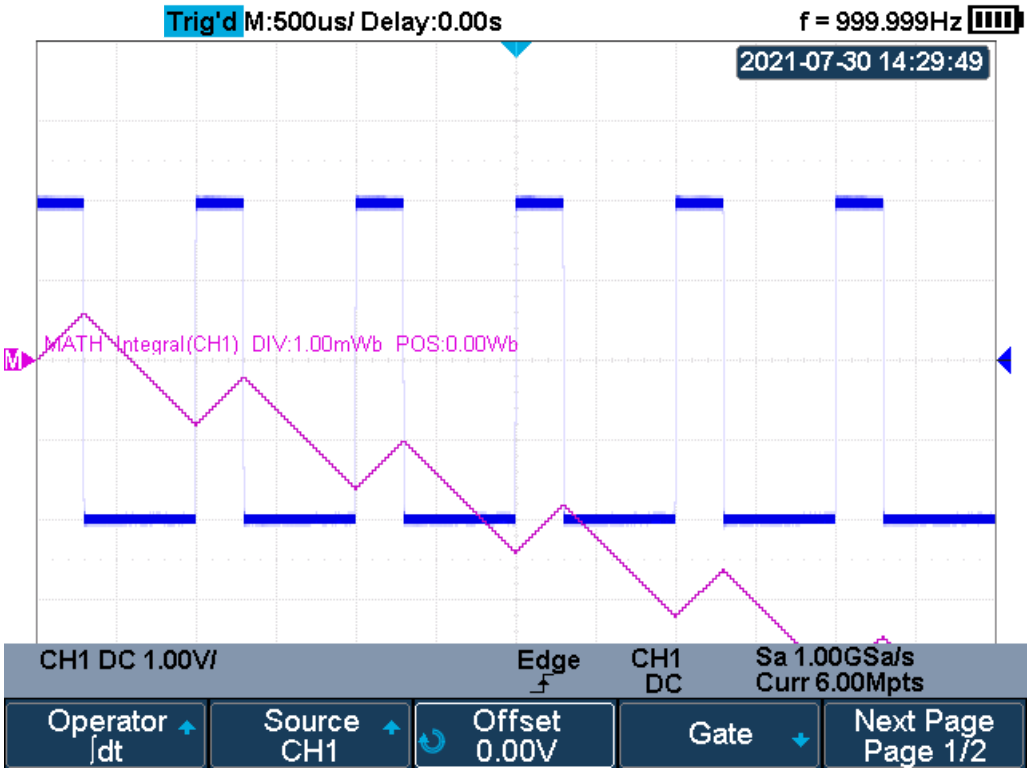


Figure 74 Integral without Offset

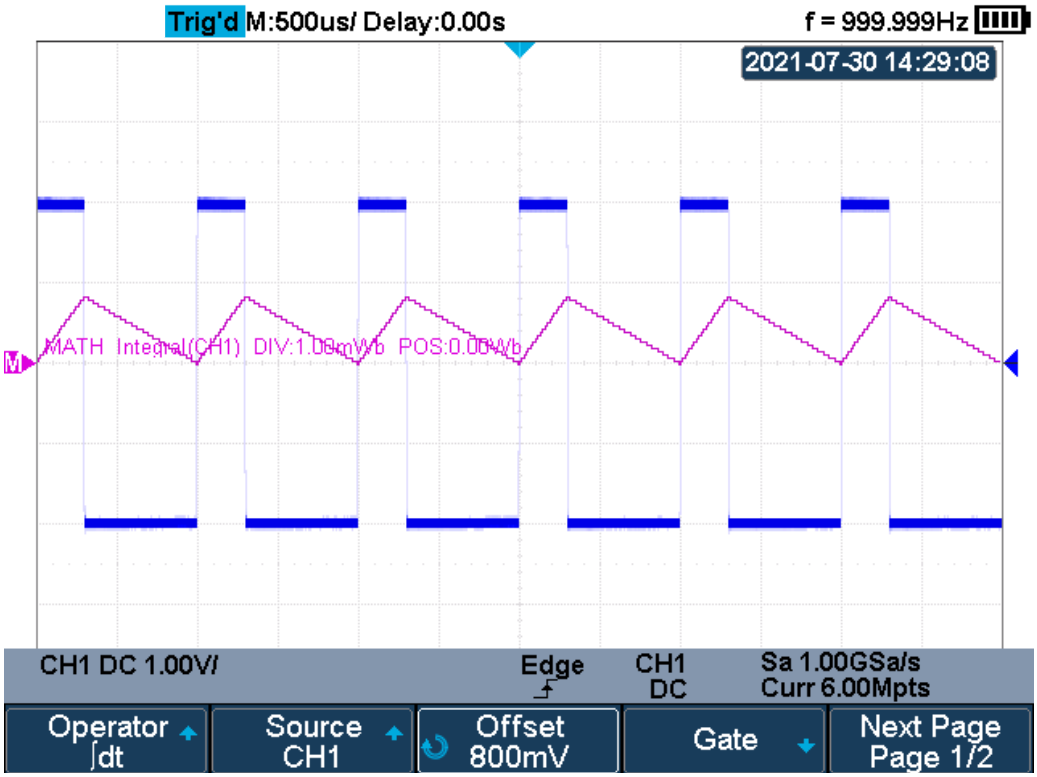


Figure 75 Integral with Offset

Square Root

Square root ($\sqrt{\ }$) calculates the square root of the selected source.

Where the transform is undefined for a particular input (where the signal is negative), holes (zero values) appear in the function output.

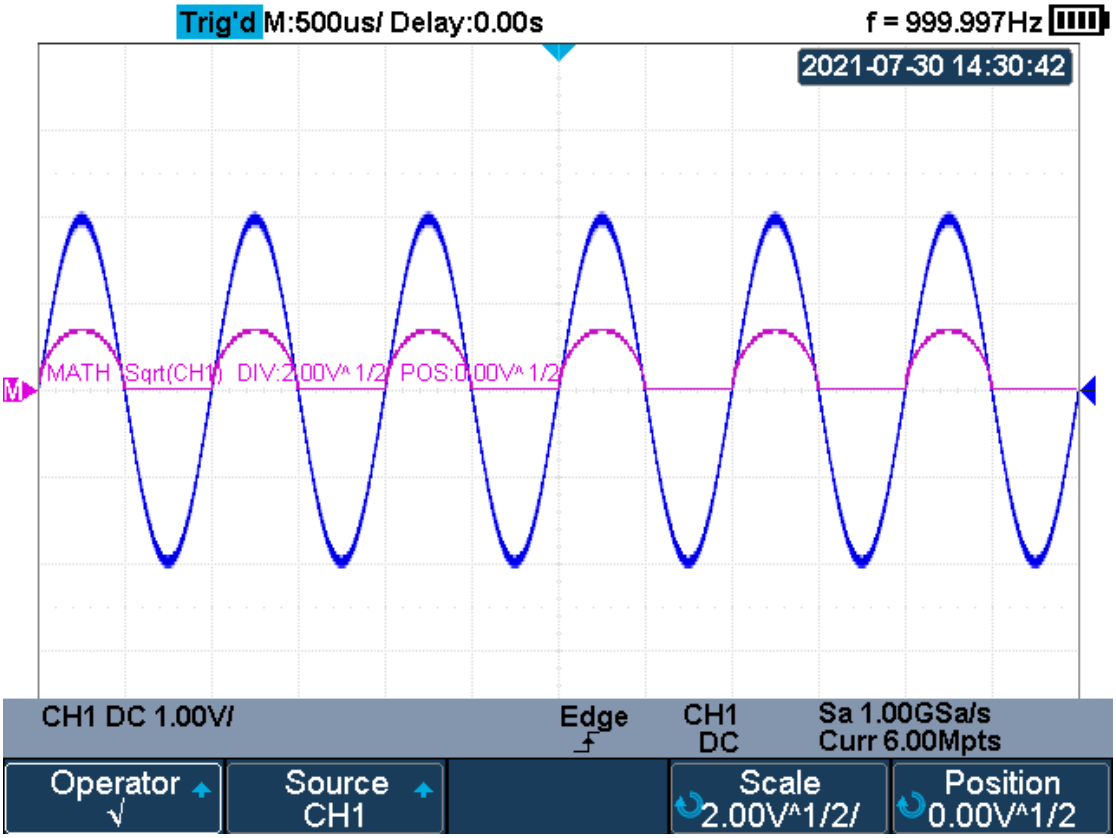


Figure 76 Square Root

Cursors

Cursors are horizontal and vertical markers that indicate X values and Y values on a selected waveform You can use cursors to make custom voltage and time measurements on oscilloscope signals.

X Cursors

X cursors are vertical dashed lines that can be moved from side to side to measure time. When the source is a FFT waveform, X cursors measure frequency.

The X1 cursor is the left (default position) vertical dotted line; it can be moved to anywhere on the screen.

The X2 cursor is the right (default position) vertical dotted line; it can also be moved to anywhere on the screen.

Use the **Universal Knob** to move the X1 and X2 cursors. The cursors' values are displayed in the cursors box in the upper-left corner of the screen, along with the (time) difference between X1 and X2 (△T) and the corresponding frequency, 1/△T. (For FFT waveforms, ΔT will have units of frequency, and 1/ΔT will be the corresponding time.)

When the cursor type is set to X2-X1, use **Universal Knob** will move the X1 and X2 cursors together. The value under the menu option is the difference between the X1 and X2 cursors.

Y Cursors

The Y cursors are horizontal dotted lines that can be moved up and down to measure voltage (V) or current (A). When the cursor's source is a math function, the unit will match the math function.

The Y1 cursor is the top (default) horizontal dotted line; it can be moved to any height on the screen.

The Y2 cursor is the bottom (default) horizontal dotted line; it can also be moved to any height on the screen.

Use the **Universal Knob** move the Y1 and Y2 cursors. Their values are displayed in the cursors box in the top left corner of the screen along with the difference △Y = Y1 – Y2.

When the cursor type is set to Y2-Y1, the **Universal Knob** will move the Y1 and Y2 cursors together. The value under the menu option is the difference between the Y1 and Y2 cursors.

Make Cursor Measurements

1. Press **Cursors** on the front panel to enter the CURSOR function menu.
2. Press the **Mode** F key and set the cursor mode to **Manual** or **Track**.
3. Press the **Source** F key, and then use the **Universal Knob** to select the desired source. Only input channels, math waveforms, and reference waveforms are displayed as available for cursors.
4. Press the **X Ref** and **Y Ref** F keys to set the reference of X cursors and Y cursors:
   - **Position**: when the horizontal/vertical scale is changed, the X/Y cursors remain at the same position on the display but the values change.
   - **Delay/Offset**: when the horizontal/vertical axis is changed, the value of X/Y cursors remain fixed, but their positions on the screen change.
5. To make cursor measurements:
   - To measure the horizontal time, use the **Universal Knob** to move the X1 and X2 cursors to the desired place. If necessary, set the cursor type to **X2-X1** and move the X1 and X2 cursors together.
   - To measure vertical voltage or current, use the **Universal Knob** to move the Y1 and Y2 cursors to desired place. If necessary, set the cursor type to "Y2-Y1" and move the Y1 and Y2 cursors together.
   - To adjust the transparency of the cursors message box, press **Display/Persist**, go to the second page, press the **Transparence** (20% to 80%) F key, and then turn the **Universal Knob** to adjust the transparency to the desired value.

Cursor examples:

Use cursors to measure pulse width:

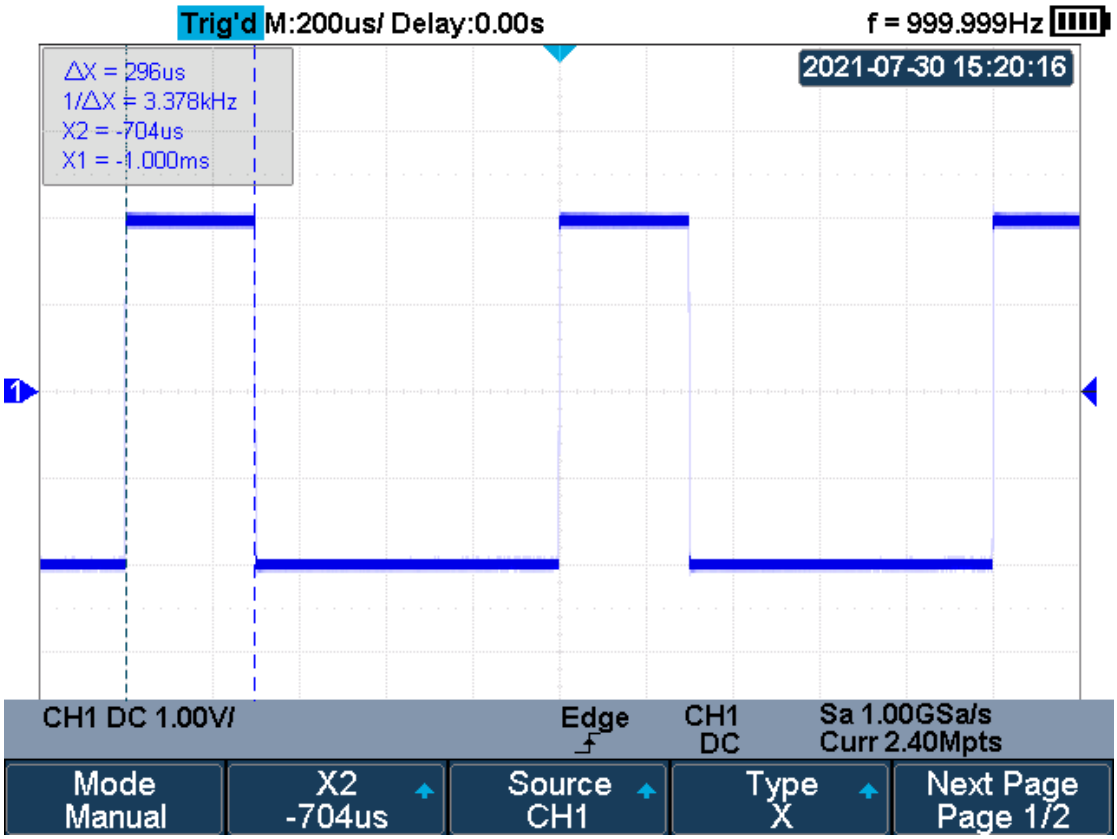


Figure 77 Measure Pulse Width

Measure

The oscilloscope can provide measurements of 38 waveform parameters and statistics. These include voltage, time, and delay parameters.

Voltage and time parameters are under the Type option. The results of the last four selected measurements are displayed at the bottom of screen and above the menu. Delay parameters are under the **All Measure** submenu. Set the Delay option to On to display all delay parameters.

Contents:

Type of Measurement

Voltage Measurements

Voltage measurements include 17 parameter measurements.

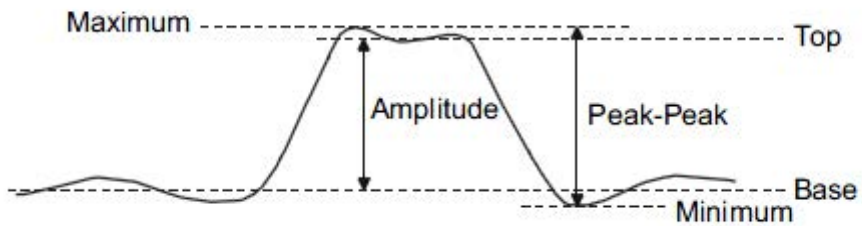


Figure 78 Voltage Measurements

1. **Peak-Peak:** Difference between maximum and minimum data values.
2. **Maximum:** Highest value in input waveform.
3. **Minimum:** Lowest value in input waveform.
4. **Amplitude:** Difference between top and base in a bimodal signal, or between max and min in a unimodal signal.
5. **Top:** Value of most probable upper state in a bimodal waveform.
6. **Base:** Value of most probable lower state in a bimodal waveform.
7. **Mean:** Average of all data values
8. **Cycle mean:** Average of data values in the first cycle.
9. **Stdev:** Standard deviation of all data values
10. **Cycle Stdev:** Standard deviation of all data values in the first cycle
11. **Rms:** Root mean square of all measured values.
12. **Cycle RMS:** Root mean square of all data values in the first cycle.
13. **Overshoot:** Overshoot is distortion that follows a major edge transition expressed as a percentage of Amplitude. ROV means rising edge overshoot and FOV means falling edge overshoot.

$$\text{Rising edge overshoot} = \frac{\text{local Maximum} - \text{D Top}}{\text{Amplitude}} \text{ X } 100$$

$$\text{Falling edge overshoot} = \frac{\text{Base} - \text{D local Minimum}}{\text{Amplitude}} \text{ X } 100$$

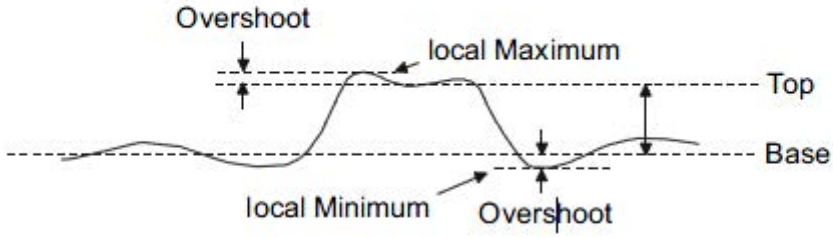


Figure 79 Overshoot

14. **Preshoot:** Preshoot is distortion that precedes a major edge transition expressed as a percentage of Amplitude. The X cursors show which edge is being measured (edge closest to the trigger reference point).

$$\text{Rising edge preshoot} = \frac{\text{local Minimum} - \text{D Top}}{\text{Amplitude}} \text{ X } 100$$

$$\text{Falling edge preshoot} = \frac{\text{Base} - \text{D local Minimum}}{\text{Amplitude}} \text{ X } 100$$

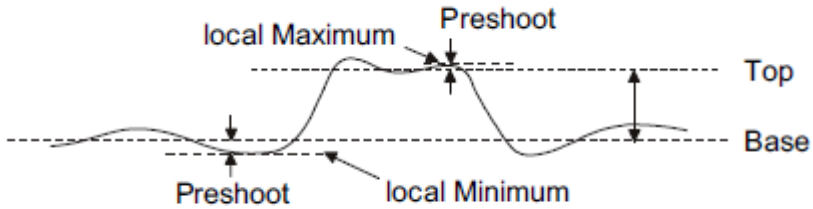


Figure 80 Preshoot

15. **Level@X: t**he voltage value between the trigger point and the vertical position of the channel

Time Measurements

Time measurements include 11 parameter measurements.

