RIGOL

User's Guide

DS6000 Series Digital Oscilloscope

Aug. 2012

RIGOL Technologies, Inc.

Guaranty and Declaration

Copyright
© 2010 RIGOL Technologies, Inc. All Rights Reserved.

Trademark Information
RIGOL is a registered trademark of RIGOL Technologies, Inc.

Publication Number
UGA06112-1110

Notices
- **RIGOL** products are protected by patent law in and outside of P.R.C.
- **RIGOL** reserves the right to modify or change parts of or all the specifications and pricing policies at company's sole decision.
- Information in this publication replaces all previously corresponding materials.
- **RIGOL** shall not be liable for losses caused by either incidental or consequential in connection with the furnishing, use or performance of this manual as well as any information contained.
- Any part of this document is forbidden to be copied or photocopied or rearranged without prior written approval of **RIGOL**.

Product Certification
RIGOL guarantees this product conforms to the national and industrial standards in China as well as the ISO9001:2008 standard and the ISO14001:2004 standard. Other international standard conformance certification is in progress.

Contact Us
If you have any problem or requirement when using our products, please contact RIGOL Technologies, Inc. or your local distributors, or visit: www.rigol.com.

Safety Requirement

General Safety Summary

Please review the following safety precautions carefully before putting the instrument into operation so as to avoid any personal injuries or damages to the instrument and any product connected to it. To prevent potential hazards, please use the instrument as specified in this manual.

Use Proper Power Cord.
Only use power cords designed for the instrument and authorized by local country.

Ground The Instrument.
The instrument is grounded through the Protective Earth lead of the power cord. To avoid electric shock, make sure that the earth terminal of the power cord is properly connected to the Protective Earth lead before connecting any of the input or output terminals of this instrument.

Connect The Probe Properly.
The earth lead of the probe should be equal to the earth potential. Please do not connect the earth lead to high voltage.

Observe All Terminal Ratings.
To avoid fire or shock hazard, observe all ratings and markers on the instrument and check your manual for detailed information about ratings before connecting the instrument.

Use Proper Overvoltage Protection.
Make sure that no overvoltage (such as that caused by thunderbolt) can reach the product, or else the operator might expose to danger of electric shock.

Do Not Operate Without Covers.
Do not operate the instrument with covers or panels removed.

Avoid Circuit or Wire Exposure.
Do not touch exposed junctions and components when the unit is powered.

Do Not Operate With Suspected Failures.

If you suspect damage occurs to the instrument, have it inspected by qualified service personnel before further operations. Any maintenance, adjustment or replacement of components must be performed by **RIGOL** authorized personnel.

Keep Proper Ventilation.

Inadequate ventilation may cause temperature increase which would damage the instrument. So please keep the instrument well ventilated during operation and inspect the intake and fan regularly.

Do Not Operate In Wet Conditions.

In order to avoid short circuit inside the instrument or electric shock, please do not operate in humid environment.

Do Not Operate in Flammable and Explosive Environment.

In order to avoid damages to the device or personal injuries, please do not operate in flammable and explosive environment.

Keep Product Surfaces Clean and Dry.

To avoid the influence of dust and/or moisture in the air on the performance of the instrument, please keep the surface of the instrument clean and dry.

Protect The Instrument from Static Electricity.

Operate in a static-free area to avoid damages caused by static electricity. Ground both the internal and external conductors of the cable to discharge static electricity before connecting it to the instrument.

Use The Battery Properly.

Do not expose the battery to high temperature or fire. Keep children away from the battery. Improper replacement of battery may cause explosion (warning: lithium battery). Only use batteries specified by **RIGOL**.

Handling Safety

Please handle with care during transportation to avoid slipping of the instrument and damages to buttons, knob, interfaces or other parts on the panels.

Safety Terms and Symbols

Terms in this Manual. The following terms may appear in this manual:

⚠ **WARNING**
Warning statements indicate the conditions or practices that could result in injuries or loss of life.

⚠ **CAUTION**
Caution statements indicate the conditions or practices that could result in damage to this product or loss of data.

Terms on the Product. The following terms may appear on the product:

DANGER   indicates an injury or hazard may immediately happen.
WARNING indicates an injury or hazard may be accessible potentially.
CAUTION   indicates a potential damage to the instrument or other property might occur.

Symbols on the Product. The following symbols may appear on the product:

Hazardous Voltage	Please Refer to Manuals	Protective Earth Terminal	Chassis Ground	Test Ground

Measurement Category

Measurement Category

DS6000 series digital oscilloscopes can make measurements in Measurement Category I.

⚠ **WARNING**
This oscilloscope can only be used for measurements within its specified measurement categories.

Measurement Category Definitions

Measurement category I is for measurements performed on circuits not directly connected to MAINS. Examples are measurements on circuits not derived from MAINS, and specially protected (internal) MAINS derived circuits. In the latter case, transient stresses are variable; for that reason, the transient withstand capability of the equipment is made known to the user.

Measurement category II is for measurements performed on circuits directly connected to the low voltage installation. Examples are measurements on household appliances, portable tools and similar equipment.

Measurement category III is for measurements performed in the building installation. Examples are measurements on distribution boards, circuit-breakers, wiring, including cables, bus-bars, junction boxes, switches, socket-outlets in the fixed installation, and equipment for industrial use and some other equipment, for example. Stationary motors with permanent connection to the fixed installation.

Measurement category IV is for measurements performed at the source of the low-voltage installation. Examples are electricity meters and measurements on primary overcurrent protection devices and ripple control units.

Ventilation Requirement

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

⚠️ **WARNING**
Inadequate ventilation may cause temperature increase which would damage the instrument. So please keep the instrument well ventilated during operation and inspect the intake and fan regularly.

Working Environment

Temperature
Operating: 0℃ to +50℃
Non-operating: -20℃ to +70℃

Humidity
Under +35℃**:** ≤90％ relative humidity
+35℃ to +40℃: ≤60％ relative humidity

⚠ **WARNING**
To avoid short circuit inside the instrument or electric shock, please do not operate in humid environment.

Altitude
Operating: less than 3 km
Non-operating: less than 15 km

Installation (overvoltage) Category
This product is powered by MAINS conforming to installation (overvoltage) category II.

⚠ **WARNING**
Make sure that no overvoltage (such as that caused by thunderbolt) can reach the product, or else the operator might expose to danger of electric shock.

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

Pollution Degree

Degree 2

Pollution Degree Definitions

Pollution degree 1: No pollution or only dry, non-conductive pollution occurs. The pollution has no influence. For example: a clean room or air-conditioned office environment.

Pollution degree 2: Normally only dry, non-conductive pollution occurs. Occasionally a temporary conductivity caused by condensation may occur. For example: general indoor environment.

Pollution degree 3: Conductive pollution occurs, or dry, non-conductive pollution occurs which becomes conductive due to condensation which is expected. For example: Sheltered outdoor environment.

Pollution degree 4: Pollution that generates persistent conductivity through conductive dust, rain, or snow. For example: outdoor locations.

Safety Class

Class 1 – Grounded Product

General Care and Cleaning

General Care:
Do not store or leave the instrument in where the instrument will be exposed to direct sunlight for long periods of time.

Cleaning:
Clean the instrument regularly according to its operating conditions. To clean the exterior surface, perform the following steps:
1. Disconnect the instrument from all power sources.
2. Clean the loose dust on the outside of the instrument with a lint- free cloth (with a mild detergent or water). When cleaning the LCD, take care to avoid scarifying it.

⚠️ **CAUTION**
To avoid damages to the instrument, do not expose it to liquids which have causticity.

⚠️ **WARNING**
To avoid injury resulting from short circuit, make sure the instrument is completely dry before reconnecting it to a power source.

Environmental Considerations

The following symbol indicates that this product complies with the requirements of European Union according to Directives 2002/96/EC on waste electrical and electronic equipment (WEEE).

Product End-of-Life Handling

The equipment may contain substances that could be harmful to the environment or human health. In order to avoid release of such substances into the environment and harm to human health, we encourage you to recycle this product in an appropriate system that will ensure that most of the materials are reused or recycled appropriately. Please contact your local authorities for disposal or recycling information.

DS6000 Series Overview

Being a multifunctional and high performance digital oscilloscope, DS6000 delivers perfect combination of easy-to-use and powerful features to help users to fulfill their tasks (such as quick measurement of parameters, remote interface configuration and easy printer connection) more quickly.