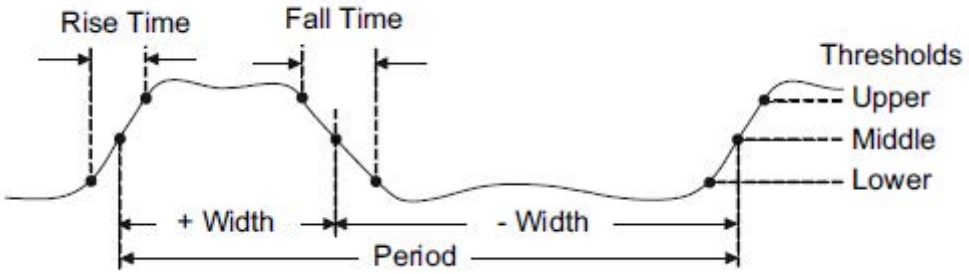


Figure 81 Time Measurements

1. **Period:** Period for every cycle in waveform at the 50% level, and positive slope.
2. **Frequency:** Frequency for every cycle in waveform at the 50% level and positive slope
3. **+ Width:** Width measured at 50% level and positive slope.
4. **- Width:** Width measured at 50% level and negative slope.
5. **Rise Time:** Duration of rising edge from 10-90%.
6. **Fall Time:** Duration of falling edge from 90-10%.
7. **BWid**: Time from the first rising edge to the last falling edge, or the first falling edge to the last rising edge at the 50% crossing.
8. **+ Duty:** Ratio of positive width to period.
9. **- Duty:** Ratio of negative width to period.
10. **Delay：** Time from the trigger to the first 50% crossing.
11. **T@L：** Time from trigger of each transition at a specific level and slope, including Current, Max, Min, Mean, and Stdev.

Delay Measurements

Delay measurements measure the time difference between various events on different channels. There are 10 kinds of delay measurements.

1. **Phase:** Calculate the phase difference between two edges.
2. **FRFR:** Time between the first rising edges of the two channels.
3. **FRFF:** Time from the first rising edge of channel A to the first falling edge of channel B.
4. **FFFR:** Time from the first falling edge of channel A to the first rising edge of channel B.
5. **FFFF:** Time from the first falling edge of channel A to the first falling edge of channel B.
6. **FRLR:** Time from the first rising edge of channel A to the last rising edge of channel B.
7. **FRLF:** Time from the first rising edge of channel A to the last falling edge of channel B.
8. **FFLR:** Time from the first falling edge of channel A to the last rising edge of channel B.
9. **FFLF:** Time from the first falling edge of channel A to the last falling edge of channel B.
10. **Skew：** Time of source A edge minus time of nearest source B edge.

Add Measurement

Perform the steps below and select voltage or time parameters to make automatic measurement.

1. Press **Measure** on the front panel to enter the MEASURE function menu. When the frequency and period are enabled for the current trigger channel, the statistics will also be enabled.
2. Press the **Source** F key, and then use the **Universal Knob** to select the desired channel. Only the displayed channels are available for measurements.
3. To select and display measurement parameters,. press the **Type** F key, and then turn the **Universal Knob** to select the desired measurement parameter.
4. Press the **Universal Knob** to add the measurement parameter. The parameter(s) and value(s) will be shown above the menu, and the statistics status will update.
5. To turn off the statistic function, press the **Statistics** F key to select **Off**.

The measurement display area can display up to 4 measurement parameters, which will be arranged in the order they were selected. If you add a fifth parameter, it will replace the first one.

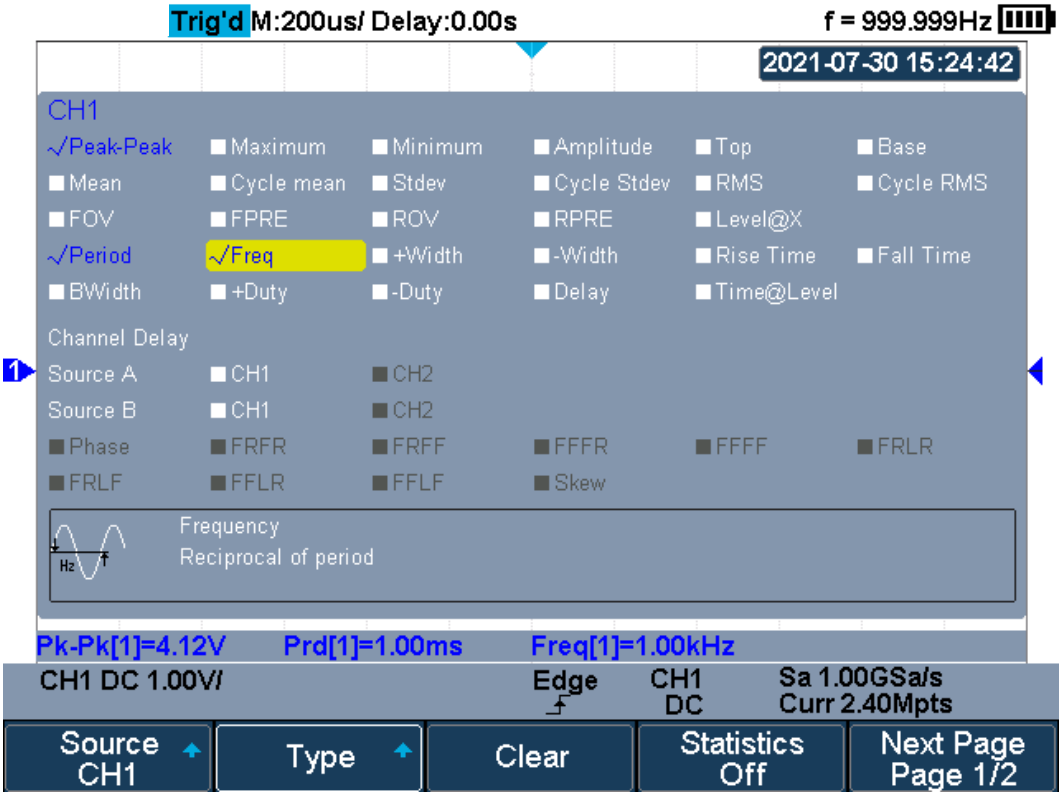


Figure 82 Select the Measurement Parameter

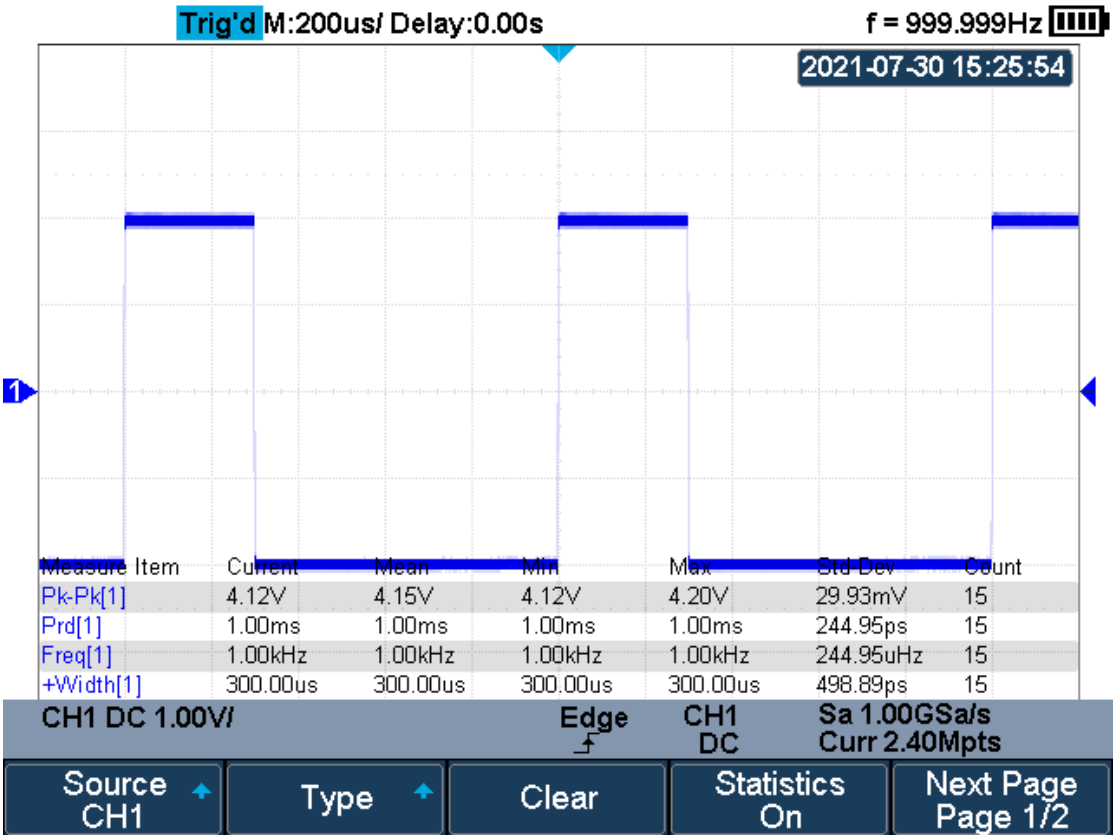


Figure 83 Added the Measurement

Note: if the parameter does not match the measure condition, it will display as "****".

Clear Measurement

Press the **Clear** F key to clear all currently displayed parameters.

All Measurement

All measurement reports all of the voltage and time parameters of the current measurement source.

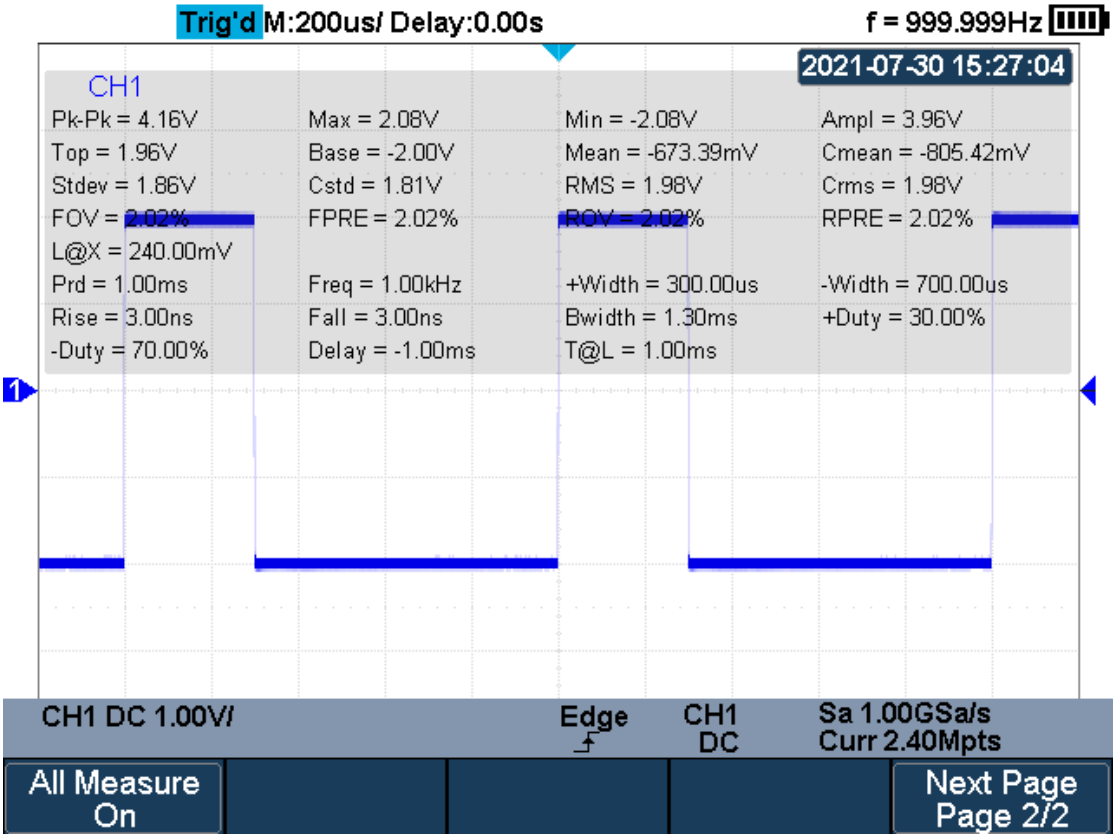


Figure 84 All Parameters Measurement

To display all parameters:

1. Press **Measure** on the front panel to enter the MEASURE function menu.
2. Press the **All Measure** F key to select **On**.
3. Press the **Source** F key to select the measure source.

Gate Measurement

The oscilloscope supports gate measurement and performs the selected measurement within the upper and lower limits of the gate. Setting the gate will affect the measurement of all voltage, time, and delay parameters.

1. Press **Measure** → **Gate** → **On** to open the gate measurement.
2. Press the **Gate A** F key to move the position of gate A with the **Universal Knob.**
3. Press the **Gate B** key to move the position of gate B with the **Universal Knob.**
4. Press the **Gate A-B** key to move the gate A and B positions of the simultaneously with the **Universal Knob**.

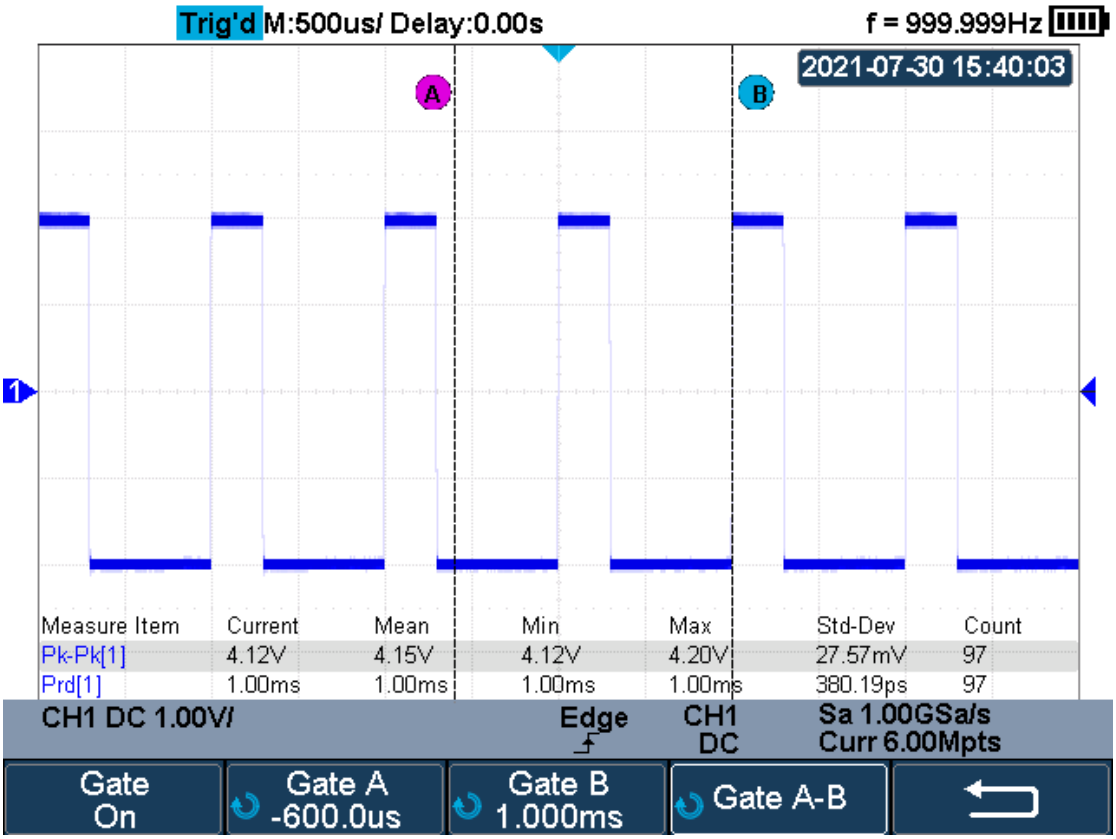


Figure 85 Gate Measurement

Display

You can set the display type, color, persistence, grid type, waveform intensity, grid brightness, and transparency.

The contents of this chapter:

- Display Type
- Color Grade
- Persistence
- Clear Display
- Grid Type
- Intensity
- Grid Brightness
- Transparence
- LCD Light

Display Type

Press **Display/Persist** on the front panel, and then press the **Type** F key to select **Vectors** or **Dots** display type.

- **Vectors:** the sample points are connected by lines and displayed. Normally, this mode can provide the most vivid waveform to view the steep edge of the waveform (such as a square waveform).
- **Dots:** display the sample points directly. You can directly view each sample point and use the cursor to measure its X and Y values.

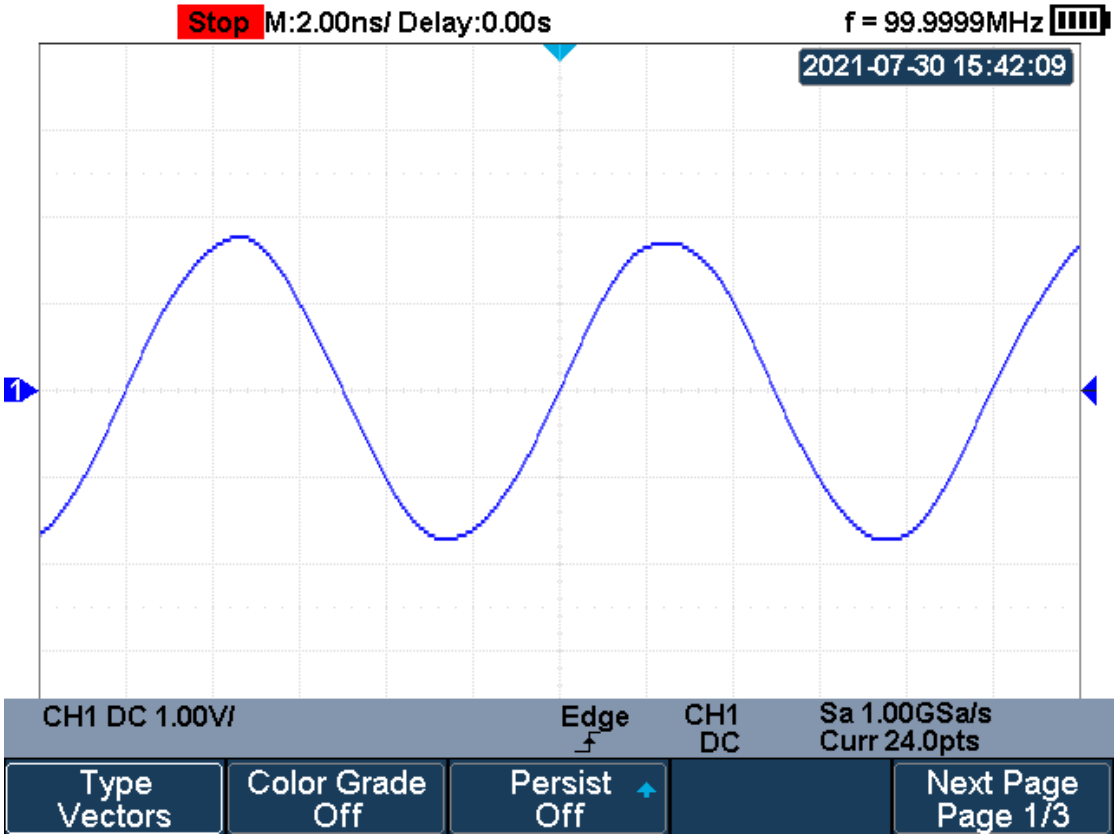


Figure 86 Vectors Display

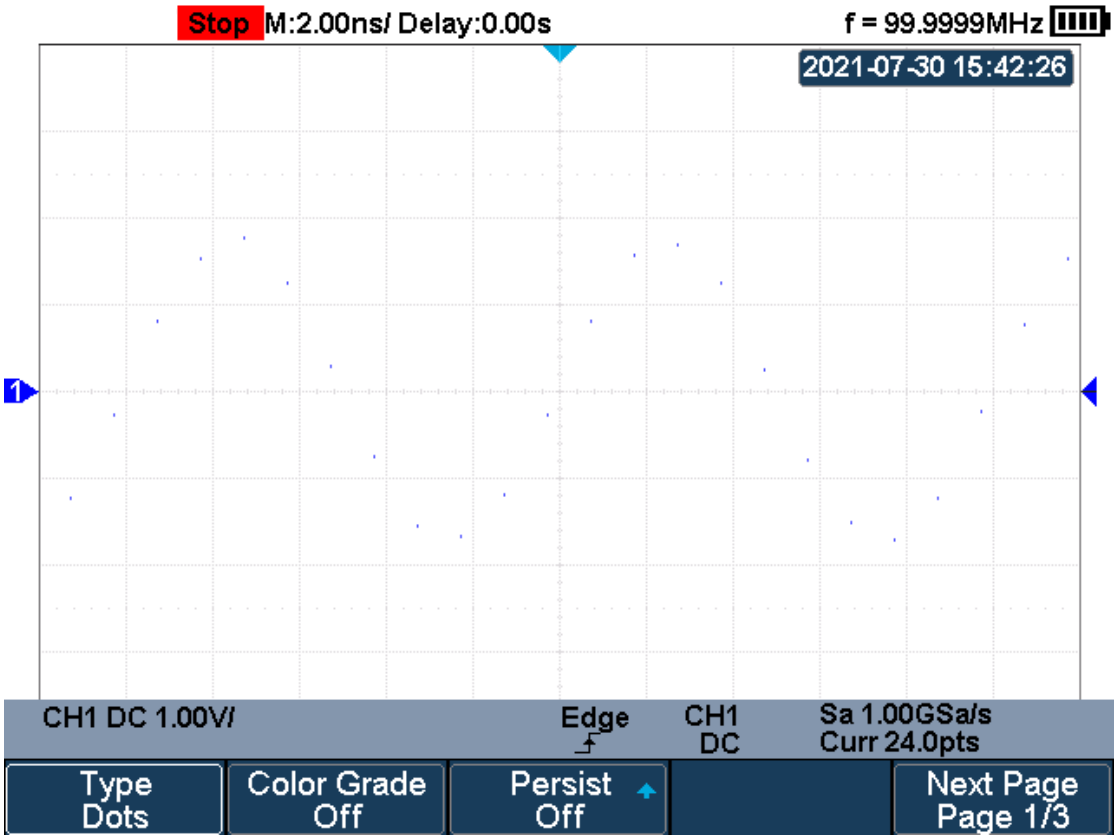


Figure 87 Dots Display

Color Grade

Color temperature adopts the change of waveforms' color to reflect the estimated probability of exactly this waveform appearing. The greater the probability that the waveform appears, the warmer the color is, the smaller the probability, the colder the color is.

The picture below shows the change of color from cold to warm. Press **Display/Persist** on the front panel, and then press the **Color Grade** F key and set the option to **On** to turn on the color temperature function. You can compare the waveform's color with the picture below to estimate the its probability.

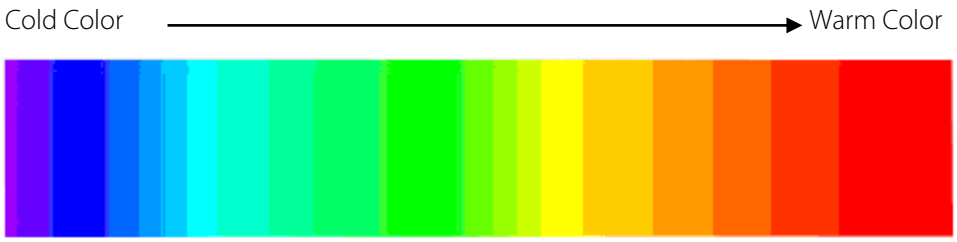


Figure 88 Color Temperature

Persistence

With persistence, the oscilloscope updates the display with new acquisitions, but does not immediately erase the results of previous acquisitions. All previous acquisitions are displayed with reduced intensity. New acquisitions are shown in their normal color with normal intensity.

Do the following steps to set and clear persistence:

1. Press **Display/Persist** on the front panel to enter the DISPLAY function menu.
2. Press the **Persist** F key; then turn the **Universal Knob** to choose the setting:
   - **Off** —turn of persistence.
   - **Variable persistence time** (1 second, 5 seconds, 10 seconds, 30 seconds) — how long prior waveforms persist. Older waveforms are erased after this delay.
   - **Infinite** —select "Infinite" Results of previous acquisitions are never erased. Use infinite persistence to measure noise and jitter, see the worst-case extremes of varying waveforms, look for timing violations, or capture rare events.

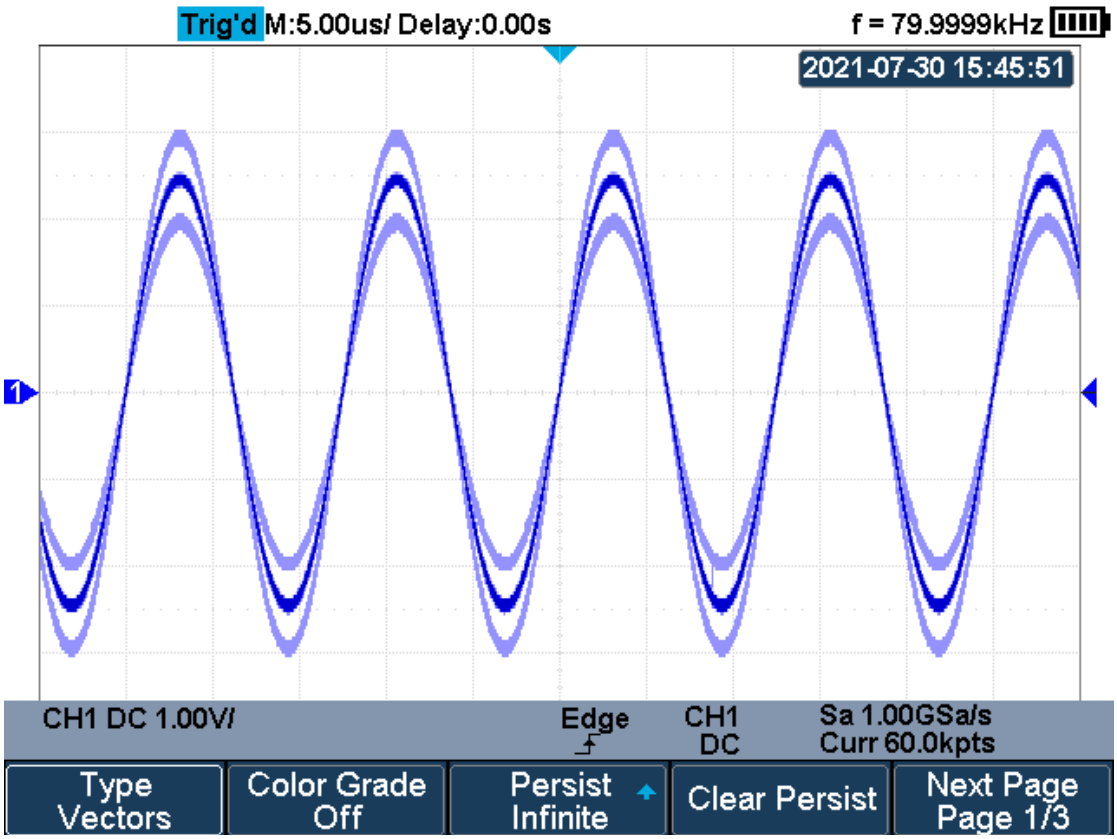


Figure 89 Persist Set to Infinite

3. When **Persist** is **On**, press the **Clear Persist** F key to erase the results of previous acquisitions from the display. The scope will start to accumulate acquisitions again.
4. To return to the normal display mode, turn off Persist, and the previous acquisitions will be cleared at once.

Clear Display

Press **Display/Persist** on the front panel to enter the DISPLAY function menu; press the **Clear Display** F key to clear all the waveforms currently on the screen, and to acquire and display new waveforms.

Grid Type

To select grid type

1. Press **Display/Persist** on the front panel to enter the DISPLAY function menu.
2. Press the **Next Page** F key to go to the second page of the Display function menu.
3. Press the **Grid** F key; and then turn the **Universal Knob** to select the desired grid type. Press the **Grid** F key repeatedly can also select the grid type.

3 types of grid are available:

 Display 12X8 grid type

 Display 2X2 grid type

 Display without grid

Intensity

Do the following steps to adjust waveform intensity:

1. Press **Display/Persist** on the front panel to enter the DISPLAY function menu.
2. Press the **Next Page** F key to go to the second page of the Display function menu.
3. Press the **Intensity** F key; and then turn the **Universal Knob** to select the desired value. The default value is 50%, and the range is from 0% to 100%.

Increasing the intensity lets you see the maximum amount of noise and infrequent events. Reducing the intensity can expose more detail in complex signals.

> Note: Waveform intensity adjustment affects analog channel waveforms only (not math waveforms, reference waveforms, digital waveforms, etc.).

Grid Brightness

To adjust the grid brightness:

1. Press **Display/Persist** on the front panel to enter the DISPLAY function menu.
2. Press the **Next Page** F key to go to the second page of the Display function menu.
3. Press the **Graticule** F key; and then turn the **Universal Knob** to select the desired value. The default value is 20%, and the range is from 0% to 100%.

Transparency

Transparency can be used to adjust the opacity of the message box of cursor, measure, Pass/Fail, and pop-up menus.

Under Cursor or Measure or any other menu operation, to change the transparency of the message box:

1. Press **Display/Persist** on the front panel to enter the DISPLAY function menu.
2. Press the **Next Page** F key to go to the second page of the Display function menu.
3. Press the **Transparence** F key; and then turn the **Universal Knob** to select the desired value. The default value is 80%, and the range is from 20% to 80%.

LCD Light

Do the following steps to adjust the LCD Light:

1. Press **Display/Persist** on the front panel to enter the DISPLAY function menu.
2. Press the **Next Page** F key to go to the third page of the Display function menu.
3. Press the **LCD Light** F key; and then turn the Universal Knob to select the desired value.

Save and Recall

Oscilloscope setups, waveforms, pictures, and CSV files can be saved to internal memory or to a USB storage device. The saved setups and waveforms can be recalled later. The oscilloscope provides a USB port on the side panel to connect flash drive or other device for external storage.

Contents:

Save Type

The oscilloscope supports setups, waveforms, images, and CSV file storage. The default save type is setups.

1. Setups

This is the default storage type. It saves the settings of the oscilloscope in internal or external memory in ".xml" format. These stored settings can be recalled.

2. Reference

The oscilloscope saves the waveform data in ".REF" format. You can choose whether to save data from Channel 1 or 2 as a reference. At recall, the data will be displayed on the screen by REFA or REFB.

3. BMP

The oscilloscope saves the screen image in ".bmp" format. You can specify the file name and what folder to save the image in. The scope cannot recall .bmp files or other images.

4. JPG

The oscilloscope saves the screen image in ".jpg" format. You can specify the file name and what folder to save the image in. The scope cannot recall .jpg files or other images.

5. PNG

The oscilloscope saves the screen image in ".png" format. You can specify the file name and what folder to save the image in. The scope cannot recall .bmp files or other images.

6. Binary

The oscilloscope saves the waveform data in ".BIN" format. Data from both channels can be saved in the same file. The scope cannot recall binary files.

7. CSV

The oscilloscope saves the waveform data in ".CSV" format. The output files contain the waveform data of the displayed channels and the main settings of the scope. The scope cannot recall .csv files.

CSV can also save waveform parameters (Vpp, etc.) Set the save type to **CSV**, and set the **Param Save** option to **On** or **Off** to turn parameter storage on or off.

8. Matlab

The oscilloscope saves the waveform data in ".DAT" format. All of the active channels' data will be saved in the same file. The scope cannot recall .dat files.

9. To Default Key

Caution: This function overwrites the settings that pressing **Default** restores.

After saving the settings with **To Default Key**, pressing **Default** will then restore the modified settings instead of the original factory default settings. Do not use this function unless you know exactly what the current settings are and you are certain that you want them as default. You will still be able to recall the factory settings through Save/Recall → Recall → Factory Default (see the final section, Factory Setup), but you will not be able to do this with **Default**.

With that in mind, to change the default settings:
After choosing the settings you want, press **Shift** and then **Cursors**. Push the **Save** F key, then the **Type** F key, choose **To Default Key**, and then the **Press to Save** F key.

10. FileConverter

The mini tool converts stored binary files to CSV format for viewing with a spreadsheet program. It can convert .bin files, sample logger .slg files, and measure logger .mlg files.

This is ideal when collecting large datasets. For a waveform frame with deep memory such as 12 Mpts, to save directly as a CSV file will take long time and will occupy a large amount of memory on a USB storage device. Instead we recommend saving as a .bin file and then converting it to CSV with FileConverter.

Internal Save and Recall

➢ **Save the specified oscilloscope setting to internal memory.**

1. Connect the signal to the oscilloscope and obtain stable display.
2. Press **Shift** and **Cursors** on the front panel to enter the SAVE/RECALL function menu.
3. Press the **Save** F key to enter the SAVE function menu.
4. Press the **Type** F key, turn the **Universal Knob** to select **Setups**; and then press the knob to confirm.
5. Press the **Press to Save** F key to enter the disk manager, enter the "local" directory, and press the **Press to Save** F key to save the current setup.

➢ **Load the specified type of file from internal memory.**

If want to recall the setup after having finished the steps above, please do the following steps:

1. Press the **Recall** F key enter the RECALLING function menu.
2. Press the **Type** F key and then turn the **Universal Knob** to select Setups; and then press the knob to confirm.
3. Press the **Press to Recall** F key to enter the disk manager, recall the current setup form the appointed location. After a few seconds, you will get the popup message, "**Recalled file successfully!**"

Note: if you need to delete a setup file in the memory, please save a new setup to the same location with the same name to overwrite it.

External save and recall

Before using external storage and recall, make sure that a USB flash drive is connected correctly. External storage can be used to save all of the filetypes above**,** but images and CSV files cannot be recalled.

➢ **Save the specified type of file to the external USB flash device.**

1. Press the **Shift** and **Cursor** button on the front panel to enter the SAVE/RECALL function menu.
2. Press the **Save** F key to enter the SAVE menu.
3. Press the **Type** F key to select **Setups** and then press the knob to confirm.
4. Press the **Press to Save** F key to enter the disk manager.

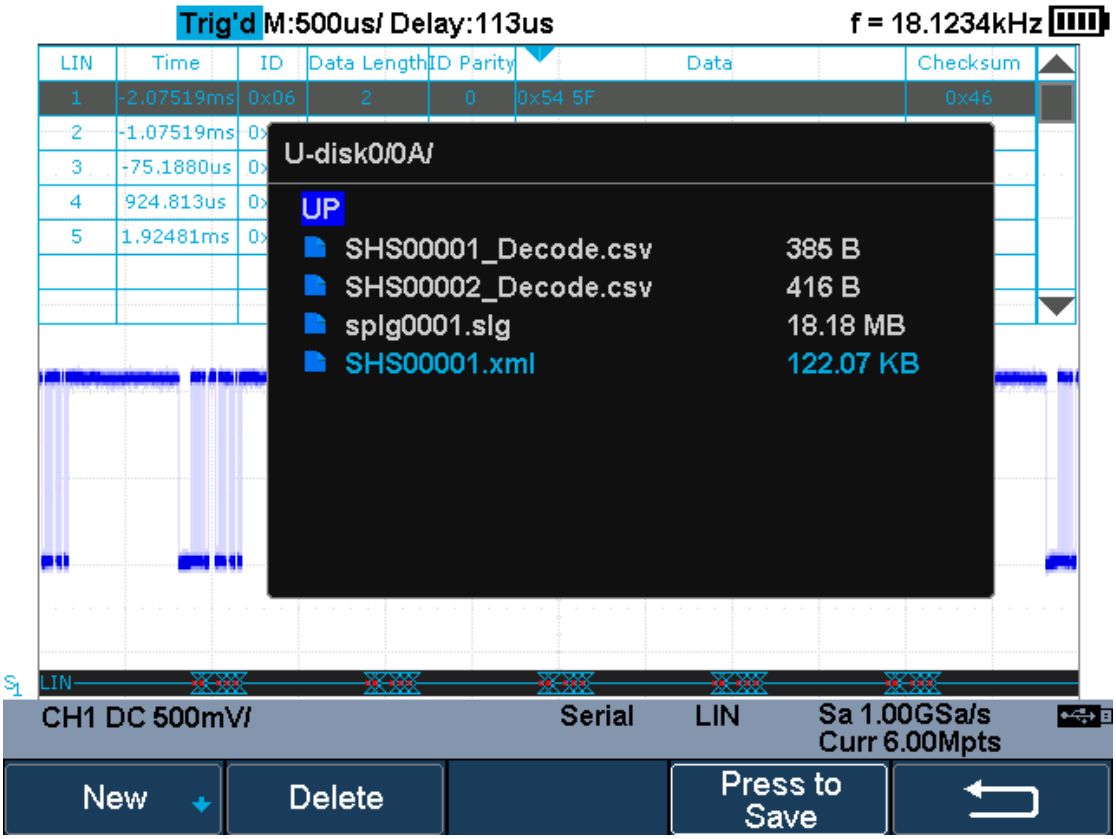


Figure 90 SAVE/RECALL File System

5. Use the **Universal Knob** to select the desired location. Files can be stored in the root (top) directory or in a folder within the root directory of the storage device.
6. After selecting the save position, press **New** F key to open the file naming window, as shown below. Refer to the descriptions in "**Create a New File or Folder"** to create a new file name.

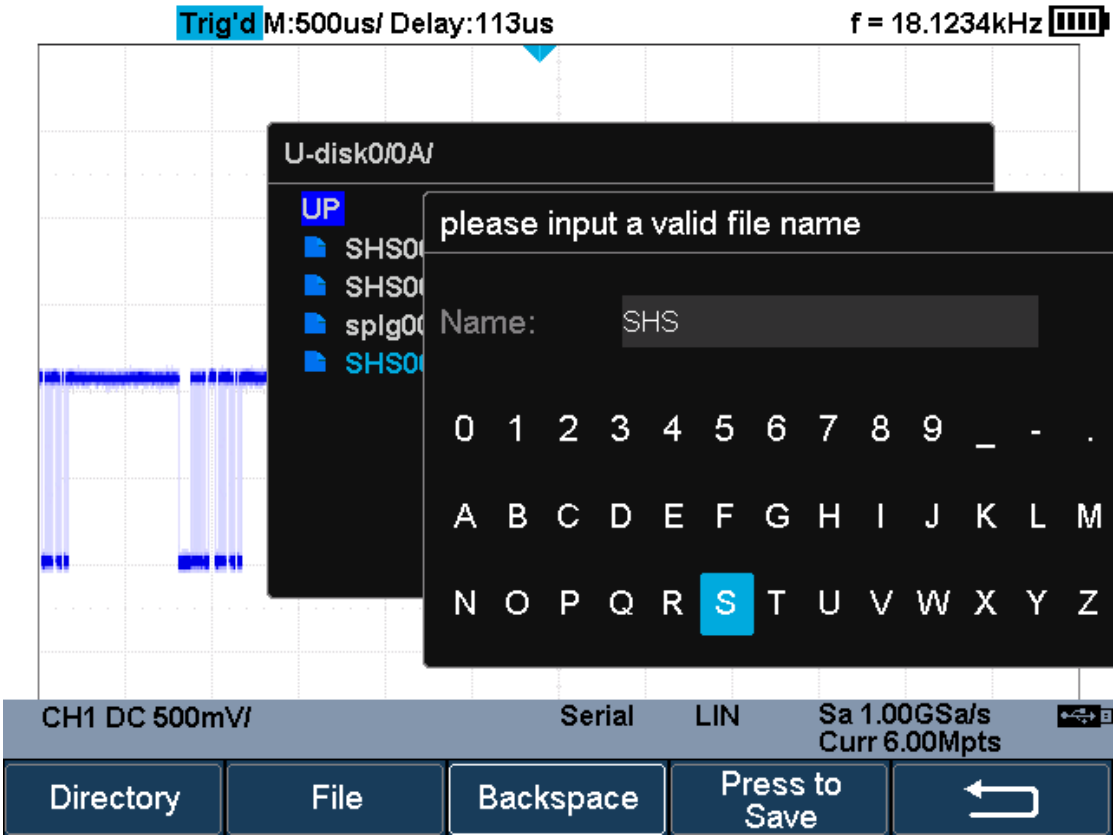


Figure 91 File Name Dialogue

7. Press the **Press to Save** F key to save the current setup to the external USB storage device.

➢ **Load the specified type of file from the external USB storage device.**

1. Press the **Shift** and **Cursor** button on the front panel to enter the SAVE/RECALL function menu.
2. Press the **Recall** F key to enter the RECALL menu.
3. Press the **Type** F key to select **Setups** or **Reference**.
4. Press the **Press to Recall** F key to enter the disk manager.
5. Turn the **Universal Knob** to select the file to be recalled, press the **Press to Recall** F key to recall the reference waveform or setup.