Main Features:
- 1 GHz or 600 MHz bandwidth.
- 2-channel or 4-channel model.
- 5 GSa/s maximum real-time sample rate, 100 GSa/s maximum equivalent sample rate, 180,000 wfms/s (dots display) maximum waveform refresh rate.
- 140 Mpts maximum memory depth (standard).
- Ultra Vision technology.
- 10.1 inches, WVGA (800*480) 160,000 color TFT LCD, with ultra-wide screen, vivid picture, low power consumption and long service life.
- Enable to identify probe type automatically.
- Adjustable brightness of analog channel waveform.
- Auto setting of waveform display (**Auto**).
- Various trigger functions including multiple protocol triggers.
- Standard Parallel decoding and multiple serial decoding options.
- Auto measurements of 24 waveform parameters and measurement functions with statistic.
- Real-time waveform recording, playback and analysis functions.
- Precise delayed sweep function.
- Built-in FFT function.
- Pass/fail test function.
- Multiple waveform math operation functions.
- Standard configuration interfaces: USB Device, dual USB Host, LAN and GPIB (optional).
- Support USB storage device storage and PictBridge printer.
- Conform to LXI-C instrument standards. Enable quick, economic, efficient creation and reconfiguration of test system.
- Support remote command control.
- Embedded help to facilitate information access.
- Support multiple languages and Chinese/English input.

- Provide shortcut keys for measurement, storage and print.
- Provide more than 2-hour lithium battery power supply (optional) to facilitate on-field test and use.
- Built-in 1 GBytes flash memory.

Document Overview

1 Quick Start
Describe the preparations before using the instrument and generally introduce the instrument.

2 To Set the Vertical System
Introduce the functions of the vertical system of the oscilloscope.

3 To Set the Horizontal System
Introduce the functions of the horizontal system of the oscilloscope.

4 To Set the Sample System
Introduce the functions of the sample system of the oscilloscope.

5 To Trigger the Oscilloscope
Introduce the trigger mode, trigger coupling, trigger holdoff, external trigger and various trigger types of the oscilloscope.

6 To Make Measurements
Introduce how to make math operation, cursor measurement and auto measurement.

7 Protocol Decoding
Introduce how to decode the input signal using those common protocols.

8 Reference Waveform
Introduce how to compare the input waveform using the reference waveform.

9 Pass/Fail Test
Introduce how to monitor the input signal using the Pass/Fail test.

10 Waveform Recording
Introduce how to analyze the input signal using waveform recording.

11 Display Control
Introduce how to control the display of the oscilloscope.

12 Store and Recall
Introduce how to store and recall the measurement result and the setting of the

oscilloscope.

13 System Function Setting
Introduce how to perform the remote interface, system-related and print operations.

14 Remote Control
Introduce how to perform remote control of the oscilloscope.

15 Troubleshooting
Introduce how to deal with common failures of the oscilloscope.

16 Specifications
List the technical resolutions and general specifications of the oscilloscope.

17 Appendix
Provide the options and accessories information.

Format Conventions in this Manual:
The function keys at the front panel: denoted by the format of "Text Box + Button Name (Bold)", for example, **SAVE**.
The menu softkeys: denoted by the format of "Character Shading + Menu Word (Bold)", for example, **Storage**.
Operation steps: denoted by the arrow "→". For example, **SAVE** → **Storage**.

Content Conventions in this Manual:
This manual takes DS6104 for example and the descriptions here have contained all the functions and performances of other models. The models of DS6000 series include:

Model	DS6104	DS6102	DS6064	DS6062
Analog bandwidth	1 GHz	1 GHz	600 MHz	600 MHz
Channels	4	2	4	2
Max. real-time sample rate	5 GSa/s			
Standard memory depth	140 Mpts			
Waveform refresh rate	Up to 180,000 wfms/s			

Contents

1 Quick Start

This chapter introduces the preparations when using the oscilloscope for the first time, the front panel, rear panel and user interface of the oscilloscope as well as the using method of the built-in help system.

The contents of this chapter:

- General Inspection
- Appearance and Dimensions
- To Prepare the Oscilloscope for Use
- Front Panel Overview
- Rear Panel Overview
- Front Panel Function Overview
- User Interface
- To Use the Security Lock
- To Use the Desk Mount Instrument Arm
- To Use the Rack Mount Kit
- To Use the Built-in Help System

General Inspection

1. **Inspect the shipping container for damage.**
    Keep the damaged shipping container or cushioning material until the contents
    of the shipment have been checked for completeness and the instrument has
    passed both electrical and mechanical tests.

    The consigner or carrier shall be liable for the damage to instrument resulting
    from shipment. **RIGOL** would not be responsible for free maintenance/rework
    or replacement of the unit.

2. **Inspect the instrument.**
    In case of any damage, or defect, or failure, notify your **RIGOL** sales
    representative.

3. **Check the Accessories**
    Please check the accessories according to the packing lists. If the accessories
    are incomplete or damaged, please contact your **RIGOL** sales representative.

Appearance and Dimensions

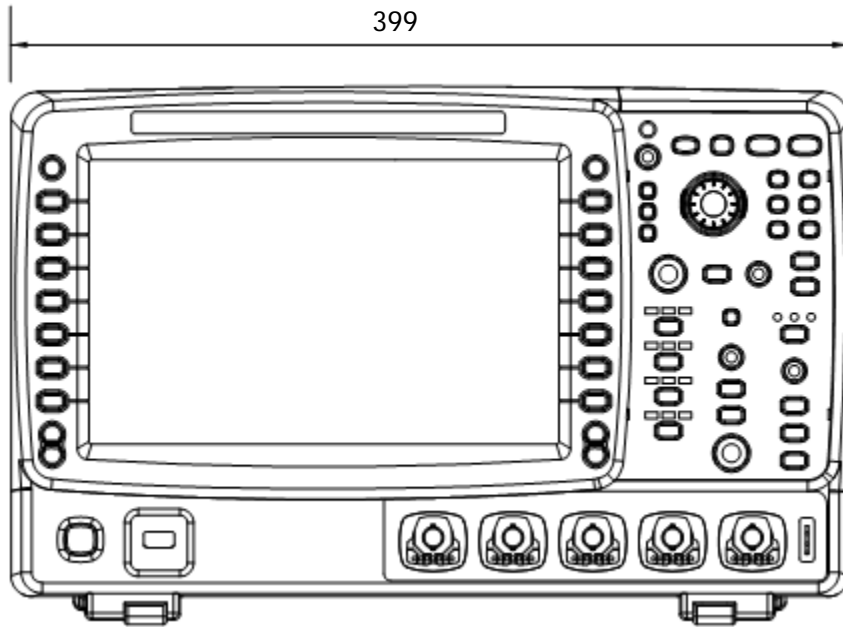Figure 1-1 Front View   Unit: mm
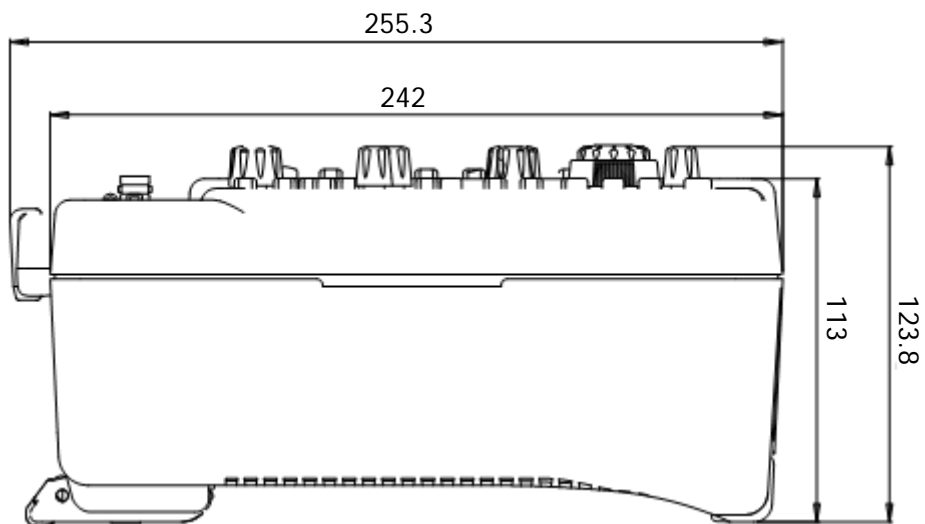
Figure 1-2 Side View   Unit: mm

To Prepare the Oscilloscope for Use

To Remove the Cover

Before using the oscilloscope, remove the front panel cover by grasping the transverse grab on each side and pull them in the arrow directions as shown in the figure below.

Figure 1-3 To Remove the Cover

To Adjust the Supporting Legs

Adjust the supporting legs properly to use them as stands to tilt the oscilloscope upwards for stable placement of the oscilloscope and   better operation and observation of the display. Unfold or fold the supporting legs in the arrow directions as shown in the figures below.
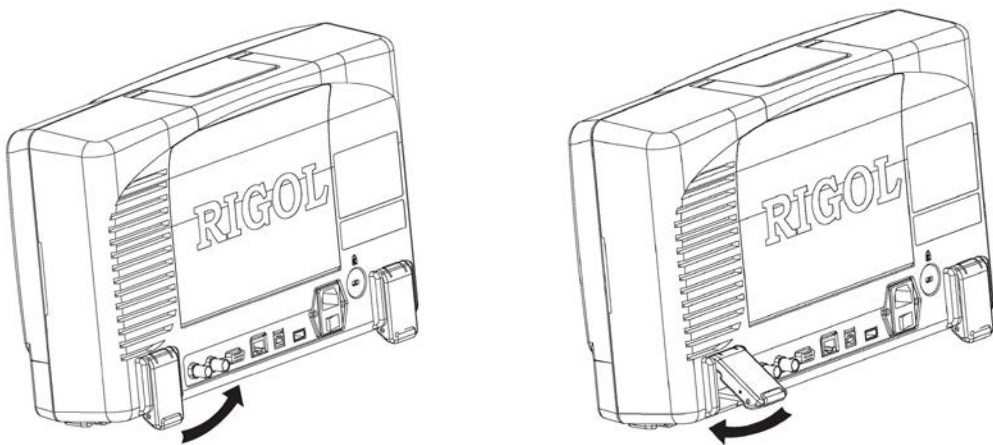
Figure 1-4 To Adjust the Supporting Legs

To Connect to AC Power Supply

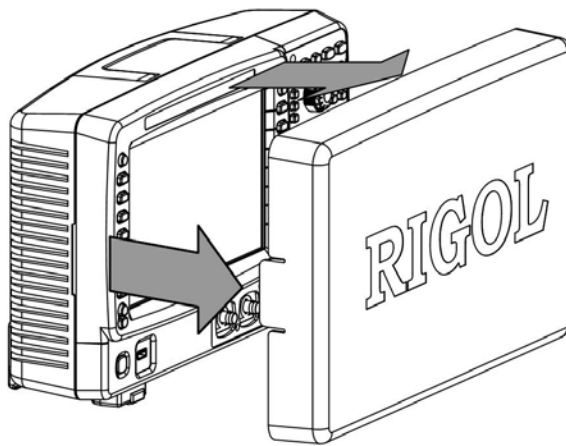This oscilloscope accepts two kinds of AC power supply: 100-127V, 45-440Hz and 100-240V, 45-65Hz. Please use the power cord supplied with the accessories to connect the oscilloscope to the power source as shown in the figure below. After the power switch under the power plug is turned on, the oscilloscope is energized and the power key ⏻ at the lower-left corner of the front panel is in breath state.
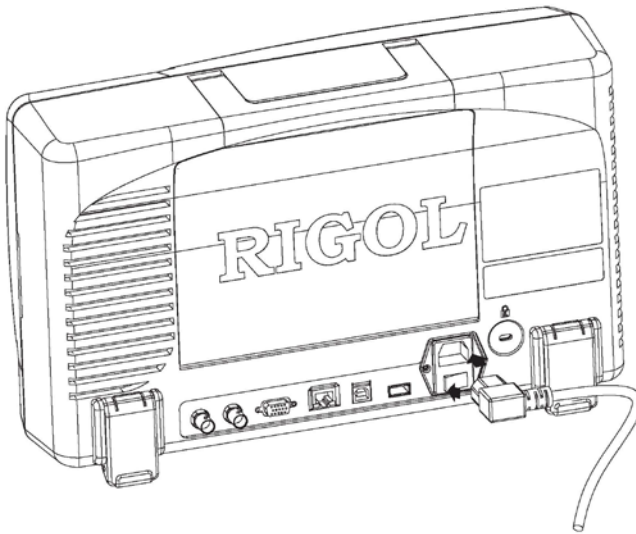
Figure 1-5 To Connect to AC Power Supply

To Use the Battery

This oscilloscope also provides rechargeable battery on your option. Please install the battery as shown in the figure below. When leaving the factory, the battery holds certain electric quantity. When use the battery alone, a battery icon will be displayed at the lower-right corner of the screen and the remaining electric quantity is indicated (no display when the AC power and battery are used at the same time). When the electric quantity of the battery is too low, "Low Battery!" will be displayed.

If recharge is needed, please install the battery correctly and then follow the instructions in **"To Connect to AC Power Supply"** to connect the oscilloscope to AC power supply to recharge the battery. It needs about 15 hours for the battery to finish recharging.
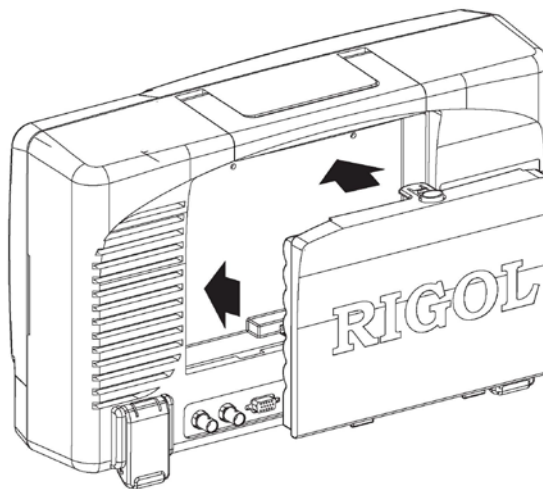
Figure 1-6 To Install the Battery

Power-on Inspection

When the oscilloscope is energized, press the power key ⏻ at the lower-left corner of the front panel to turn on the oscilloscope. During the start-up process, the oscilloscope performs a series of self-tests and you can hear the sound of relay switching. After the self-test is finished, the welcome screen is displayed and you can view the Option Type, Option name, Option Edition and Left time of the option currently installed in the "Installed Options" pop-up dialog box on the screen. When the instrument is shipped, a trial version of the option is provided and the left time is about 2000 minutes. Press **UTIL** → **System** → **SelfCalInfo** to view the self-test results.

To Connect the Probe

RIGOL provides passive and active probes for DS6000 series oscilloscopes. For detailed technical information of the probes, please refer to corresponding Probe User's Guide. The following are the probes recommended for this oscilloscope.

Model	Description
RP5600	600 MHz, passive probe, standard, auto detection
RP3500	500 MHz, passive probe, optional, auto detection
RP6150	1.5 GHz, passive probe, DS610X standard, auto detection
RP7150	1.5 GHz, active probe, optional, auto detection

Connect the Probe:
1. Connect the BNC terminal of the probe to a channel BNC connector of the oscilloscope at the front panel.
2. Connect the probe tip to the circuit point under test and connect the ground alligator clip of the probe to the circuit ground terminal.
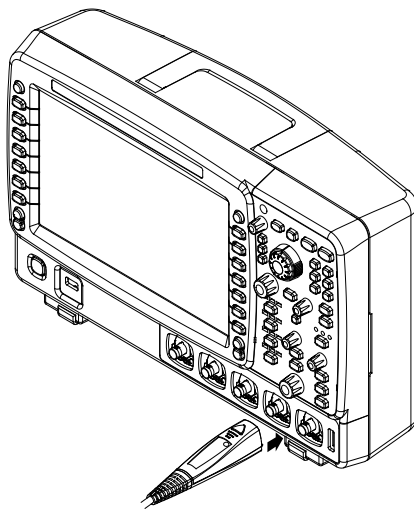
Figure 1-7 To Connect the Probe