Disk Management

Execute the following operations through the disk management menu:

- Create a New File or Folder
- Delete a File or Folder
- Rename a File or Folder

Create a New File or Folder

File and folder names can contain letters, numbers, underscores, and spaces. Let's use an example to show how to create a file or folder.

Example: create a file or folder named "SHS1000X_TEST"

1. Press the **Shift** and **Cursors** button on the front to enter the SAVE/RECALL function menu.
2. Press the **Save** F key, and then turn the **Universal Knob** to select one of the type.
3. Press the **Press to Save** F key to enter the SAVE/RECAL file system.
4. Press the **New** F key to open the interface shown as the figure below. It divides into two parts: name input area and keyboard area.

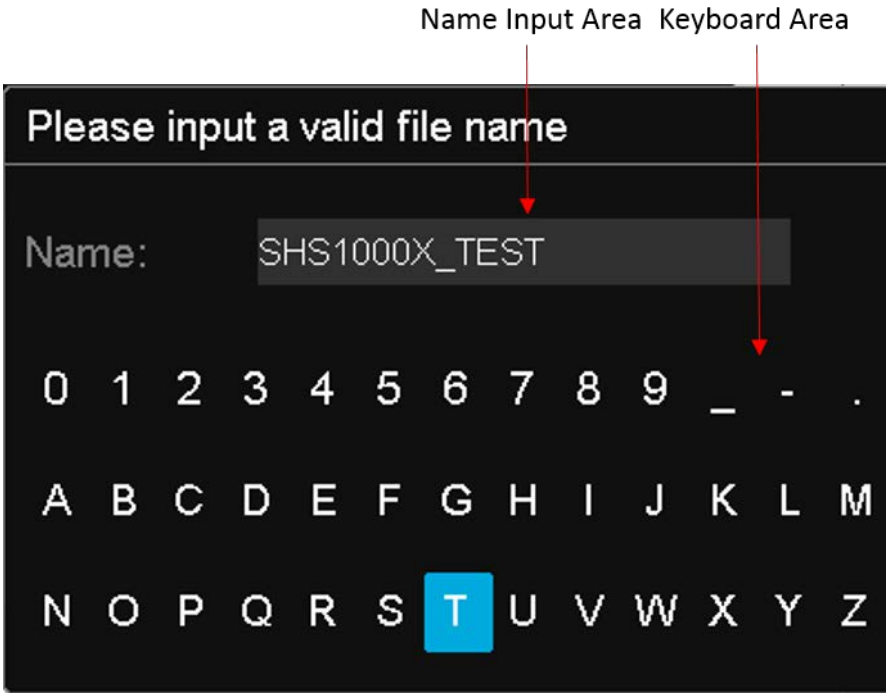


Figure 92 Input Keyboard

5. To delete the name in the name input area, press the **Backspace** F key repeatedly to delete the characters one by one.

Delete a File or Folder

1. Press **Shift** and **Cursors** on the front to enter the SAVE/RECALL function menu.
2. Press the **Save** F key, and then turn the **Universal Knob** to select the type of file you want to delete.
3. Press the **Press to Save** or **Press to Recall** F key to enter the SAVE/RECALL file system.
4. Turn the **Universal Knob** to select the file or folder to be deleted, and then press the **Delete** F key.

Rename a File or Folder

Only files of the corresponding type can be renamed.

1. Press **Shift** and **Cursors** on the front to enter the SAVE/RECALL function menu.
2. Press the **Save** F key, and then turn the **Universal Knob** to select a type.
3. Press the **Press to Save** or **Press to Recall** F key to enter the SAVE/RECAL file system.
4. Turn the **Universal Knob** to select the file or folder to be rename, and then press the **Rename** F key. For details, see the instructions in "Create a New File or Folder."

System Settings

Contents:

View System Status

Do the following steps to view the system status:

1. Press **Utility** on the front to enter the UTILITY function menu.
2. Press the **System Status** F key to view the system status of the oscilloscope. The system status includes the information below:
   - **Startup Times:** Number of times the scope has been turned on
   - **Software Version:** Current software version
   - **Uboot-OS Version:** Current U-Boot and OS versions
   - **FPGA Version**
   - **Hardware Version**
   - **Product Type:** Part number
   - **Serial No.**

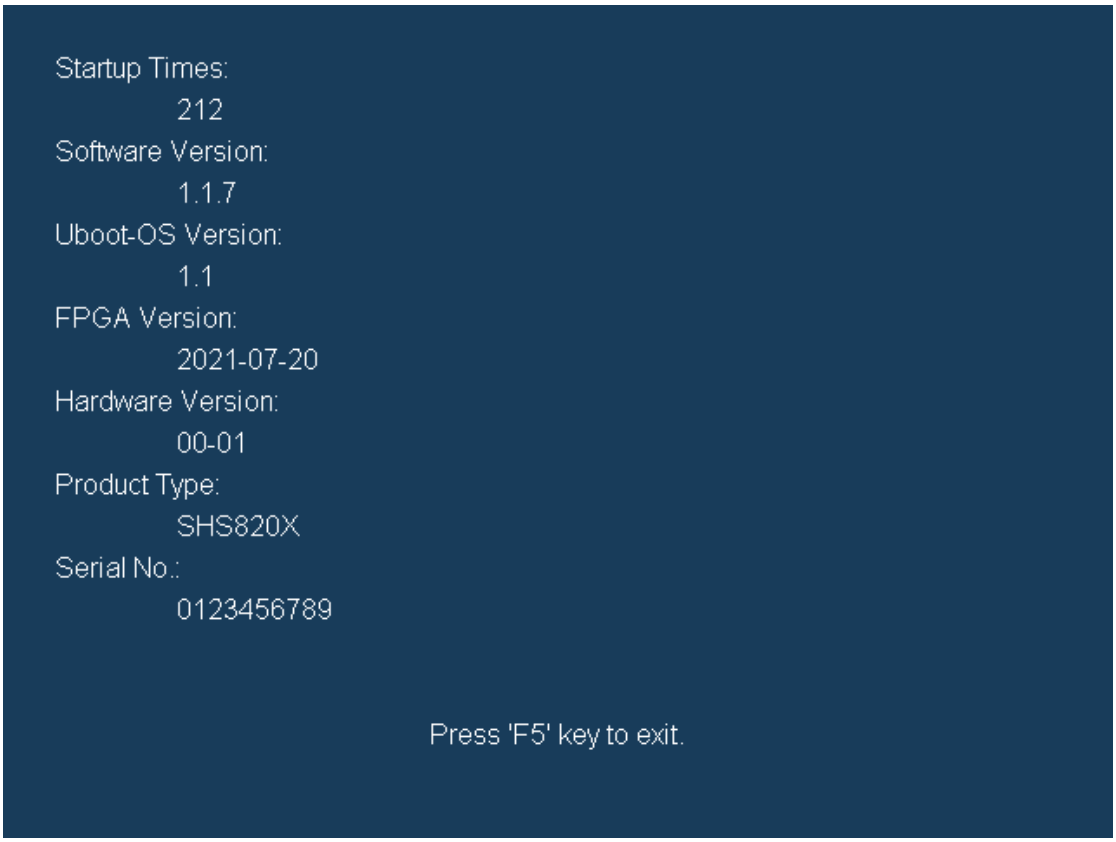


Figure 93 System Status

3. Press the **F5** button on the front panel to exit.

Do Self Cal

The self-calibration program can quickly adjust the internal settings to give the most precise measurement values. You can perform self-calibration at any time, but it's especially important when the temperature changes by 11° F (5° C) or more.
First, power on the scope and let it run for 30 minutes or more before the self-calibration. Then:

1. Disconnect any probes from the scope.
2. Press **Utility** on the front panel, and then press the **Do Self Cal** F key, and the oscilloscope will pop-out the message box shown as below:

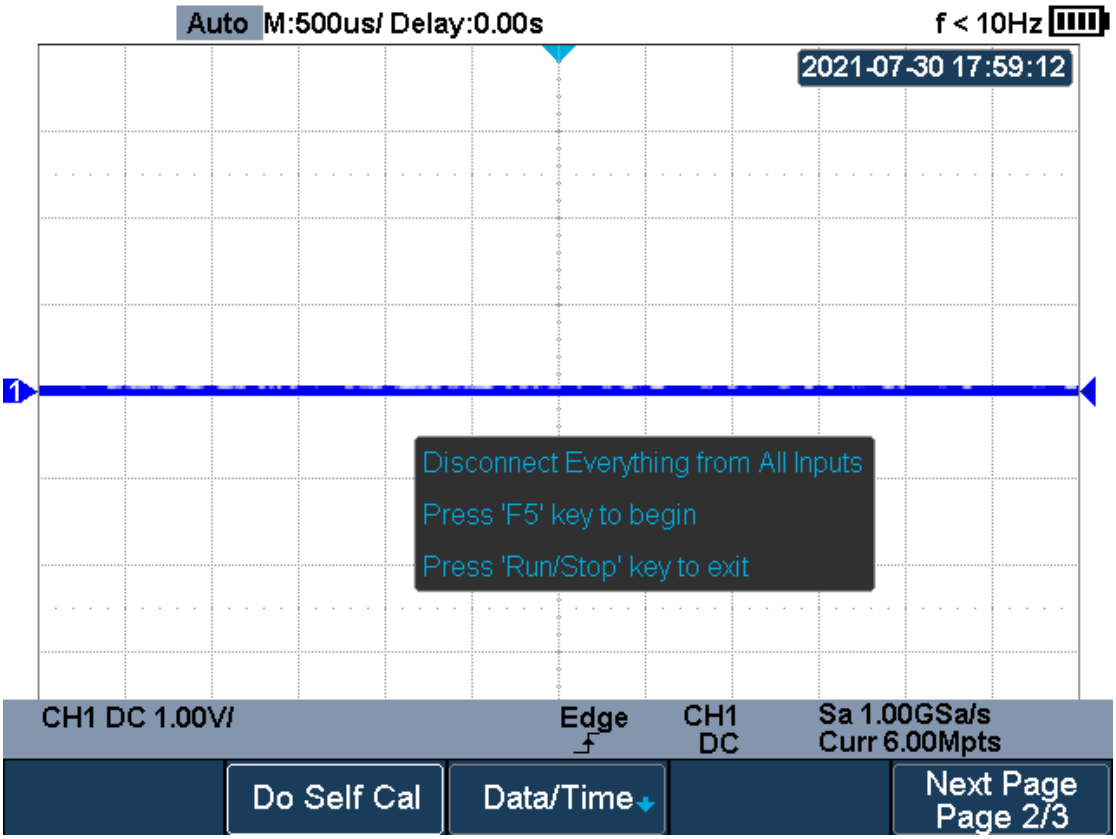


Figure 94 Do Self Cal

3. Press the **F5** button on the front panel to perform the self-calibration program. During the calibration, most of the keys are disabled.
4. When the self-calibration program is finished, it will give the message, "**Press Run/Stop key to exit**". Press **Run/Stop** to exit the calibration interface.

Quick-Cal

Quick calibration can correct minor deviations caused by temperature changes.

If you are using the oscilloscope in an area with unstable temperature, press **Utility** → **Quick-Cal** F key to select **On** to enable quick calibration.

Sound

When the sound is enabled, the oscilloscope will beep when you press a button and when the prompt message pops up.

Press **Utility** on the front panel to enter the UTILITY function menu; then press **Sound** F key to select  or  turn the sound on or off.

Language

The oscilloscope supports multiple languages for menus, Chinese/English help, and prompt messages.

1. Press **Utility** on the front panel to enter the UTILITY Function menu.
2. Press the **Language** F key; and then turn the **Universal Knob** to select the desired language. Then push down the knob to select the language.

The available languages are Simplified Chinese, Traditional Chinese, English, French, German, Spanish, Russian, Italian, and Portuguese.

Update Firmware and Configuration

The firmware and configuration can be updated directly via USB flash drive.

To update the firmware, first plug the scope in to AC power. Then:

1. Insert the flash drive with the new firmware file into the USB port on the side panel of the oscilloscope.
2. Press **Utility** on the front panel to enter the UTILITY function menu.
3. Press the **Next Page** F key to go to the third page of the UTILITY function menu.
4. Press the **Update** F key to enter the UPDATE function menu.

5. Press the **Firmware** F key to open the file manager.
6. Turn the **Universal Knob** to select the update file with a .ADS suffix; and then press the **Press to Update** F key to start updating the firmware.
   The process needs about 4 minutes. During the update, do not remove power to the oscilloscope, otherwise the oscilloscope may be permanently damaged and may not restart again.
7. After the update, a message will pop up: "**Firmware decompressed. Please restart and wait…**"
8. Restart the oscilloscope to finish the firmware update.

To update the configuration:

1. Insert the flash drive with the new configuration file inside into the USB port on the side panel of the oscilloscope.
2. Press **Utility** on the front panel to enter the UTILITY function menu.
3. Press the **Next Page** F key to go to the third page of the UTILITY function menu.
4. Press the **Update** F key to enter the UPDATE function menu.
5. Press the **Configure** F key to open the file manager.
6. Turn the **Universal Knob** to select the update file with a .CFG suffix, and then press the **Press to Update** F key to start the update. The process needs about 30 seconds.
7. After the update, a message will pop up: "**Firmware decompressed. Please restart and wait…**"
8. Restart the oscilloscope to finish the configuration update.

Do Self-Test

Self-tests can be used to test the screen, buttons, knobs and LED lights.

Screen Test

1. Press **Utility** on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** F key to go to the third page of the UTILITY function menu.
3. Press the **Do Self Test** F key to enter the SELFTEST function menu.
4. Press the **Screen Test** F key to enter the screen test interface, shown below.

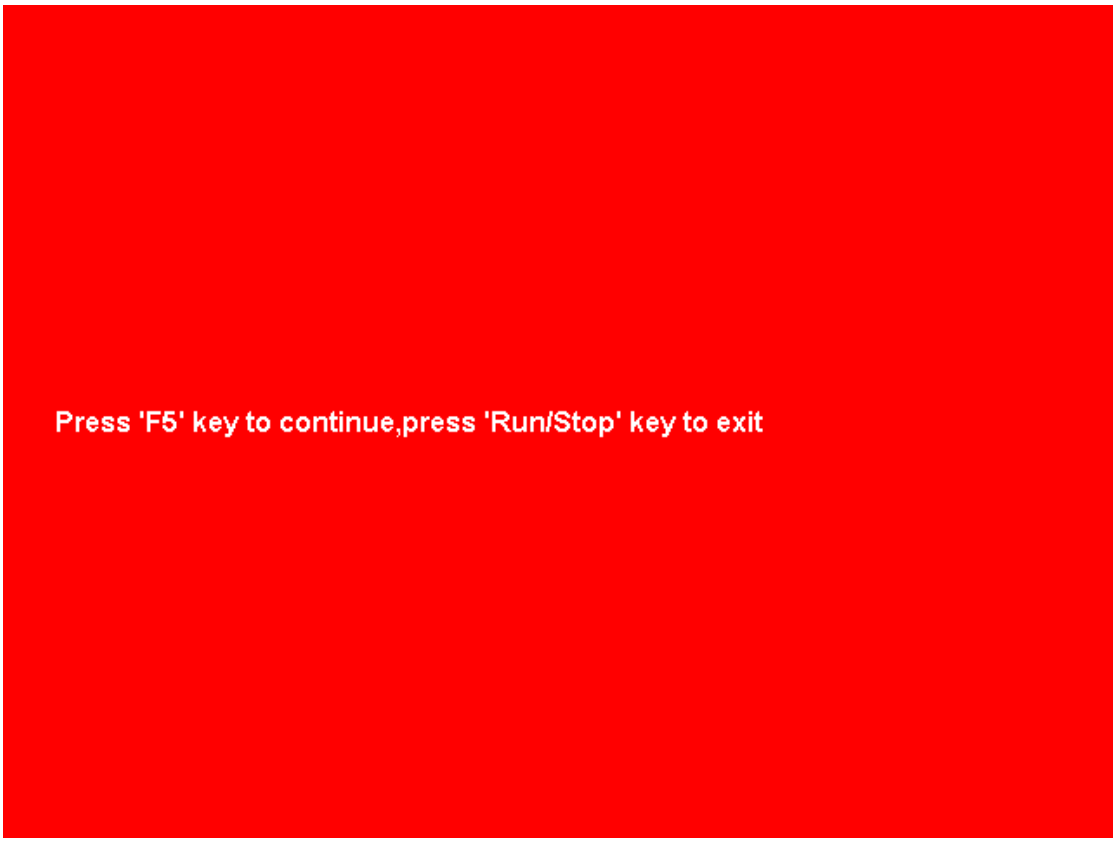


Figure 95 Screen Test

5. Press the **F5** button on the front panel repeatedly as it says in the picture above. The screen displays red, green, blue, and red again. It is easy to check for chromatic aberration, stains, and scratches on the screen.
6. Press **Run/Stop** on the front panel to exit the screen test program.

Keyboard Test

The keyboard test is used to verify that the keys and knobs work properly.

1. Press **Utility** on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** F key to go to the third page of the UTILITY function menu.
3. Press the **Do Self Test** F key to enter the SELFTEST function menu.
4. Press the **Keyboard Test** F key to enter the keyboard test interface, as the picture shown below.

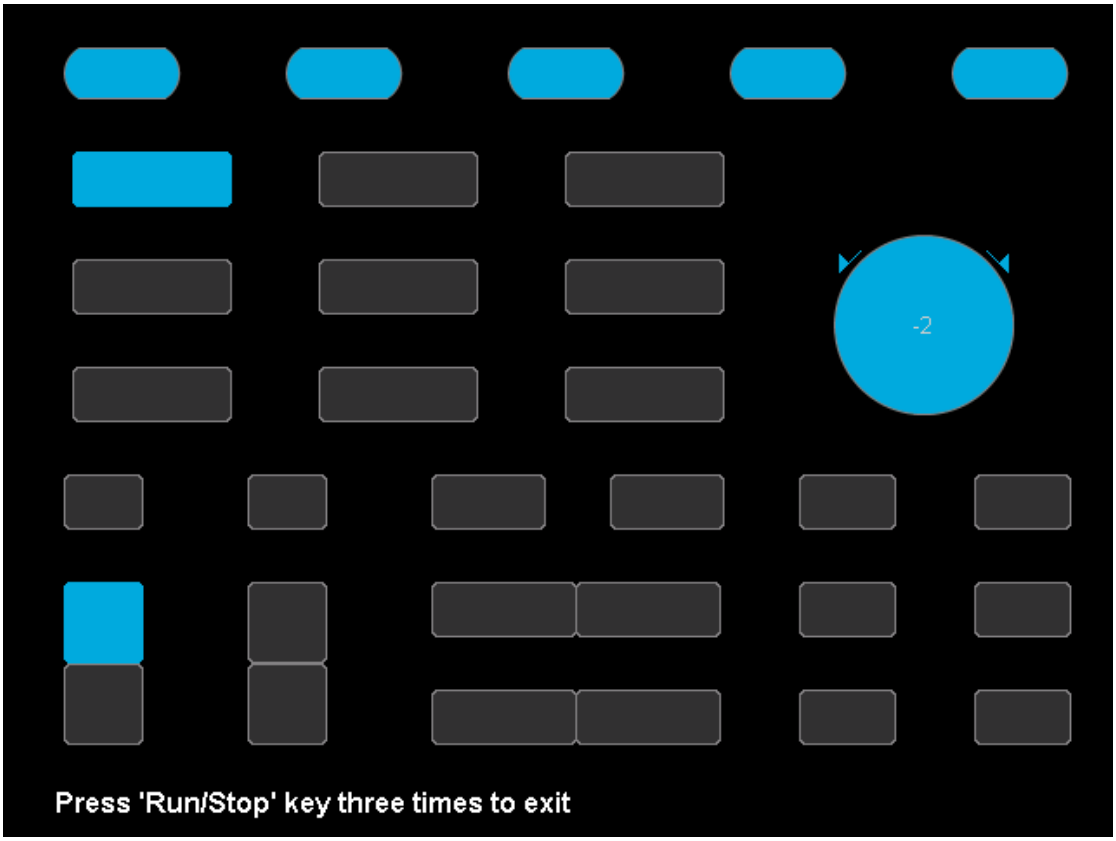


Figure 96 Keyboard Test

To perform the knobs and the buttons test:

5. Knobs test: the default value is 0. Turn the knob left to increase the value, and right to decrease; push the knob to set the value to 0.
6. Keys test: The first time you press a key, it will light up, and pressing it again will turn it off. Test every button randomly to ensure they all light up and turn off.
7. Press **Run/Stop** 3 times to exit the keyboard test program.

LED Test

The LED test is used to test that the backlights behind the buttons work properly.

1. Press **Utility** on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** F key to go to the third page of the UTILITY function menu.
3. Press the **Do Self Test** F key to enter the SELFTEST function menu.
4. Press the **LED Test** F key to enter the keyboard test interface, as below.

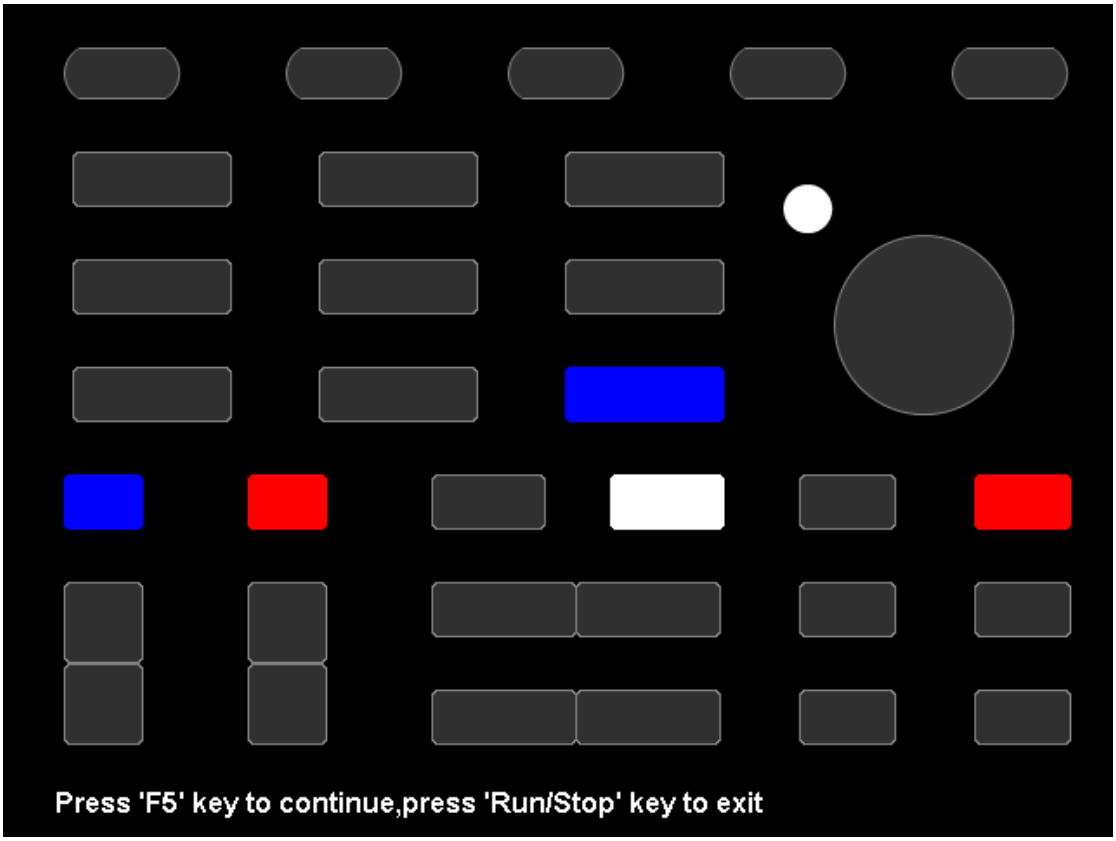


Figure 97 LED Test

5. Following the prompts on the screen, press the **F5** button repeatedly to light the button lights one by one. The first that **Run/Stop** lights， it displays as yellow, and the second time it will light up red. At the end, all of the lights will be on at the same time.
6. Press the **Run/Stop** button to exit the LED test program.

Screen Saver

When the oscilloscope enters an idle state and is not used for a certain length of time, the screen saver will be activated.

To set the screen saver delay time:

1. Press **Utility** on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** F key to go to the third page of the UTILITY function menu.
3. Press the **Screen Saver** F key; and then turn the **Universal Knob** to select the desired screen saver time. The screen saver time can be set to **Off** (default), **1min**, **5min**, **10min**, **30min**, and **1hour**.

Figure 98 Screen Saver Interface

4. When the screensaver is active, press any button on the front to wake the screen.

Date/Time

The oscilloscope includes a clock/calendar. You can update the date and time. After resetting the oscilloscope, you will need to reset the system time.

1 Press **Utility** → **NextPage2/3** → **Date/Time** to enter the DATA/TIME function menu.
2 Press the **Display** F key to select **On** to display the date and time.

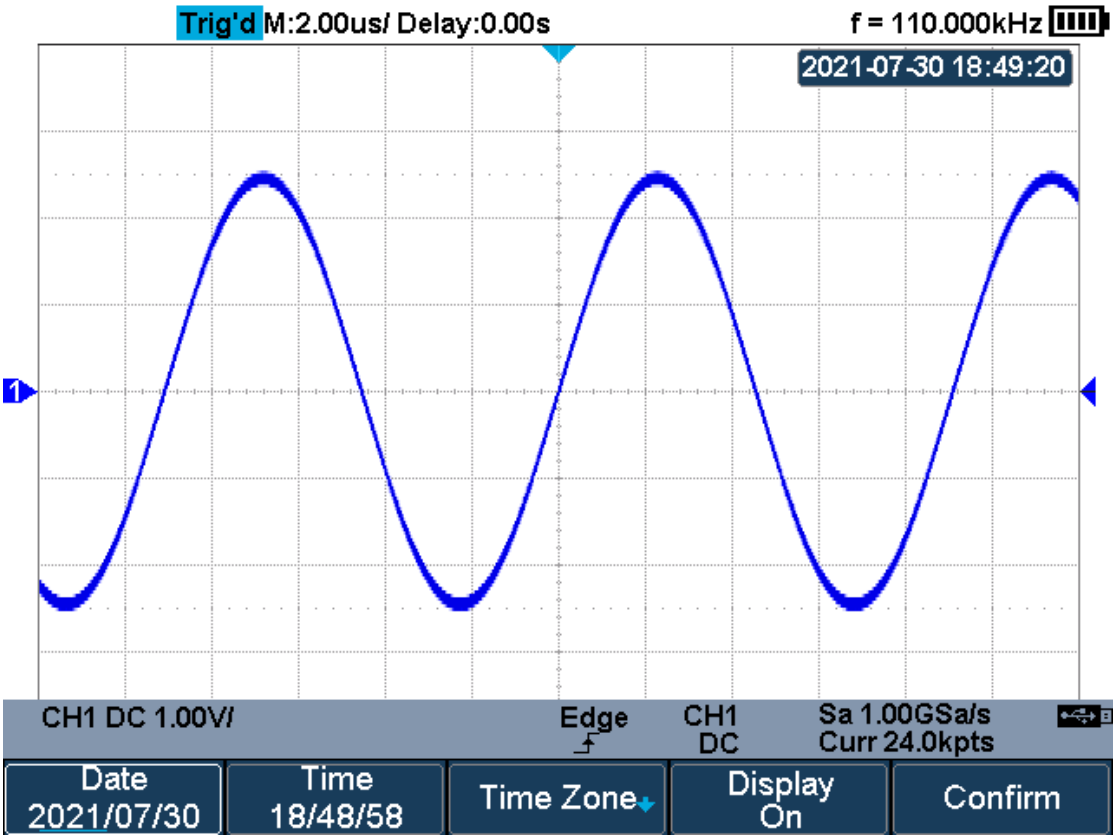


Figure 99 Date/Time Function Interface

Set Date/Time

Press the **Date/Time** F key to enter the Data/Time setting menu. Press the **Date** F key and press the **Universal Knob** to cycle between year, month, and day (the selected value will be underlined with blue). Then rotate the **Universal Knob** to change the value. Modifying the time is similar to the date.

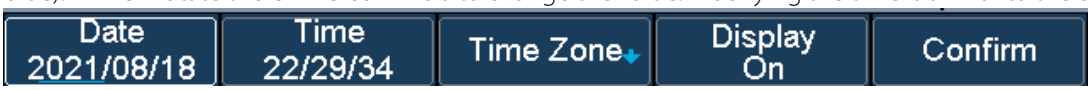


Figure 100 Setting Date/Time

Set Time Zone

1 Press the **Time Zone** F key to enter the Time Zone function menu.
2 Rotate the **Universal Knob** to select the time zone.
3 Press the **Confirm** F key to confirm the selection.

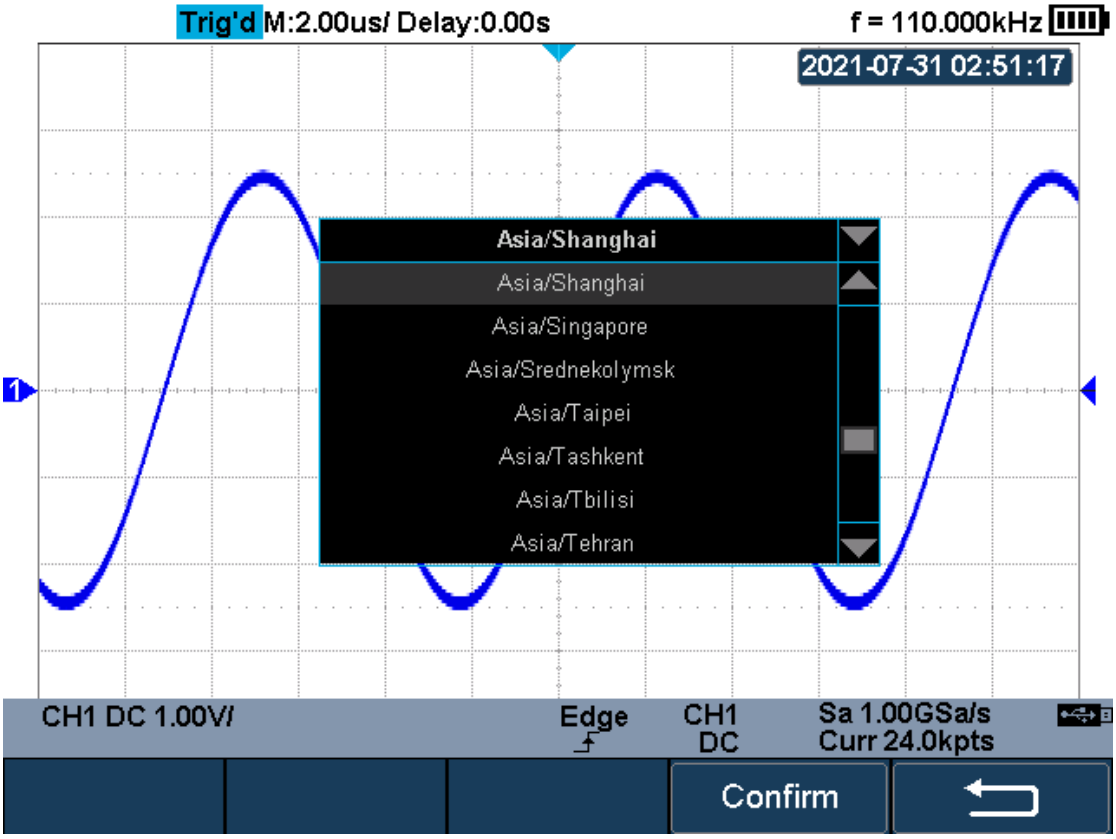


Figure 101 Time Zone Setting Interface

Reference Position

The reference position setting determines the physical point that the oscilloscope uses during vertical and horizontal scale changes. In some situations, it is more convenient to use a fixed position on the display.

Press **Utility** on the front panel and then press the **Reference Pos.** F key to enter the Reference POS menu**.**

Press the **Vertical** F key to select **Fixed Offset** or **Fixed Position**.
Fixed Position: (Default) The oscilloscope will keep the vertical offset level indicator stationary when the vertical gain is changed.
Fixed offset: The oscilloscope will have the vertical offset level indicator move with the actual voltage level when the vertical gain is changed.

Press the **Horizontal** F key to select **Fixed Delay** or **Fixed Position**.
Fixed Position: (Default) When the timebase changes, the oscilloscope will keep the horizontal offset indicator stationary.
Fixed Delay: When the timebase changes, the oscilloscope will have the horizontal offset indicator move with the trigger point.

Search

The oscilloscope includes a search function. It can search for specified events in the acquired data. Matches are displayed with a black triangle symbol. In YT mode or Roll mode with the acquisition in stop, the maximum number of matches is 600. In Roll mode with acquisition in run, the maximum number of matches is unlimited. The waveform can be zoomed while Search is enabled.

Setting

1. Press **Shift** and **Scope** on the front panel to enter the SEARCH function menu.
2. Press the **Mode** F key and then use the **Universal Knob** to select the desired search type. There are five search types: Edge, Slope, Pulse, Interval and Runt.

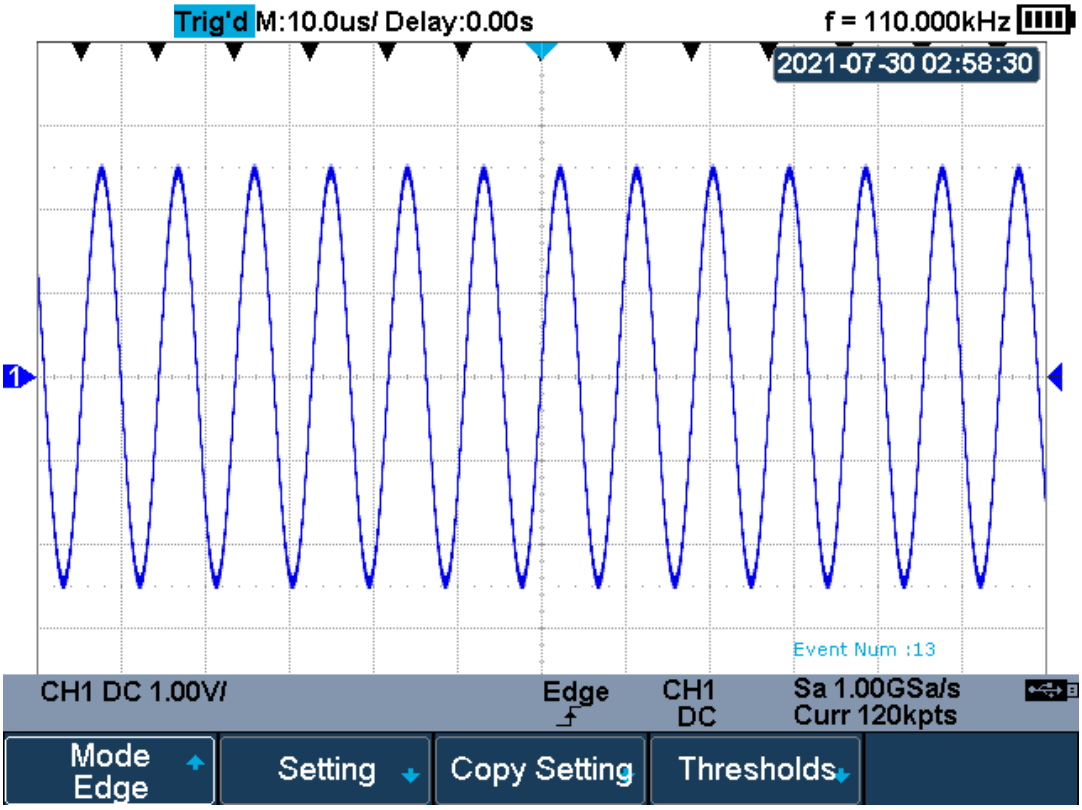


Figure 102 Search Menu

3. Press the **Setting** F key to enter the SETTING function menu. The setting menu is different for every search type. The details show as the following table.

Search Mode	Setting Menu Description
Edge	Slope includes Rising, Falling, Either.
Slope	Slope includes Rising, Falling Limit Range includes four types: <=, >=, [--,--] and --] [--. You can select the desired type and then input the time value.
Pulse	Polarity includes Positive and Negative

	Limit Range includes four types: <=, >=, [--,--] and --] [--. You can select the desired type and then input the time value.
Interval	Slope includes Rising, Falling Limit Range includes four types: <=, >=, [--,--] and --] [--. You can select the desired type and then input the time value.
Runt	Polarity includes Positive and Negative Limit Range includes four types: <=, >=, [--,--] and --] [--. You can select the desired type and then input the time value

4. Press the **Copy Setting** F key to enter COPY function menu.

- **Copy from Trig**: Copy the trigger setup for the selected search type to the search setup.
- **Copy to Trig**: Copy the setup for the selected search type to the same trigger type.
- **Cancel Copy**: Undo a copy.

5. Press the **Thresholds** F key to enter THRESHOLDS function menu to set channel search thresholds.

Results

When the acquisition is started, "EVENT NUM: 7" means the total number of events detected.