Function Inspection

1. Press **SAVE** → **Factory** to restore the oscilloscope to its default configuration.
2. Connect the ground alligator clip of the probe to the "Ground Terminal" under the probe compensation signal output terminal
3. Use the probe to connect the input terminal of CH1 of the oscilloscope and the "Compensation Signal Output Terminal" of the probe.
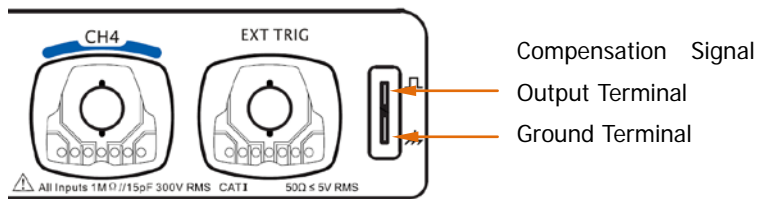
Figure 1-8 To Use the Compensation Signal

4. Press **Auto**.
5. Observe the waveform on the display. In normal condition, the display should be a square waveform as shown in the figure below:
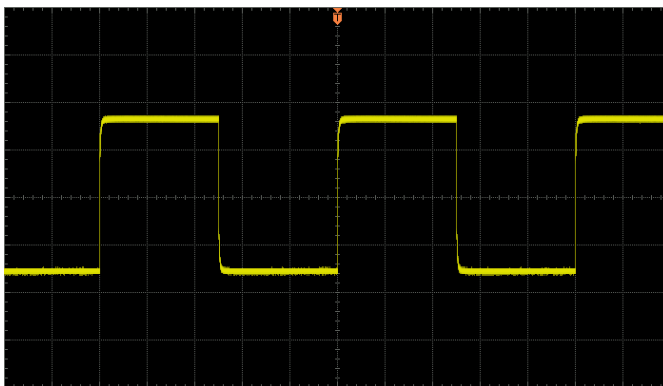
Figure 1-9 Square Waveform

6. Use the same method to test the other channels. If the square waveforms actually shown do not match that in the figure above, please perform **"Probe Compensation"** in the next section.

WARNING

To avoid electric shock during the use of probe, please make sure 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.

Tip

The signal output from the probe compensation connector can only be used for probe compensation adjustment and can not be used for calibration.

Probe Compensation

When the probes are used for the first time, you should compensate the probes to match the input channels of the oscilloscope. Non-compensated or poorly compensated probes may cause measurement inaccuracy and error. The probe compensation procedures are as follows.

1. Perform steps 1, 2, 3 and 4 of **"Function Inspection"** in the previous section.
2. Check the displayed waveforms and compare them with the following figures.

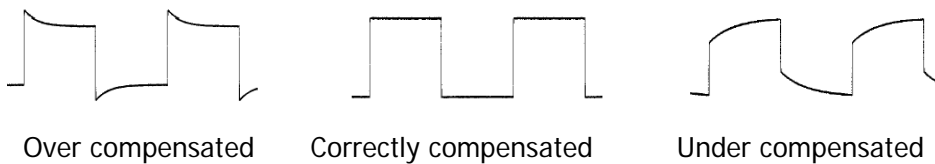Over compensated   Correctly compensated   Under compensated

Figure 1-10 Probe Compensation

3. Use a nonmetallic driver to adjust the low-frequency compensation adjustment hole on the probe until the displayed waveform is as the "Correctly compensated" in the figure above.

Front Panel Overview
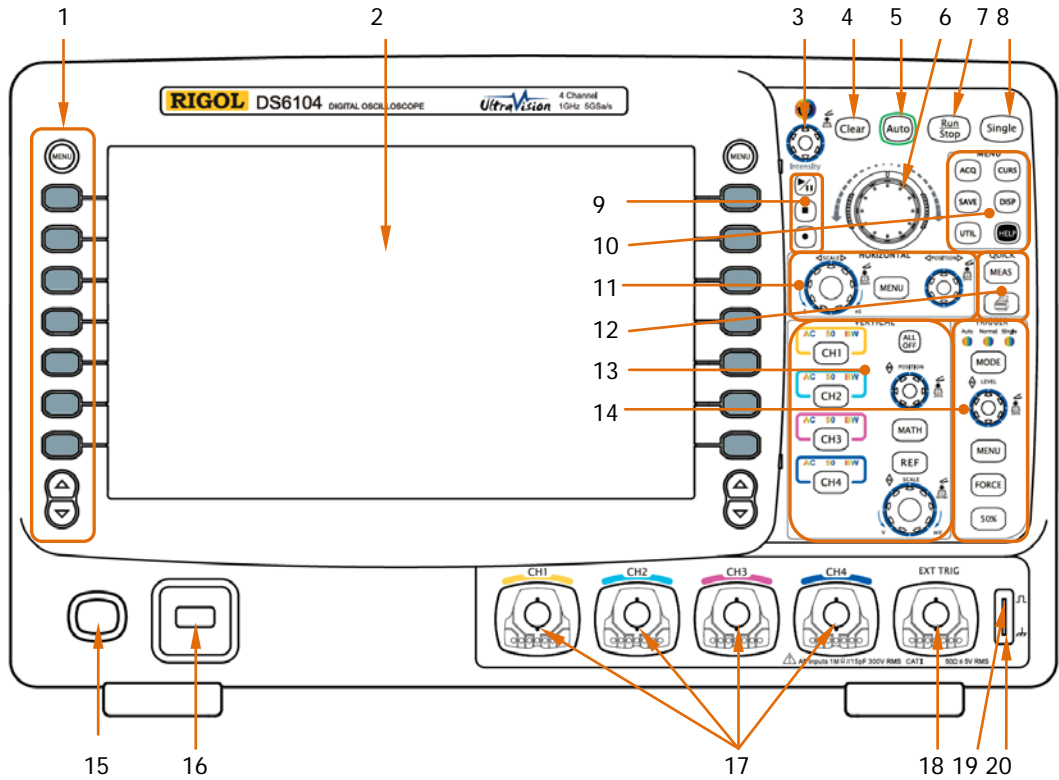
Figure 1-11 Front Panel Overview

Table 1-1 Front Panel Description

No.	Description	No.	Description
1	Menu	11	HORIZONTAL Area
2	LCD	12	Measurement Setting and Quick Print
3	Knob	13	VERTICAL Area
4	Clear	14	TRIGGER Area
5	Auto	15	Power Key
6	Navigation Knob	16	USB HOST Port
7	Run/Stop	17	Analog Channel Input Terminal
8	Single	18	EXT TRIG Input Terminal
9	Record/Playback	19	Probe Compensation Signal Output Terminal
10	Function Menu	20	Probe Compensation Ground Terminal

Rear Panel Overview

Figure 1-12 Rear Panel Overview

1. **Trig Out/Calibration**

   This connector can output various signals (press **UTIL** → **Aux Output** to select the desired output type.):

   1) **TrigOut:** after this type is selected, the oscilloscope output a signal that can reflect the current capture rate of the oscilloscope at each trigger.
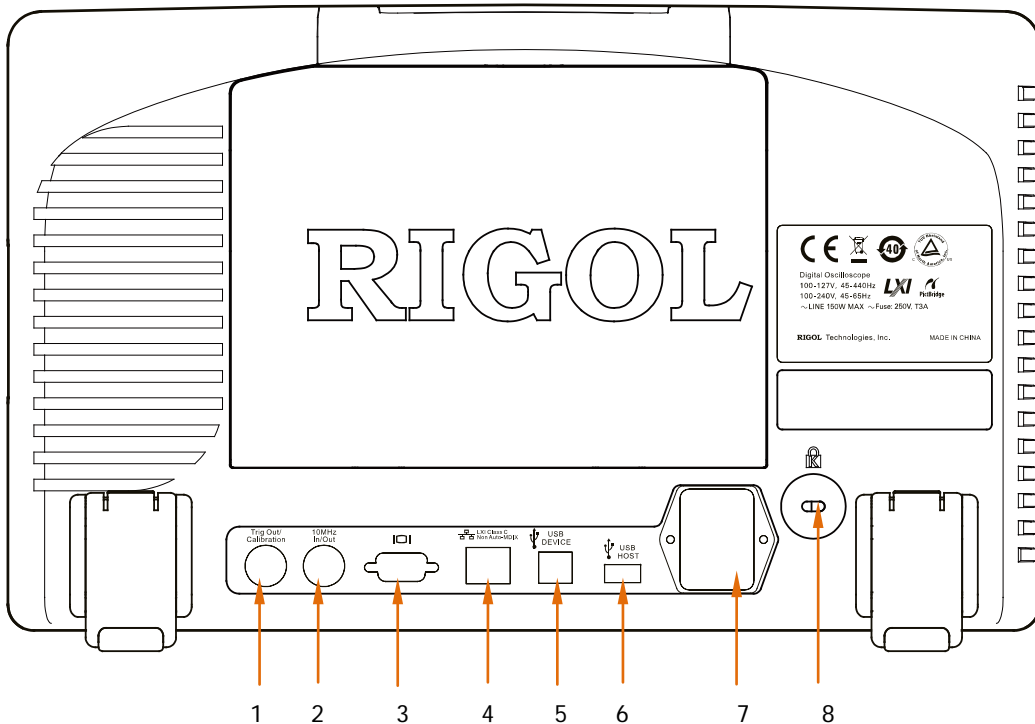
   2) **Fast:** after this type is selected, the oscilloscope can output a fast edge signal which can be used in the self-calibration of the oscilloscope.

   3) **GND:** after this type is selected, the oscilloscope can output a ground level.

   4) **PassFail:** after this type is selected, the oscilloscope will output a pulse signal when failed waveforms are detected. Connect this signal to other control systems to conveniently view the test results.

2. **Reference Clock**

   Provide more precise sample clock signal for the oscilloscope and it can also

synchronize two or more oscilloscope clocks.

3. **Video Output**

   Through this interface, the oscilloscope can be connected to external monitors such as projector to get clearer waveform display. Note that at this point, the display of the oscilloscope is still valid.

4. **LAN**

   Through this interface, the oscilloscope can be connected to the network for remote control. As the oscilloscope conforms to the LXI-C instrument standard, a test system can be built quickly.

5. **USB DEVICE**

   Through this interface, the oscilloscope can be connected to PictBridge printer to print waveform data or be connected to PC to control the oscilloscope through PC software.

6. **USB HOST**

   Through this interface, the oscilloscope can be connected to normal printers to print waveform data or be connected to a USB storage device to store waveform files.

   GPIB interface communication can be realized by using the USB-GPIB interface converter (optional) provided by **RIGOL**.

   Note that the front panel also provides this interface.
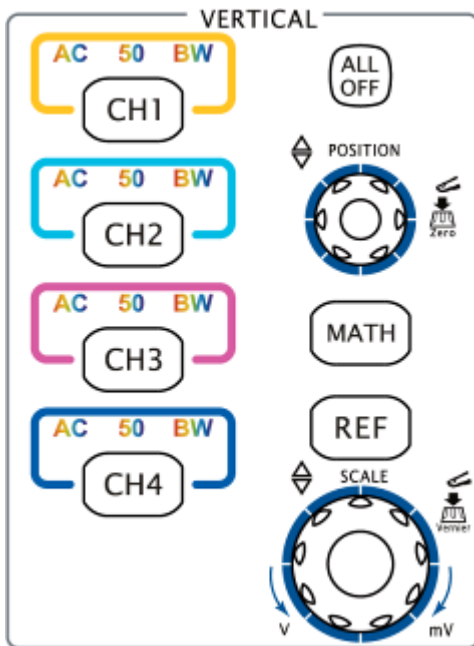
7. **AC Input/Switch**

   When using AC power supply, please insert the power cord plug into the socket vertically and use the switch under the socket to power the oscilloscope on or off.

8. **Lock Hole**

   Use the security lock (please buy it yourself) to lock the oscilloscope to fixed location.

Front Panel Function Overview

VERTICAL

CH1, **CH2**, **CH3**, **CH4**: analog input channels. The four channels are marked by different colors which are also used to mark both the waveforms on the screen and the channel input connectors. Press any key to turn on the corresponding channel menu and press again to turn off the channel.
AC: illuminated when AC channel coupling mode is enabled.
50: illuminated when the channel input impedance is 50Ω.
BW: illuminated when the bandwidth limit is turned on.

ALL OFF: press this key to turn off all the channels that have been turned on.

MATH: press this key to turn on the math operation and decoding function menus under which to perform add, subtract, multiply, divide, FFT, logic and advanced operations as well as parallel, RS232, SPI, I2C, CAN and FlexRay decodings.

REF: press this key to turn on the reference waveform function to compare the waveform actually tested with the reference waveform to decide circuit failures.

VERTICAL ⊚ **POSITION**: modify the vertical position of the current channel waveform. Turn clockwise to increase the position and turn counterclockwise to decrease. During the modification, the waveform would move up and down and the position message (e.g. POS:-136.0mV) at the lower-left corner of the screen would change accordingly. Press down this knob to quickly reset the vertical position to zero.

VERTICAL ⊙ <u>SCALE</u>: modify the vertical scale of the current channel. Turn clockwise to decrease the scale and turn counterclockwise to increase. During the modification, the amplitude of the waveform would enlarge or reduce and the scale information (e.g. `⎓ 100mV`) at the lower side of the screen would change accordingly. Press down this knob to quickly switch the vertical scale adjustment modes between "Coarse" and "Fine".

HORIZONTAL
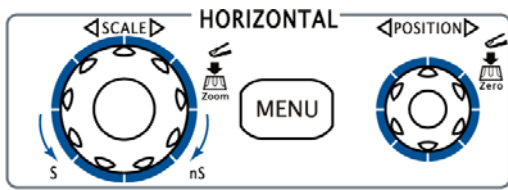
MENU: press this key to turn on the horizontal control menu under which to turn on or off the delayed sweep function, switch between different time base modes, switch between "Coarse" and "Fine" adjustment of scale as well as modify the horizontal reference setting.

HORIZONTAL ⚙ **SCALE**: modify the horizontal time base. Turn clockwise to reduce the time base and turn counterclockwise to increase the time base. During the modification, waveforms of all the channels will be displayed in expanded or compressed mode and the time base message (e.g. H 500.0us) at the upper side of the screen would change accordingly. Press down this knob to quickly switch to delayed sweep state.

HORIZONTAL ⚙ **POSITION**: modify the trigger position. The trigger point moves left or right relative to the center of the screen when you turn the knob. During the modification, waveforms of all the channels would move right or left and the trigger position message (e.g. D 780.000000ns) at the upper-right corner of the screen would change accordingly. Press down this knob to quickly restore the trigger position (or the delayed sweep position).

TRIGGER

MODE: press this key to switch the trigger mode to **Auto**, **Normal** or **Single** and the corresponding state backlight of the current trigger mode would be illuminated.

TRIGGER ⊙ **LEVEL**: modify the trigger level. Turn clockwise to increase the level and turn counterclockwise to reduce the level. During the modification, the trigger level line moves up and down and the value in the trigger level message box (e.g. Trig Level:880mV) at the lower-left corner of the screen changes accordingly. Press down the knob to quickly reset the trigger level to zero point.

MENU: press this key to turn on the trigger operation menu. This oscilloscope provides various trigger types.

FORCE: in **Normal** and **Single** trigger modes, press this key to generate a trigger signal forcefully.

50%: press this key to set the trigger level to the vertical midpoint of the trigger signal amplitude.

Run/Stop

Press this key to set the state of the oscilloscope to "Run" or "Stop". In "Run" state, the key is illuminated in yellow and red in "Stop" state.

Single

Press this key to set the trigger mode to "Single". In single trigger mode, the oscilloscope generates a trigger when the trigger conditions are met and then stops.

Auto

Press this key to turn on 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 and provide quick parameter measurement (please refer to **Quick Measurement after Auto**).

Note: auto setting requires that the frequency of the signal under test should be no lower than 50 Hz, the duty cycle be greater than 1% and the amplitude be at least 20 mVpp. If the parameters exceed these limits, "Auto failed!" would be displayed after pressing this key and the quick parameter measurement menu might not be displayed.
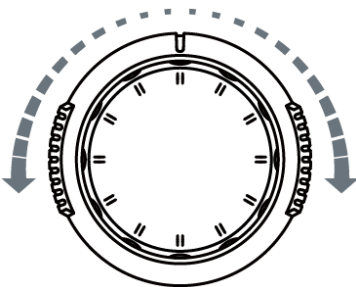
Knob

Adjust waveform brightness:

In non-menu-operation mode (menu is hidden), turn this knob to adjust the brightness of waveform display. The adjustable range of the brightness is from 0% to 100%. Turn clockwise to increase the brightness and counterclockwise to reduce. Press down this knob to reset the brightness to 50%.

You can also press **DISP** → **WaveIntensity** and use the knob to adjust the waveform brightness.

Multifunctional Knob (the backlight goes on during operation):

In menu operation, press some menu softkey and turn the knob to select the submenus under this menu and then press down the knob to select the currently selected submenu. It can also be used to modify parameters and filename input.