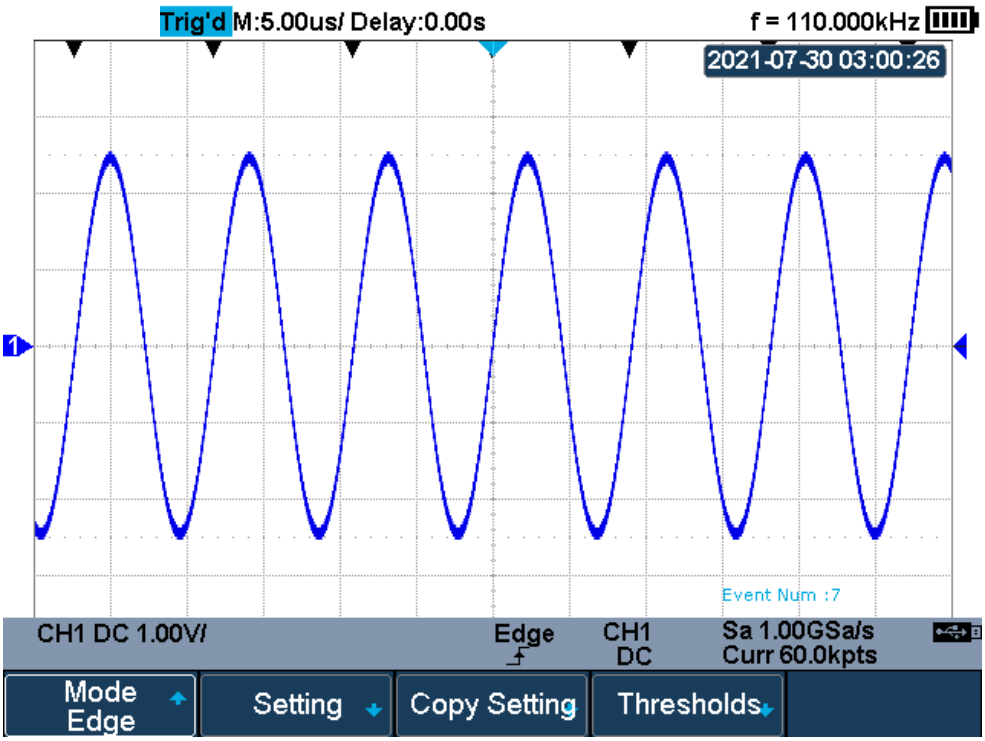


Figure 103 Search in Run

When the acquisition is stopped, "EVENT NUM: 4/7" means current event number/total number of events. The current event is the closest event to the middle of the screen.

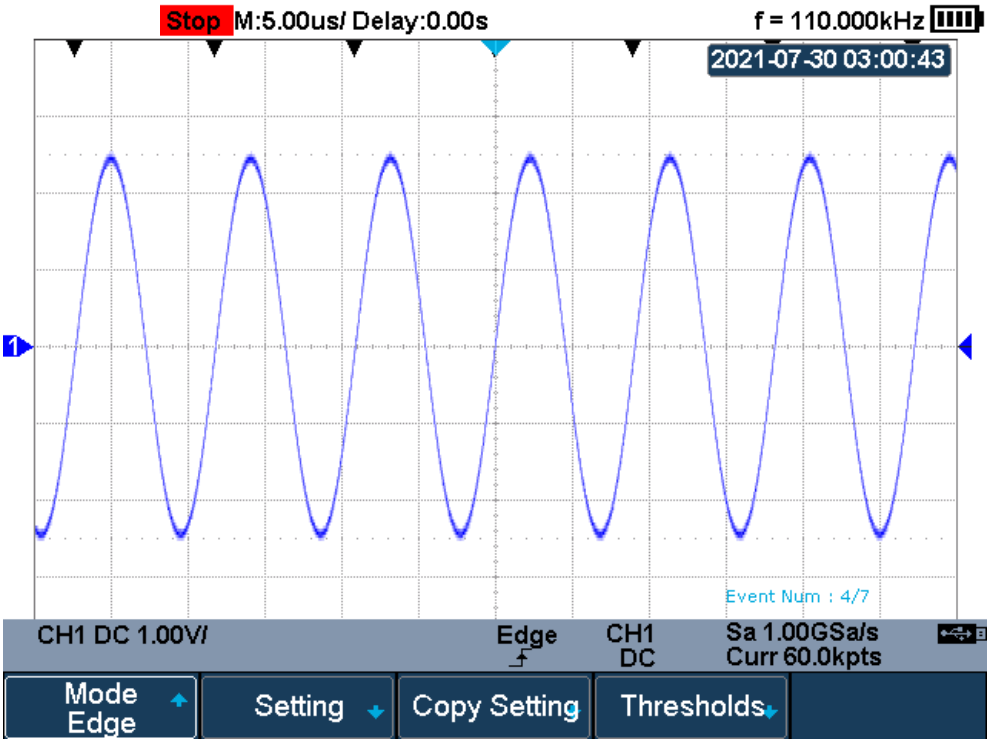


Figure 104 Search in Stop

Navigate

The oscilloscope offers three types of navigation: Search Event, Time, and History Frame.

Time Navigate

1. Press **Shift** and **Recorder** on the front panel to enter the NAVIGATE function menu.
2. Press the **Type** F key In the NAVIGATE function menu, then select **Time**.
3. There are two ways to navigate time.
   a. Press the **Time** F key, then, turn the **Universal Knob** to select the desired value or press the **Universal Knob** and enter the value by the popup keyboard.
   b. Press the F keys ◀ ⏸ ▶ to play backward, stop, or play forward in time. You can press the ◀ or ▶ key multiple times to speed up the playback. There are three speed levels: Low Speed, Medium Speed, and High Speed.

History Frame Navigate

You can use the navigation controls to play through the acquired frames when the History function is enabled.

1. Press **Shift** and **Recorder** on the front panel to enter the NAVIGATE function menu.
2. Press the **Type** F key in the Navigate Menu, then select **History Frame**.
3. Press the **Frame Num** F key, then there are two ways to navigate history frames:
   a. Turn the **Universal Knob** to select the desired number or press the **Universal Knob** then enter the number by the pop keyboard.
   b. Press the F keys ◀ ⏸ ▶ to play backward, stop, or play forward.

Search Event Navigate

When the Search function is enabled and acquisitions are stopped, you can use the navigation controls to go to found search events

1. Press **Shift** and **Recorder** on the front panel to enter the NAVIGATE function menu.
2. Press the **Type** F key in the NAVIGATE function menu, then select **Search Event**.
3. There are two ways to navigate search events.
   a. Press the **Event Num** F key, then, turn the **Universal Knob** to select the desired value or press the **Universal Knob** then enter the value by the pop keyboard.

Press the F keys ◀ ▶ to go to the previous or next search event.

History

The history function can record the waveforms of the input channels until you press **Run/Stop** button. In the Run state, the oscilloscope records input waveform repeatedly; when the memory fills up (reaches the maximal frame), new frames will be recorded over the old frames. The latest frames will be kept.

To use the History function, the HORIZONTAL **Format** must be set to **YT**.

To record and replay waveforms:

1. Press **Shift** and **Meter** on the front panel to enable the History function.
   - If you are in the Run state, the waveform will enter the stop state.
   - If you are in the Stop state, and then enable the History function, the oscilloscope will keep the stop state.
   - Press **Shift** and **Meter** again to turn off History.
2. Press the **List** F key to turn on or off the list display. The list records the timestamp of every frame. It is accurate to microseconds.

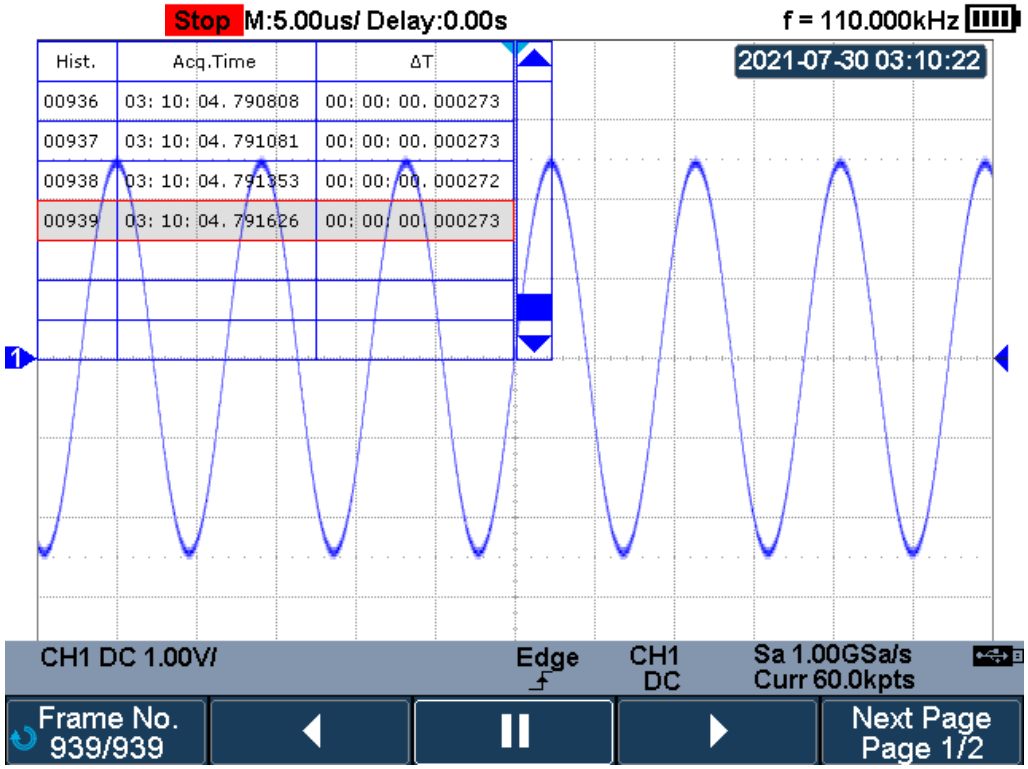


Figure 105 History

3. Press the **Frame No.** F key; then turn the **Universal Knob** to select the frame to display.
   - The Frame format is A/B; A is the frames number that displays on the screen and B is the maximum frame number you can set.
   - The maximum frame number is determined by the current sampling point (**Curr** value) and sampling rate.

- When you press the **Run/Stop** or enable History, you may not get the maximal frames, because the memory is not filled. So if you want to get the maximal frames, please wait long enough to acquire a full allotment of frames.

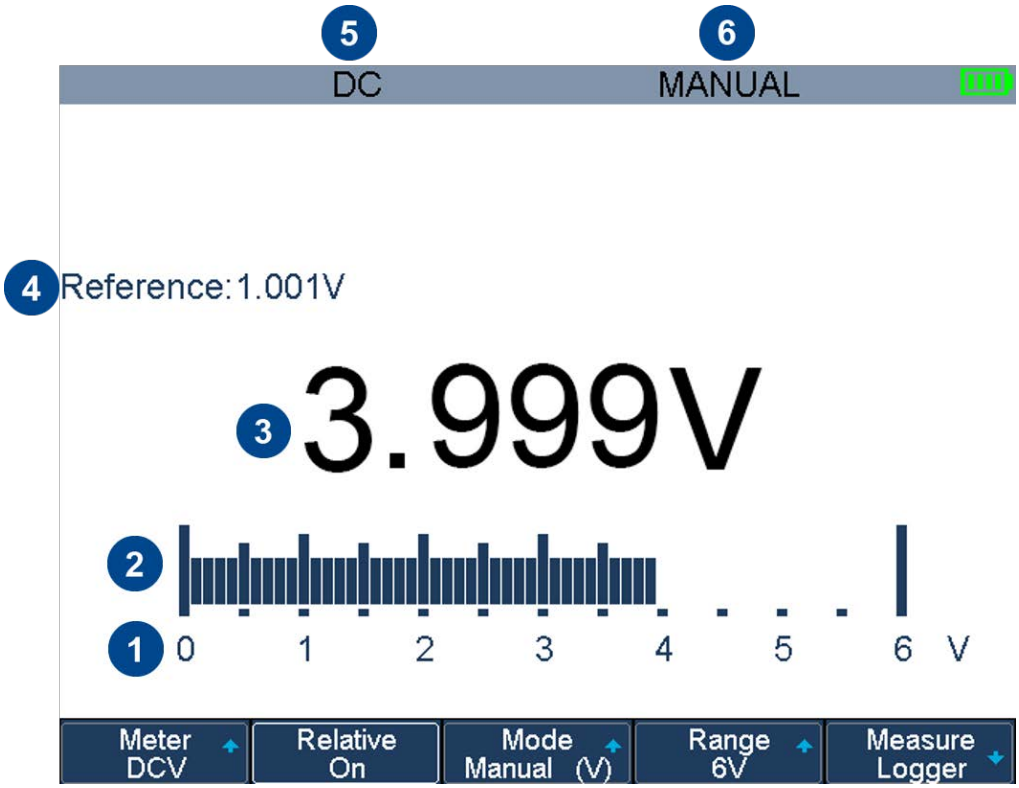
4. Press the F key to replay the waveform from the current frame to 1.
5. Press the F key to stop replay.
6. Press the F key to replay the waveform from the current frame to the last frame.

Meter

This chapter provides a step-by-step introduction to the multimeter functions of AEGIS-OSC-9200 Handheld Digital Oscilloscope.

The digital multimeter provides the following measurements: DC voltage, AC voltage, resistance, diode, continuity, capacitance, DC current, and AC current.

Press **Meter** to bring up the Meter menu.



No.	Description	No.	Description
1	Range of the reading	4	Relative value
2	Bar graph of the measured value	5	Measurement type
3	Reading value	6	Measurement mode

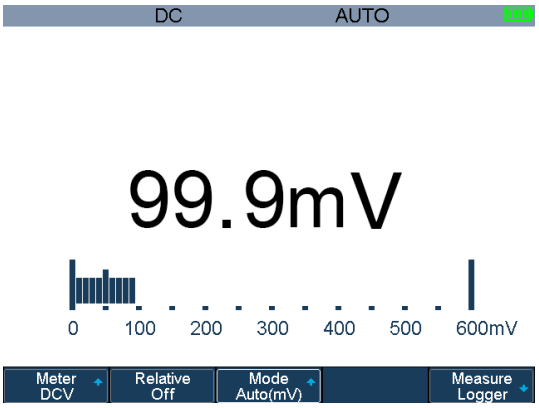
DCV/ACV

The multimeter's maximum voltage depends on the CAT rating of the system to be measured.

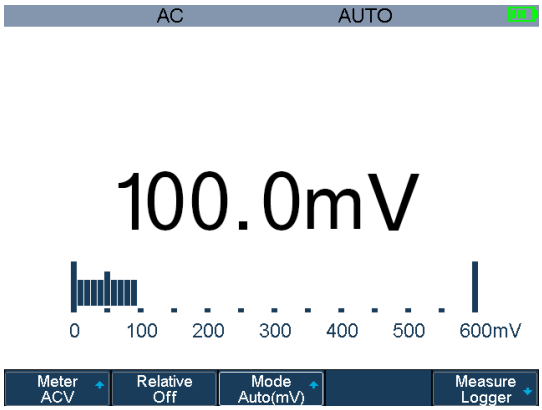
For CATII, the maximum input voltage is 600 VDC/ VAC. For CATIII, the maximum input voltage is 300 VDC/ VAC.

To connect to and measure DC/AC voltage:

1. Press **Meter** to enter multimeter mode and press the **F1** F key (repeatedly if necessary) to choose **DCV** or **ACV** measurement.
2. Insert the red probe to the **V.Ω.C** banana jack input and the black probe to the **COM** jack. Connect the other end of probes to the power or load to be measured.
3. Activate or deactivate **Relative** measurement mode with the **F2** key according to the system.
4. Choose **Manual** or **Auto** range according to the reading. If you are in manual range mode, press the **F4** F key to adjust the measurement range, or press the middle button of the universal knob to quickly switch the range.
5. Read the voltage value.
6. Press **Run/Stop** to pause measurement. The status "HOLD" is displayed on the upper left of the screen. Press **Run/Stop** again to resume measurement.



DC Voltage Measurement



AC Voltage Measurement

Table 3-1 DC and AC Function Menu

Option	Setting	Description
Relative	On	Save the current input value as a reference and record again. The actual value equals the relative value plus the displayed value
	Off	Real value equals measured value
Mode	Auto(V/mV)	Choose the best measurement scale automatically
	Manual(V/mV)	Choose measurement scale manually

Range	60mV 600mV	In manual mode, choose the measurement scale manually. There will be a warring when over the scale.
	600mV 6V 60V 600V 1000V	
Measure Logger	On	Plot the measurement vs. time. See the "Measure Logger" section of "Recorder" for details.

Resistance

To measure resistance:

1. Press **Meter** to enter multimeter mode and press the **F1** F key to select **Res.**
2. Plug the red probe into the **V.Ω.C** banana jack input and the black probe into the **COM** input. Connect the other end of probes to the system to be measured.
3. Activate or deactivate **Relative** measurement mode with the **F2** key according to the system.
7. Choose **Manual** or **Auto** range according to the reading. If you are in manual range mode, press the **F4** F key to adjust the measurement range, or press the middle button of the universal knob to quickly switch the range.
8. Read the resistance value.
9. Press **Run/Stop** to pause measurement. The status "HOLD" is displayed on the upper left of the screen. Press **Run/Stop** again to resume measurement.

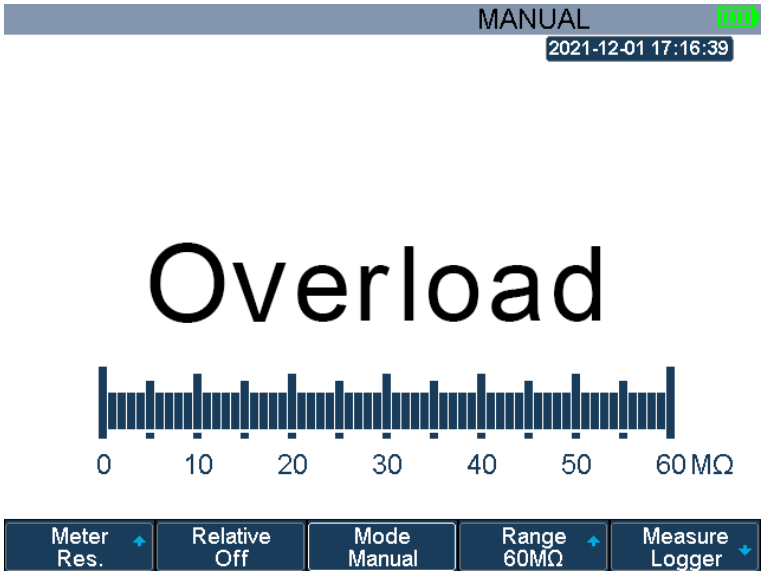


Figure 106 Resistance Measurement

Note: When measuring resistance, be sure that the test system is powered off and that any capacitance is discharged.

Diode

To test a diode:

1. Press **Meter** to enter multimeter mode and press the **F1** F key to choose **Diode**.
2. Plug the red probe into the **V.Ω.C** banana jack input and the black probe into the **COM** input. Connect the other end of probes to the diode to be measured.