Navigation Knob

This knob provides quick Adjust/Locate function for numerical parameters with relatively large settable range. Turn clockwise (counter-clockwise) to increase (reduce) the value. Note that the inner knob is used for fine adjustment and the outer knob for coarse adjustment. The change rate of the value depends on the rotation amplitude of the outer knob.

For example, this knob can be used to quickly locate the waveform frame ("Current Frame" menu) to be played back in the waveform playback function. Similar menus include trigger holdoff, pulse width setting, slope time etc.

Menu
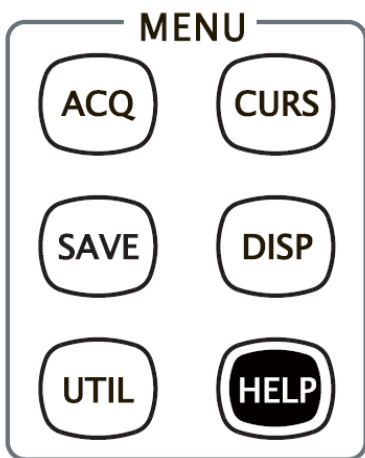
ACQ: press this key to enter sample setting menu to set the acquisition mode, the sampling mode, the memory depth and the antialiasing function of the oscilloscope.

CURS: press this key to enter cursor measurement menu. The oscilloscope provides three cursor modes: manual, track and auto.

SAVE: press this key to enter file store and recall interface. The storable file types include traces, waveforms, setups, picture and CSV. Support internal and external storage as well as disk management.

DISP: press this key to enter display setting menu to set the display type, persistence time, wave intensity, grid type, grid brightness and menu display time of the waveform.

UTIL: press this key to enter the system function setting menu to set the system-related functions or parameters, such as I/O setting, sound and language. Support some advanced functions such as pass/fail test, waveform record and print setting.

HELP: press this key to turn on the help interface. For detailed information, please refer to the introduction in **"To Use the Built-in Help System"**.

Record

Play/Pause: in stop or pause state, press this key to play back the waveform and press again to pause the play. The backlight is illuminated in yellow.

Stop: press this key to stop the waveform in record or playback mode. The backlight is illuminated in orange.

Record: press this key to start recording the waveform. The backlight is illuminated in red.

MEAS

Press this key to enter the measurement menu which supports measurement parameter setting, all measure, statistic analysis and counter. Press **MENU** at the left of the screen to quickly turn on the selecting menu of the 24 measurement parameters and easily realize "one-key" measurement of common parameters.

Clear

Press this key to clear all the waveforms on the screen. If the oscilloscope is in "Run" state, new waveforms will still be displayed.

Print

Press this key to execute print function or save the screen in the USB storage device. If the oscilloscope is currently connected to a printer (PictBridge or normal) and the printer is in idle state, press this key to execute print function. If no printer but an USB storage device is currently connected, press this key to save the screen to the USB storage device in ".bmp" format (if the current storage type is picture, the screen will be saved in the USB storage device in the specified picture format). When printer and USB storage device are connected at the same time, the printer enjoys higher priority.

User Interface

DS6000 oscilloscope provides 10.1 inches, WVGA (800*480) 160,000 color TFT LCD. What is worth mentioning is that the 14-grid ultra-wide screen makes you view "longer" waveform.
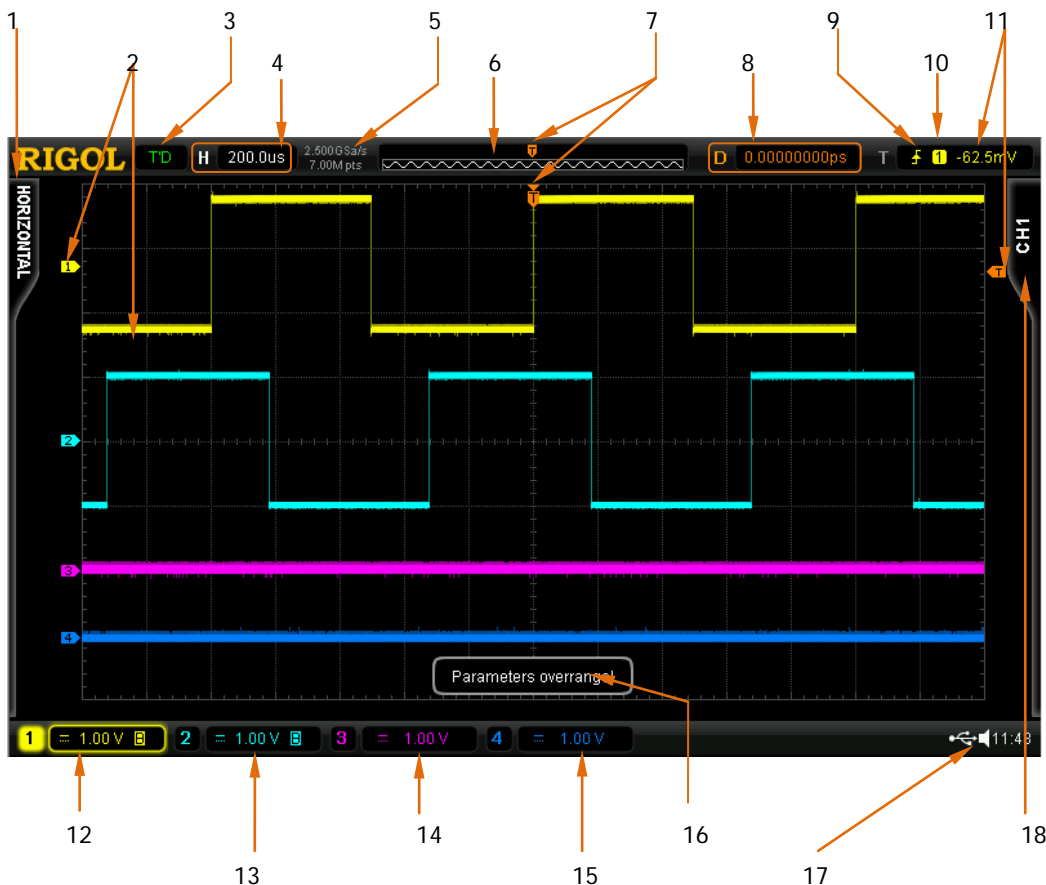

Figure 1-13 User Interface

1. Auto Measurement Menu

Provide 12 horizontal (HORIZONTAL) and 12 vertical (VERTICAL) measurement parameters. Press the softkey at the left of the screen to activate the corresponding measurement item.

2. Channel Label/Waveform

Different channels are marked by different colors and the color of the waveform complies with the color of the channel.

3. Status

Available states include RUN, STOP, T'D (triggered), WAIT and AUTO.

4. Horizontal Time Base

- Represent the time per grid on the horizontal axis on the screen.
- Use **HORIZONTAL** ⚙ **SCALE** to modify this parameter. The range available is from 500 ps to 1.000 ks.

5. Sample Rate/Memory Depth

- Display the current sample rate and memory depth of the oscilloscope.
- Use **HORIZONTAL** ⚙ **SCALE** to modify this parameter.

6. Waveform Memory

Provide the schematic diagram of the memory position of the waveform currently on the screen.

waveform on the screen
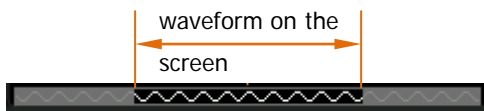
7. Trigger Position

Display the trigger positions of the waveform in the waveform memory and on the screen.

8. Trigger Position

Use **HORIZONTAL** ⚙ **POSITION** to modify this parameter. Press down the knob to automatically set the parameter to zero.

9. Trigger Type

Display the currently selected trigger type and trigger condition setting. Different labels are displayed when different trigger types are selected. For example, ⚡ represents triggering on the rising edge in "Edge" trigger.

10. Trigger Source

Display the trigger source currently selected (CH1-CH4, EXT, EXT/5 or AC Line). Different labels are displayed when different trigger sources are selected and the color of the trigger parameter area will change accordingly. For example, **1** denotes that CH1 is selected as the trigger source.

11. Trigger Level

● **T** at the right of the screen is the trigger level label and the trigger level value is displayed at the upper-right corner of the screen.

● When Using **TRIGGER** ⊙ **LEVEL** to modify the trigger level, the trigger level value will change with the up and down of **T**.

Note: in slope trigger, there are two trigger level labels (**T1** and **T2**). For detailed information, please refer to the introduction in **"Vertical Window"** on page 5-12.

12. CH1 Vertical Scale

● Display the voltage value per grid of CH1 waveform vertically.

● Use **VIRTICAL** ⊙ **SCALE** to modify this parameter.

● The following labels will be provided according to the current channel setting: "**Channel Coupling**" (such as ▧ in AC coupling), "**Bandwidth Limit**" (such as **B** when bandwidth limit is enabled) and "**Input Impedance**" (such as **Ω** when the input impedance is **50 Ω**).

13. CH2 Vertical Scale

● Display the voltage value per grid of CH2 waveform vertically.

● Use **VIRTICAL** ⊙ **SCALE** to modify this parameter.

● The following labels will be provided according to the current channel setting: "**Channel Coupling**" (such as ▧ in AC coupling), "**Bandwidth Limit**" (such as **B** when bandwidth limit is enabled) and "**Input Impedance**" (such as **Ω** when the input impedance is **50 Ω**).

14. CH3 Vertical Scale

● Display the voltage value per grid of CH3 waveform vertically.

● Use **VIRTICAL** ⊙ **SCALE** to modify this parameter.

● The following labels will be provided according to the current channel setting: "**Channel Coupling**" (such as ▧ in AC coupling), "**Bandwidth Limit**" (such as **B** when bandwidth limit is enabled) and

"**Input Impedance**" (such as ⊙ when the input impedance is **50 Ω**).

15. CH4 Vertical Scale
- Display the voltage value per grid of CH4 waveform vertically.
- Use **VIRTICAL** ⊙ **SCALE** to modify this parameter.
- The following labels will be provided according to the current channel setting: "**Channel Coupling**" (such as ▦ in AC coupling), "**Bandwidth Limit**" (such as 🅱 when bandwidth limit is enabled) and "**Input Impedance**" (such as ⊙ when the input impedance is **50 Ω**).

16. Message Box
Display prompt messages.

17. Notification Area
Display system time, sound icon, battery icon and USB storage device icon.
- System Time: displayed in "hh:mm (hour:minute)" format. When printing or storing the waveform, the output file will contain this time message. Press **UTIL** → **System** → **System Time** to set through the following format:
  yyyy -mm-dd   hh-mm-ss (year-month-date   hour-minute-second)

- Sound Icon: when sound is turned on, this area displays ◩. Press **UTIL**

  → **Sound** to turn the sound on or off.
- Battery Icon: when the battery is used for power supply, this area displays ▥.
- USB storage device: when the oscilloscope detects a USB storage device, this area displays ⟜.

18. Operation MENU

Press any softkey to activate the corresponding menu. The following symbols might be displayed in the menu:

↻	Denote that ↻ on the front panel can be used to select parameter items. The backlight of ↻ turns on when parameter selection is valid.
↻	Denote that ↻ can be used to modify parameter values. The backlight of ↻ turns on when parameter input is valid.
⊙	Denote that you can use the Navigation Konb to quickly adjust/locate parameters.
↻	Denote that ↻ can be used to adjust parameters and then press ↻ to select the parameter.
◀	Denote that the current menu has several options.
▼	Denote that the current menu has a lower level menu.
↵	Press this key to return to the previous menu.

Note: the following direction keys might appear in the grid at the lower-left corner of the menu bar:

▽	Denote that you can turn on the next page menu.
△	Denote that you can turn back the previous page menu.

To Use the Security Lock

If needed, you can use the security lock (please buy it yourself) to lock the oscilloscope to a fixed location. The method is as follows, align the lock with the lock hole and plug it into the lock hole vertically, turn the key clockwise to lock the oscilloscope and then pull the key out.
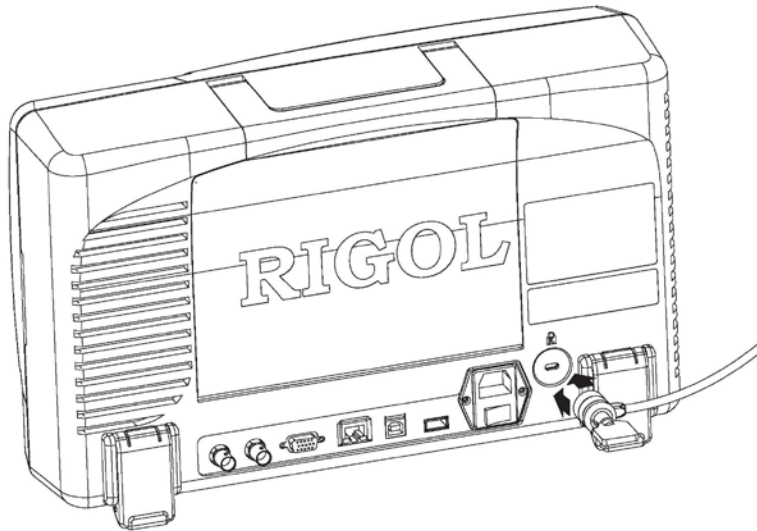
Figure 1-14 To Use the Security Lock

Note: do not insert other articles into the security lock hole to avoid damaging the instrument.

To Use the Desk Mount Instrument Arm

Using an arm, the oscilloscope could be mounted on the workbench to save your operation space. The height and angle of the instrument could be adjusted freely to acquire supreme comfort and efficiency and to convenient your measurement and observation. If needed, please buy and install the corresponding option.

Figure 1-15 Working Sketch of the Instrument with the Instrument Arm

To Use the Rack Mount Kit

This oscilloscope can be mounted in a standard 19-inch rack cabinet that conforms to the Electric Industrial Association (EIA) standard. If needed, please buy and install the corresponding option.

Kit Parts List

No.	Name	Qty	Part No.	Description
1-1	Front Plate	1	RM-DS-6-01	
1-2	Support Board	1	RM-DS-6-02	
1-3	Left Plate	1	RM-DS-6-03	
1-4	Right Plate	1	RM-DS-6-04	
1-5	Back Batten	2	RM-DS-6-05	
2-1	M4 Screw	24	RM-SCREW-01	M4 x 6 Phil-Slot Pan Head Machine Screw Nail
2-2	M6 Screw	4	RM-SCREW-02	M6 x 16 Phil-Slot Pan Head Machine Screw Nail
2-3	M6 Nut	4	RM-SCREW-03	M6 x 5 Square Machine Female Screw Contain Lock Blade
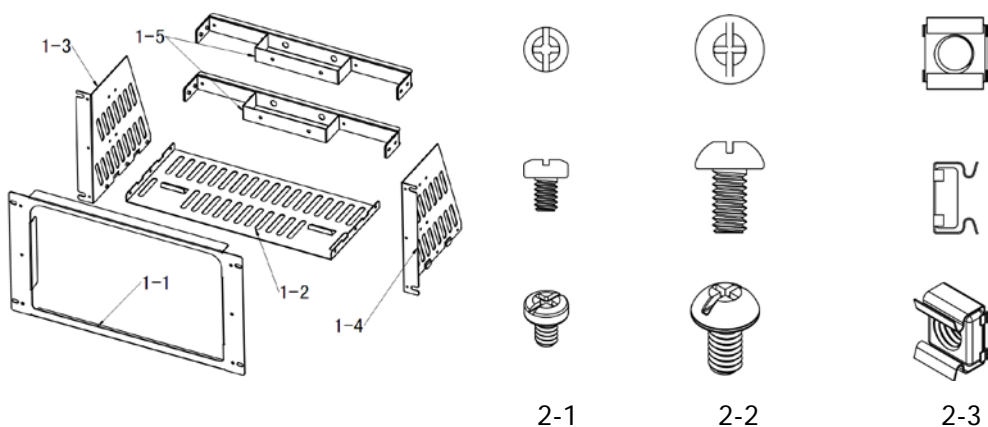
Installation Tool

PH2 Phillips Screwdriver (recommended).

Installation Space

The following requirements must be fulfilled by the machine cabinet in which the instrument is mounted.

- The machine cabinet must be a standard 19-inch one.
- At least 6U (266.7 mm) height should be provided by the machine cabinet.
- The depth inside the machine cabinet should not be less than 400 mm.