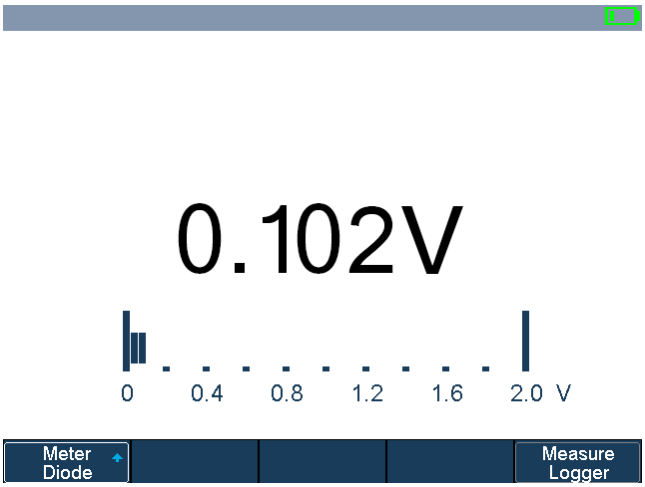


Figure 107 Diode Measurement

Continuity

The continuity test uses the double leads method to measure the resistance of the measured circuit, by passing a low current (about 0.2mA). When the measured resistance in the circuit is lower than the selected one, it is considered to be electrically continuous with the instrument.

To test continuity:

1. Press **Meter** to enter multimeter mode and press the **F1** F key to choose **Continuity** measurement.
2. Plug the red probe into the **V.Ω.C** banana jack input and the black probe into the **COM** input. Connect the other end of the probes to the object to be measured.
3. When the measured resistance is under 50Ω, the multimeter will beep and display the value.
4. When the measured resistance is above 50Ω, the multimeter will not beep, but will display the value.

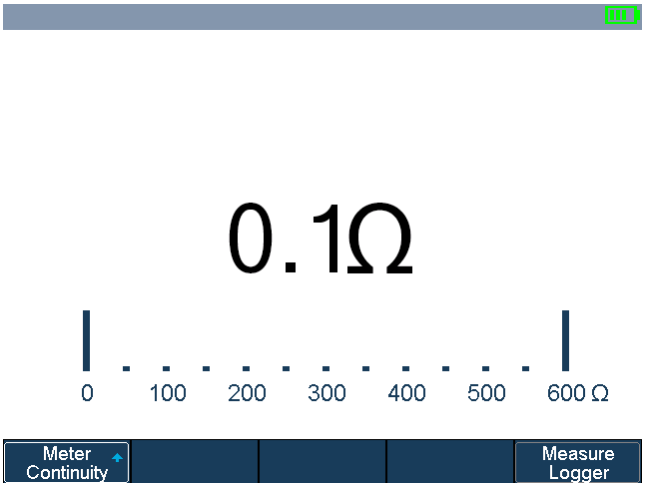


Figure 108 Continuity Measurement

Capacitance

The multimeter can measure capacitance up to 400μF. To measure capacitance:

1. Press **Meter** to enter multimeter mode and press the **F1** F key to choose **Cap.** measurement.
2. Plug the red probe into the **V.Ω.C** banana jack input and the black probe into the **COM** input. Connect the other end of the probes to the object to be measured.
3. Activate or deactivate **Relative** measurement if necessary.
4. Read the measured value.

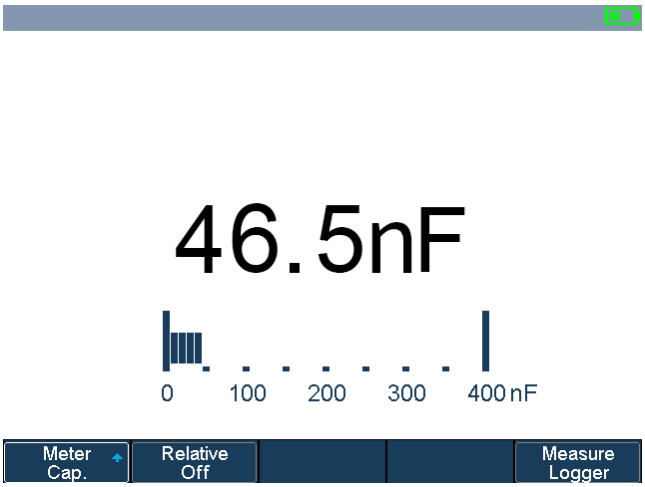


Figure 109 Capacitance Measurement

DCI/ACI

The AEGIS-OSC-9200 includes two accessories for measuring current, the SCD600MA and the SCD10A. For low currents, up to 600mA, use the SCD600MA. For currents up to 10A, use the SCD10A.

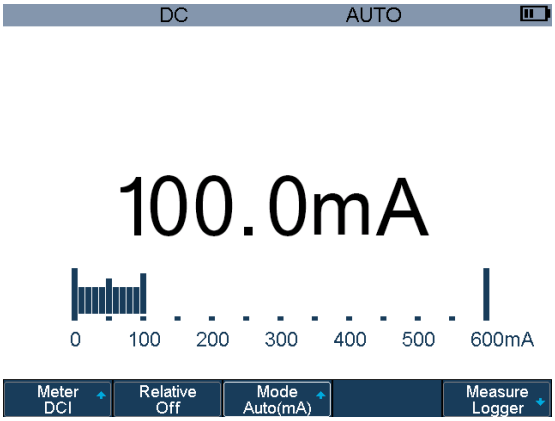
When measuring current with either accessory, the voltage of the measured circuit must not exceed 60VDC/ VAC.

SCD10A

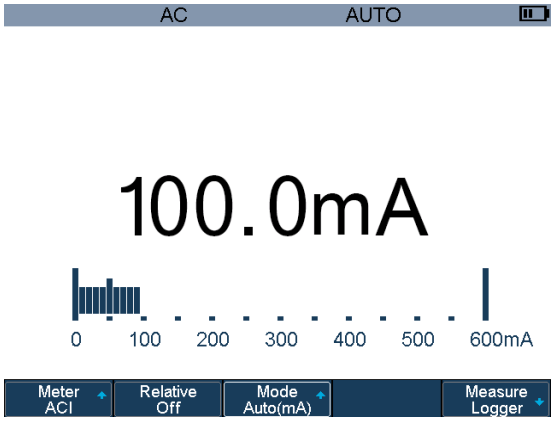
SCD600MA

To measure DC or AC current:

1. Press **Meter** to enter multimeter mode, press the **F1** F key to choose **DCI**/**ACI** measurement.
2. Insert the SCD10A or SCD600MA to the Multimeter inputs. If you are not sure how large the current will be, start with the SCD10A, and if it is under 600 mA, switch to the SCD600MA.
3. Insert the red probe to the **"+"** banana jack input and the black probe to the **"-"** input. Connect the other end of probes in series with the system to be measured.
4. Activate or deactivate **Relative** measurement if necessary.
5. Choose **Manual** or **Auto** range according to the reading. If you are in manual range mode, press the **F4** F key to adjust the measurement range, or press the middle button of the universal knob to quickly switch the range.
6. Read the current value.
7. Press **Run/Stop** to pause the meter measurement. The status "HOLD" will be displayed on the upper left of the screen. Press **Run/Stop** again resume measurement.



DC Current Measurement



AC Current Measurement

Recorder

The AEGIS-OSC-9200 includes a Sample Logger and Measure Logger, which record, respectively, waveform and Measure data over time. The loggers are mainly useful for relatively low-frequency signals. The sample logger can capture at most 25000 measurements per second, and the measure logger can record 10 per second. The loggers will not efficiently capture high frequency signals.

Press **Recorder** to enter the RECORDER function menu.

Figure 110 Recorder Menu

Sample Logger

The Sample logger can record points from waveforms in real time at equal intervals to allow long-term observation of low frequency signals. The recorded data can be stored in the internal storage or on an external storage device. After stopping recording, users can replay the waveform on the oscilloscope or export the recorded data for analysis on a PC.

Press the **Sample Logger** F key to enter the sample logger interface. At this time, most of the buttons on the front panel will not respond, except for the menu F keys, **Meter**, **Scope**, **Hide Menu** and the **Universal Knob**. Press the **Exit** F key to exit the sample logger.

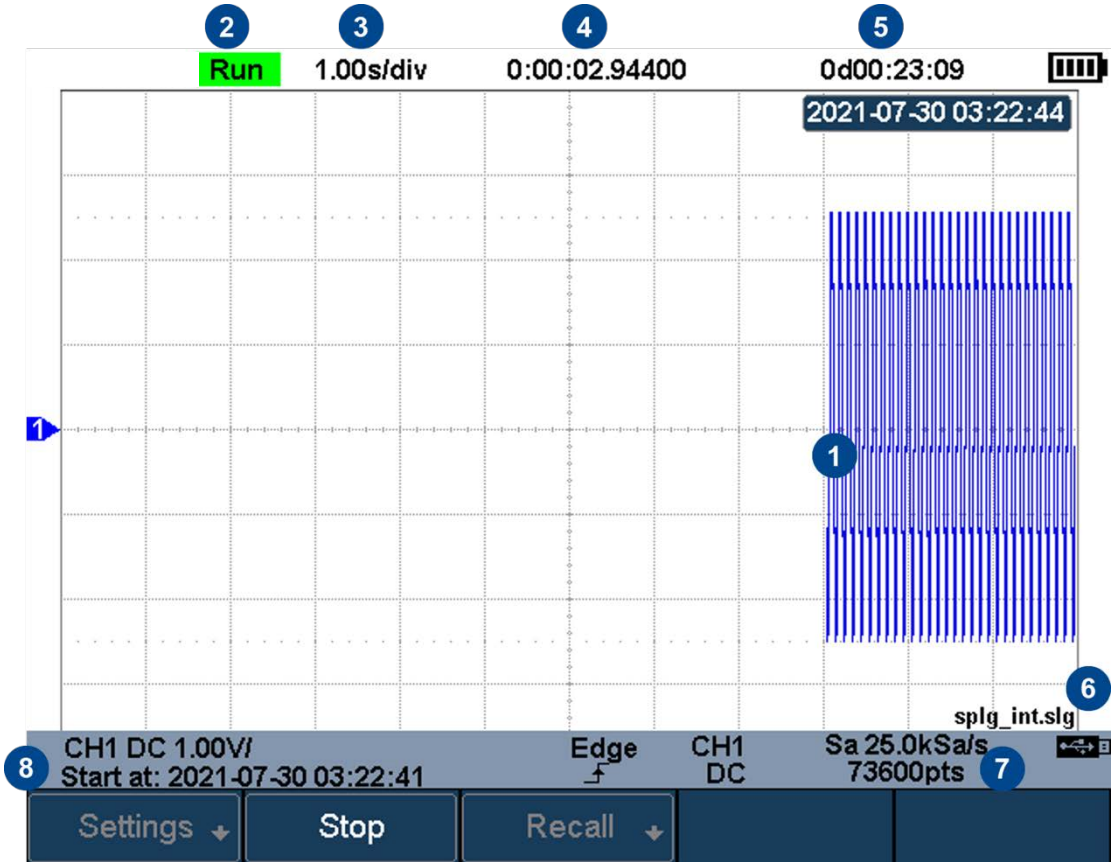


Figure 111 Sample Logger Interface

No.	Description
1	Recorded waveform
2	Recording status （Run/Stop）
3	Horizontal scale
4	Recorded time
5	Remaining recordable time
6	Storage location
7	Sample rate and recorded points of waveform
8	Start time

Record Waveform

1 Press the **Record** F key to enter the RECORD function menu.

2 Press the **Setting** F key to enter the SETTINGS function menu. Press the **Rate** F key to choose the sampling rate of the waveform (from 1 to 25.0k Sa/s). Press the **Record to** F key to set the storage location of the recorded data:

- **Internal**: Record the data to internal memory. The last recorded internal data will be overwritten. You will be asked to confirm the overwrite when you start to record.
- **External**: Record to an external storage device and set the file name and location.

Figure 112 Settings Interface

3 Press the **Return** F key to return to the RECORD function menu.

4 Press the **Start** F key to start to record, and the record status at the top will display as "Run".

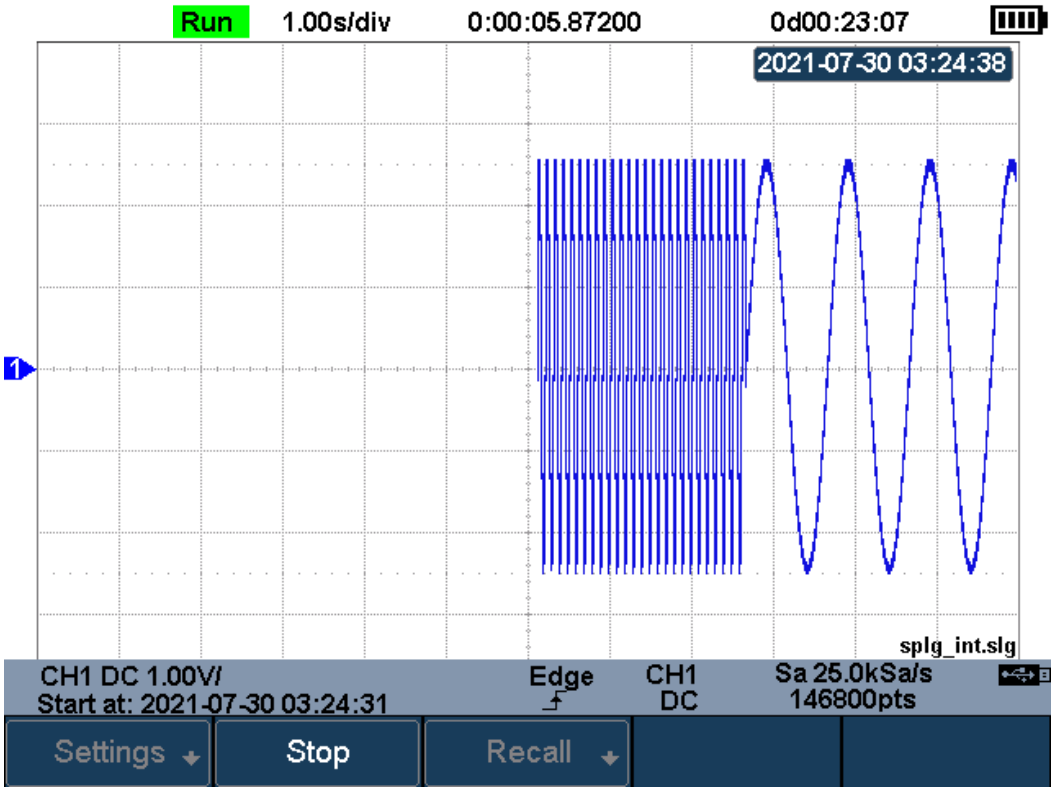


Figure 113 Start Recording Waveform

5 Press the **Stop** F key to stop recording. The record status at the top will display as "Stop". The data is automatically saved in the storage location.

6 Press the **Recall** F key to look back and analyze the waveform.

Note: When recording to a USB drive, the waveform data will be saved to the drive in real time. Please do not remove the disk during the recording!

Recall Recorded Waveform

1 Press the **Recall** F key to enter the RECALL function menu.

2 Press the **Recall from** F key to select the storage path.

3 Press the **Press to Recall** F key to recall the recorded waveform data and enter the CONTROL menu automatically.

Display Control

1 Press the **Horizontal Ref** F key to set the horizontal reference position.
2 Press the **Horizontal Scale** button to zoom the waveform with the horizontal reference as the center. Press the **Horizontal Position** button to move the waveform.
3 Press the **View all** F key to return to the initial configuration to view all waveforms.
4 After recalling the waveform, press the **Cursors** button to turn on the cursors to measure and analyze the waveform on the screen. Please refer to the **Cursors** chapter for details.

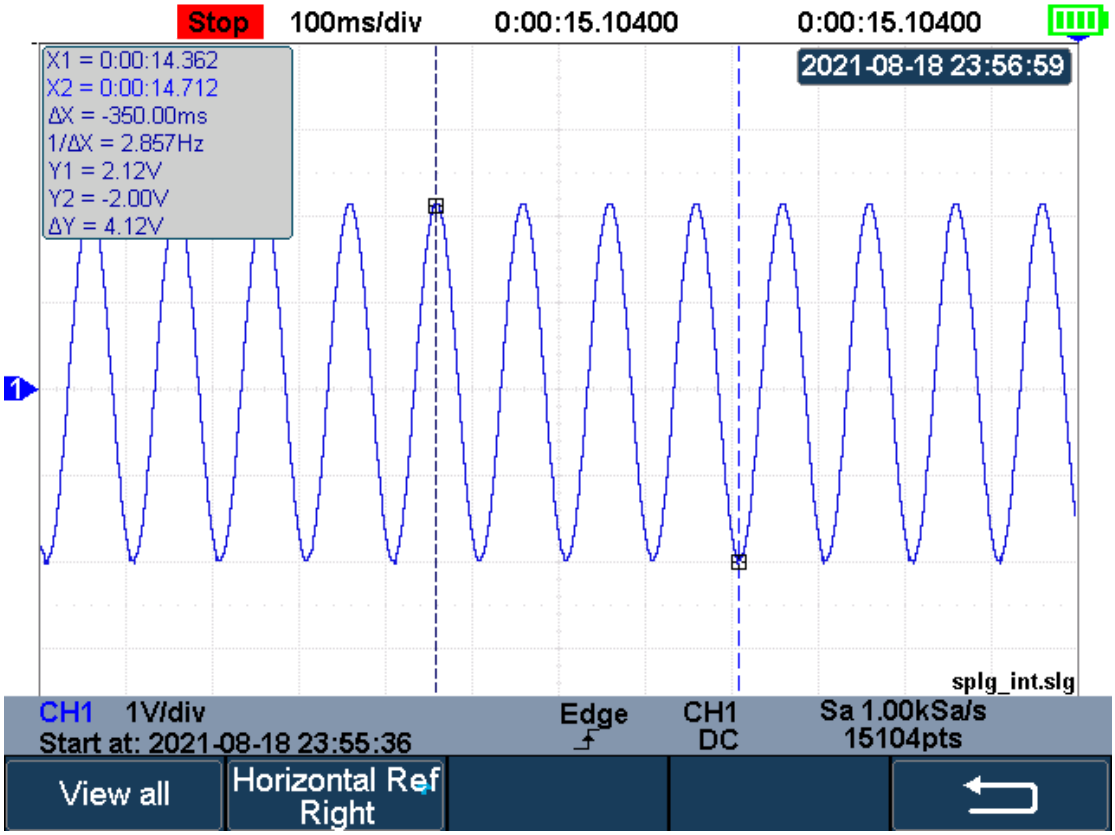


Figure 114 View Recorded Waveform

Manage Internal Sample Logs

1 Press the **Manage** F key to enter the MANAGE function menu.
2 Press the **Export Internal Data** F key to export the internal record to an external storage device. (See the **Save and Recall** chapter for details.) The oscilloscope provides a tool named FileConverter to convert the waveform file (*.slg) to CSV format. You can export it from the Save/Recall menu (Save, Type: FileConverter).
3 Press the **Delete Internal Data** F key to delete the internal record.

Measure Logger

The Measure Logger can record the measured values of waveforms in real time to allow long term measurement of low-speed signal. The measurement data is recorded in internal memory, and can be saved to internal storage or a USB drive after stopping recording. The Measure Logger can record up to four measurements at once.

To use the measure logger, first press **Measure** to turn on the measure function (while in Scope mode), and select up to four measurements to display. Then press **Recorder** →**Measure Logger** to turn on the measure logger. When finished, press the **Exit** F key to exit the measure logger.

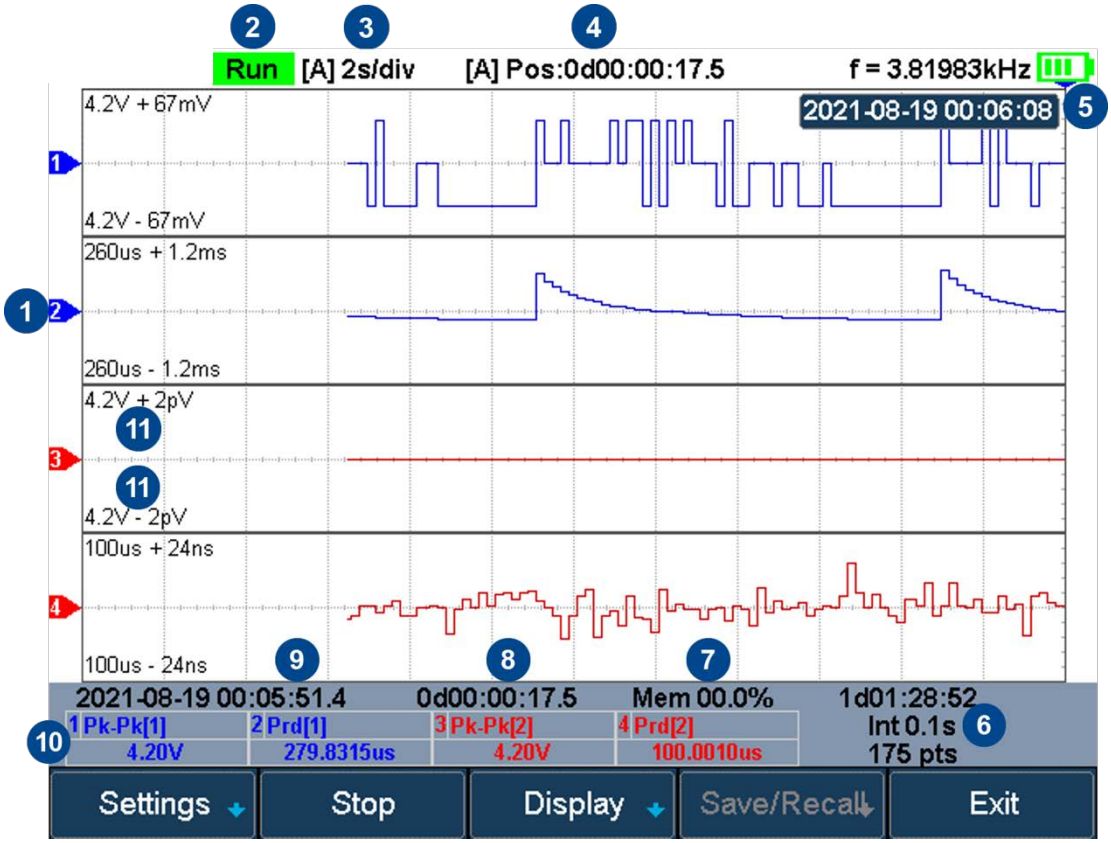


Figure 115 Measure Logger Interface

No.	Description
1	Measurement trace
2	Record status（Run/Stop）
3	[Auto/Manual] Horizontal scale
4	[Auto/Manual] Time value of the first point (centered on horizontal reference position)
5	Horizontal reference position
6	Remaining recordable time, Recording interval, Number of recorded points

7	Used memory
8	Recorded time
9	Start time
10	The current value of measurement items
11	Upper and lower scale of the trace

Start/Stop Recording

1 Press the **Setting** F key to enter the SETTINGS function menu. Set the log interval and then assign up to four traces to one of the Measure parameters you turned on. If you don't want to log them all, set the extra traces to Null.

Figure 116 Settings Interface

2 Press the **Return** F key to return to the previous menu.

3 Press the **Start** F key to start to record. The record status at the top will display as "Run". During the recording, the measure logger automatically adjusts the horizontal scale to show the trace. You can also press the **Horizontal Scale/Position** button to enter manual mode to zoom and move the recording trace.

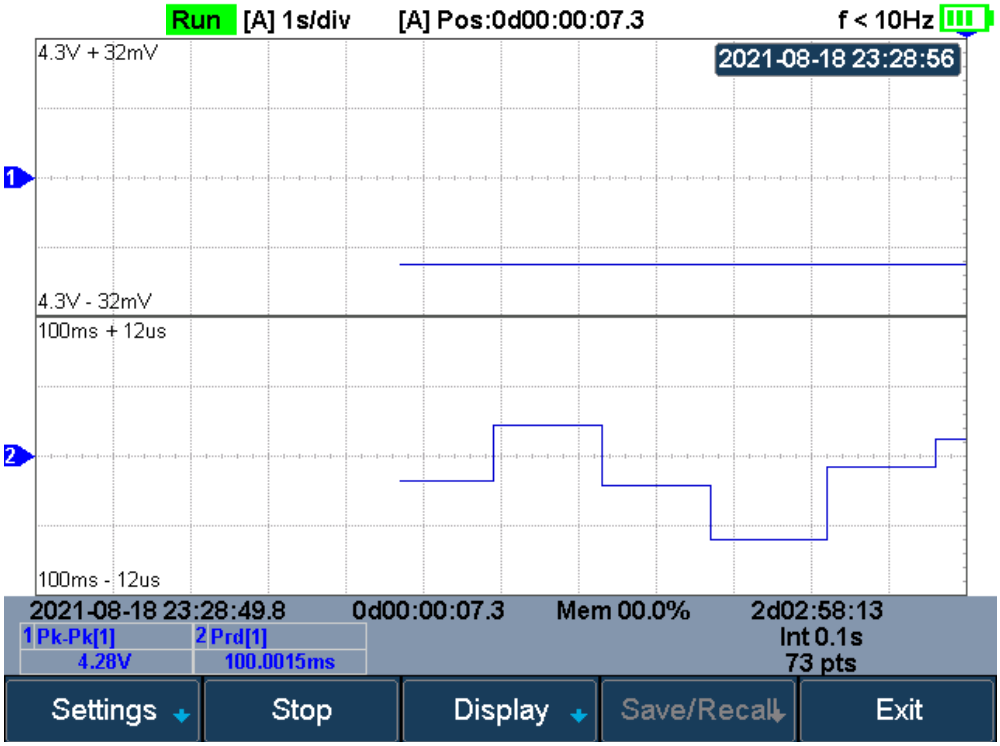


Figure 117 Start to Record Measurement Data

4 Press the **Stop** F key to stop recording. The record status at the top will change to "Stop".

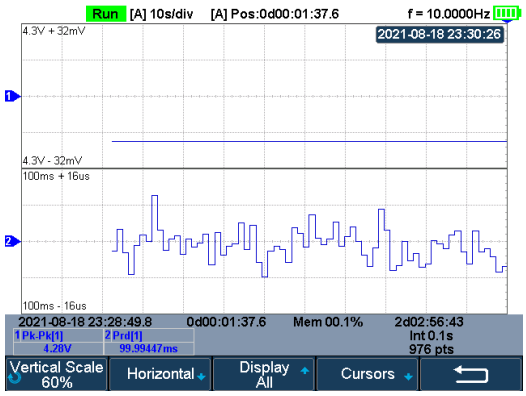
Display Control

1 Press the **Display** F key to enter the DISPLAY settings menu.

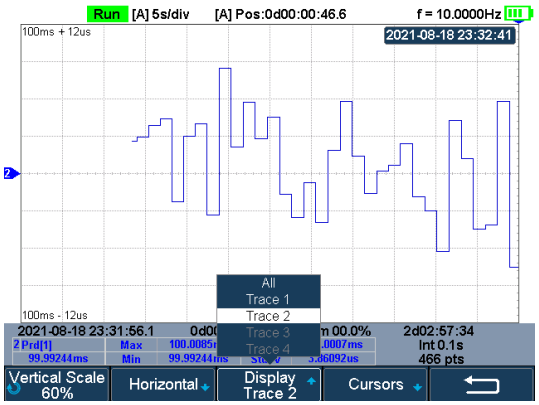
2 Press the **Vertical Scale** F key and turn the **Universal Knob** to set the vertical scale of the trace. You can also adjust the scale through the vertical scale button of the channel (**mV** or **V**) on the front panel.

3 Press the **Horizontal** F key to enter the horizontal display settings menu. You can zoom and move the logger trace and set the horizontal reference position. Press the **Auto Set** F key to reset the horizontal settings.

4 Press the **Display** F key and select **All**, all traces are displayed in the waveform area. When a trace is selected, only it is displayed in the waveform area. Press the channel button corresponding to the trace to switch between all traces and individual traces.



All Traces Display



Single Trace Display

5 Press the **Cursors** F key to enter the cursors function menu. Move the cursor to obtain the measured value of each point.

a. Press the **Status** F key and select **ON** to turn on the cursors. You can also press **Cursors** to turn on it.

b. Press the **Select** F key to select the cursor **T1**, **T2** or **T1-T2**.

c. Press the **Strategy** F key to choose the behavior of cursors when horizontal scale or position is changed.

- **Fixed Position**: cursors remain fixed to the grid position on the display
- **Fixed Time**: the value of cursors remains fixed.

d. Press the **Track Mode** F key to select the track mode of T-Cursors.

- **Normal**: Track the data at the time of T-Cursors
- **Maximum**: Track the maximum value of the data within a pixel where T-Cursors are located.
- **Average**: Track the average value of the data within a pixel where T-Cursors are located.

- **Minimum**: Track the minimum value of the data within a pixel where T-Cursors are located.
- **Peak**: Track the data with the maximum deviation from the overall average value in a pixel where T-Cursors are located. When two cursors are at the same position, T1 will track the maximum value and T2 will track the minimum value.

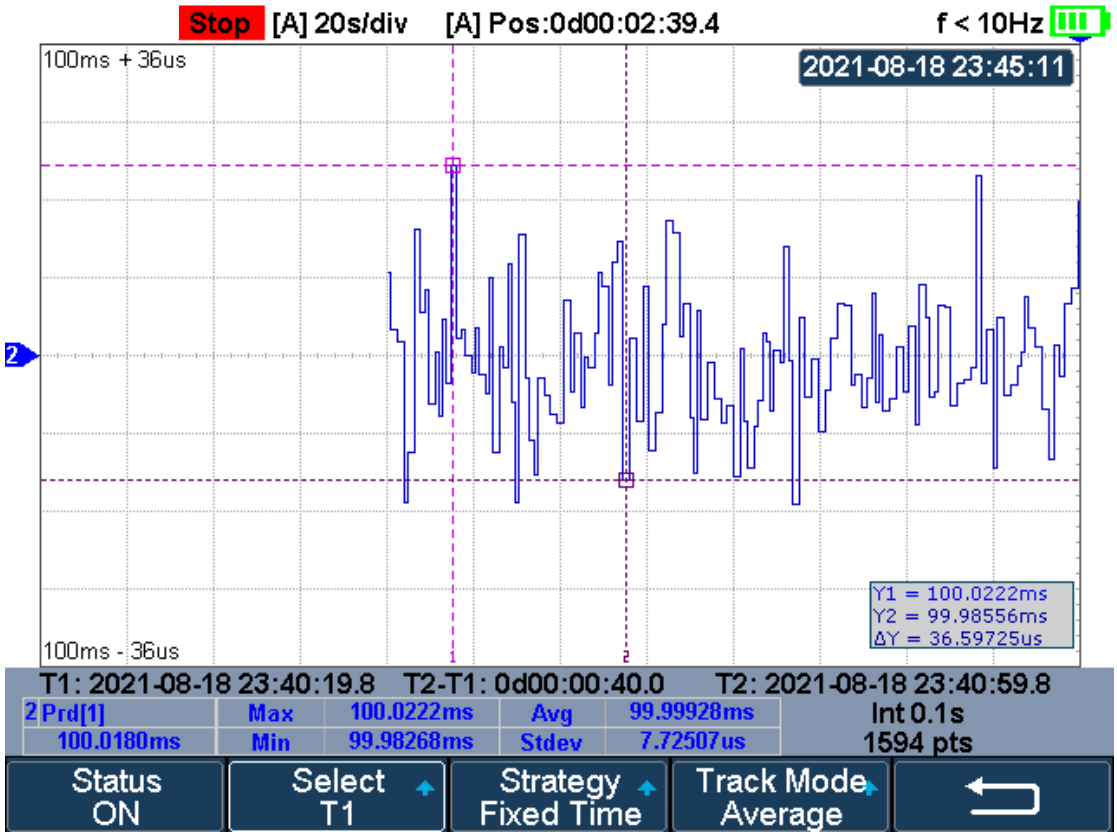


Figure 118 Cursors of Measure Logger

Saving Measure Logs

1 Press the **Save/Recall** F key to enter the SAVE/RECALL function menu.

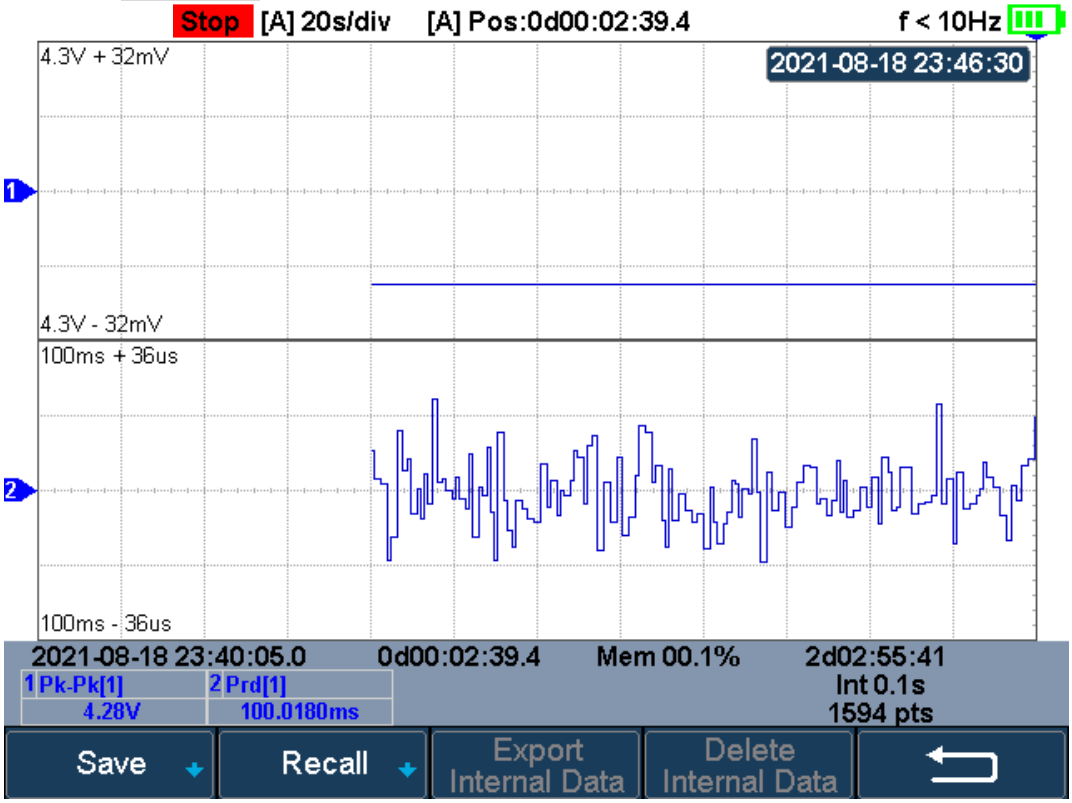


Figure 119 Save/Recall of Measure Logger

2 Press the **Save** F key to enter the SAVE menu.

Figure 120 Save of Measure Logger

3 Press the **Save to** F key to select where to store.

- **Internal**: The measurement data will be saved in binary data format, and the last data stored internally will be overwritten.
- **External**: The measurement data can be saved as binary data(*.mlg), CSV data, or MATLAB data

4 Press the **Press to Save** F key to save.

Recalling Measure Logs

1 Press **Save/Recall** → **Recall** to enter the RECALL function menu.

2 Press the **Recall from** F key to select where to recall from (Internal or External). Only binary files can be recalled from internal storage, while binary files or MATLAB files can be recalled from external. If the current data is not saved, it will be replaced by the recalled data.

3 Press the **Press to Recall** F key to recall.

Manage Internal Measure Logs

Exporting or deleting internal measure logs is similar to "**Manage Internal Record**" of Sample Logger.

Restore Factory Defaults

If you have not used Save/Recall – To Default Key, you can press **Default** twice to restore factory settings. Otherwise, press **Shift** and **Save/Recall**, then press the **Recall** F key. Choose Type: **Factory Default**, and then **Press to Recall**.

Troubleshooting

The most common problems and their solutions are listed below. When you encounter these problems, please follow the steps below. If the problem remains still, please contact Electro Static Technology .

1. The screen is still dark (no display) after powering on:

1) Check whether the power switch was actually switched on.
2) Check whether the power and/or battery are correctly connected.
3) Restart the instrument after finishing the above inspections.

2. The signal is sampled but no waveform of the signal is displayed:

1) Check whether the probe is correctly connected to the cable.
2) Check whether the cable is correctly connected to the BNC (namely channel connector).
3) Check whether the probe is correctly connected to the system be tested.
4) Check whether probe receives the 1 kHz signal from the compensation output terminal (this will tell you whether the problem is in the probe/scope or the tested system)
5) Check whether there is signal generated from the test system.
6) Resample the signal.

3. The tested voltage amplitude is greater or lower than the actual/expected amplitude:

Check whether the attenuation coefficient of the channel complies with the attenuation ratio of the probe. If for example the probe is set to 10X, the channel must also be set to 10X.

4. There is waveform display but not stable:

1) Check the trigger signal source: check whether the **Source** item at the trigger panel is the channel you are trying to measure on.
2) Check the trigger type: General signals should use "Edge" trigger and video signal should use "Video" trigger. Only when the proper trigger type is used can the waveform be displayed stably.
3) Change the trigger holdoff setting.

5. No display after pressing Run/Stop:

Check whether the mode at the trigger panel (TRIGGER) is on "Normal" or "Single" and whether the trigger level exceeds the waveform range. If yes, set the trigger level to the middle or set the mode to "Auto".

Note: using **AUTO** could automatically finish the above setting.

6. The display of waveform is ladder-like:

1) The horizontal time base (seconds/div) might be too low. Increase the horizontal time base to increase the horizontal resolution and improve the display.
2) If the display **Type** is "Vectors", the lines between the sample points may cause a ladder-like display. Setting **Type** to "Dots" should solve the problem.

7. Fail to connect PC through USB:

Check the **IO Setting** in **Utility** to ensure whether the USB device setting matches the device currently connected. If needed, restart the oscilloscope.

8. The USB storage device cannot be recognized:

1) Check whether the USB storage device can work normally.
2) Check whether the USB interface can work normally.
3) Make sure that the USB storage device is a flash drive (solid state). This oscilloscope cannot use external hard drives (HDD).
4) Restart the instrument and then insert the USB storage device to check it.

9. The multimeter measurements aren't correct:

1) Check whether the test circuit's property of interest (voltage/current/etc.) is in the multimeter's range.

AEGIS®

Electro Static Technology™
An ITW Company

info@est-aegis.com | www.est-aegis.com