The dimension of the instrument after being installed is shown below (the unit is mm).
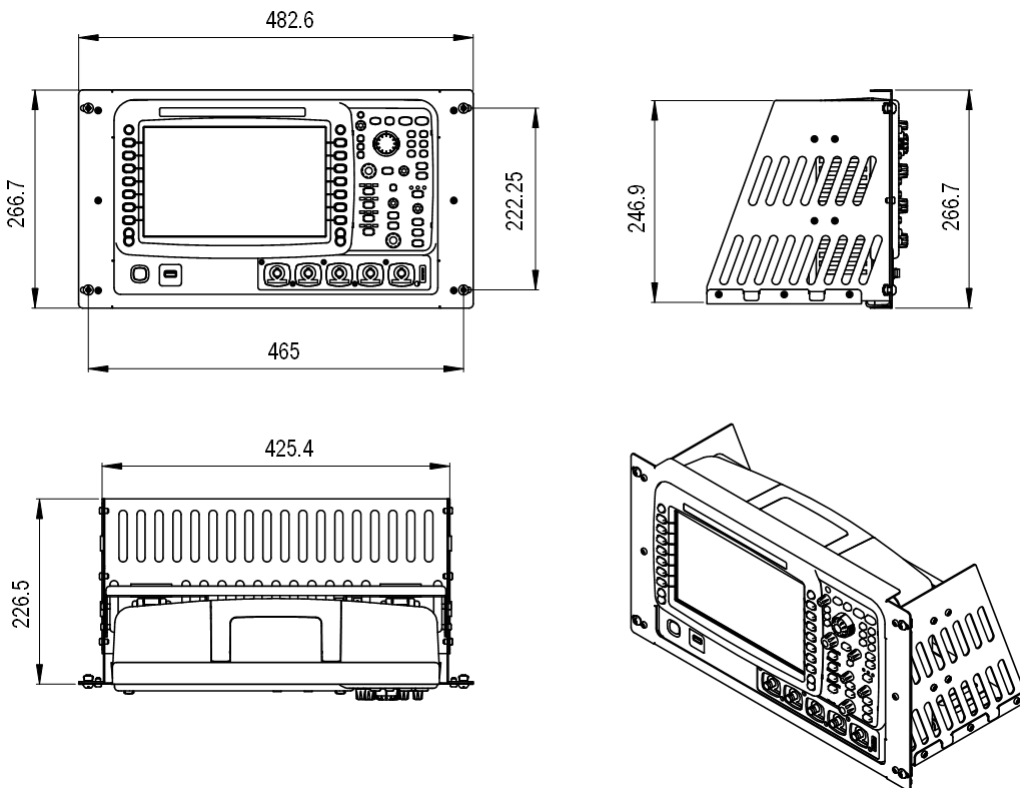
Installation Procedures

Only authorized operators can execute the installation operation. Improper installation might result in damage of the instrument or incorrect installation of the instrument on the rack.

1. Install the right and left plates: align the detents of the right and left plates with the openings on the support board and insert them into the support board respectively, then fix them with six M4 screws.
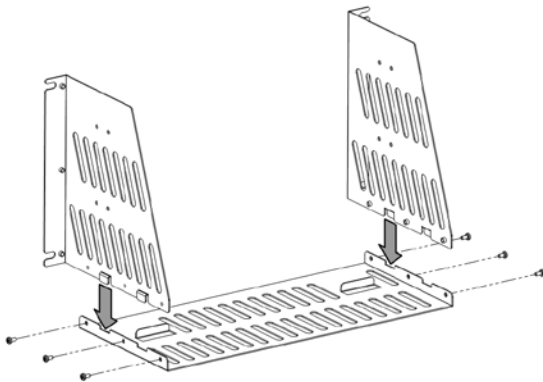
2. Place the Instrument: press the button above the battery on the instrument to remove the battery. Then align the instrument with the spacing on the support board and place it on the support board.
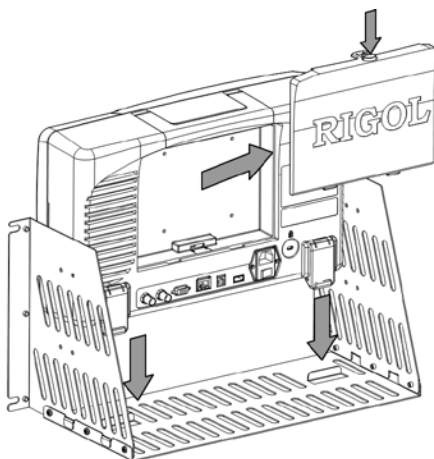
3. Fix the instrument: fix the instrument onto the support board with two back battens and twelve M4 screws.
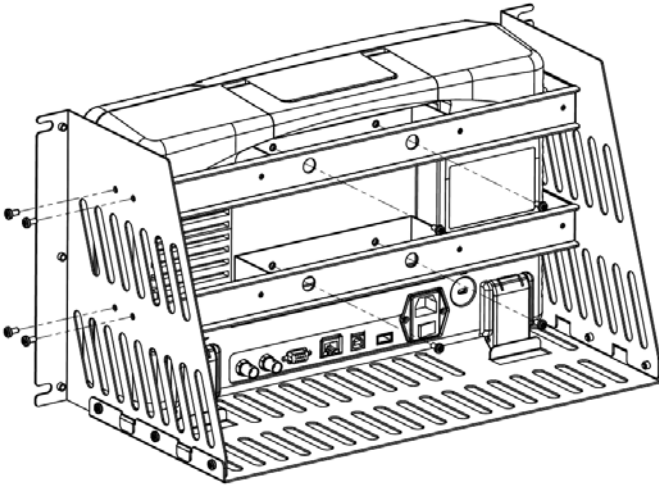
4. Install the front plate: align the screw holes on the right and left plates with the screw holes on the front plate of the rack and let the front panel of the oscilloscope protrude from the opening. Then fix them with six M4 screws.
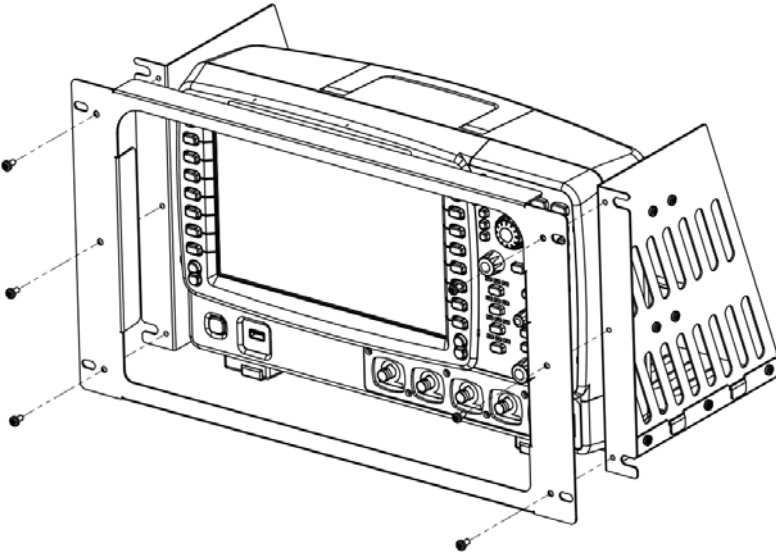
5. Load into the machine cabinet: mount the rack with the instrument fixed to it into a standard 19-inch machine cabinet with four M6 screws and four M6 square nuts.
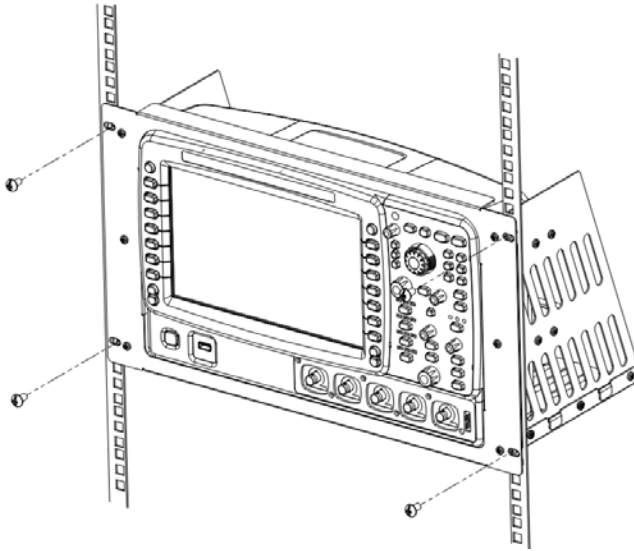
6. Post-installation notice: the rack occupies a height of 6U. The holes pointed out by the arrows are installation holes. Note that they should be aligned with during installation.
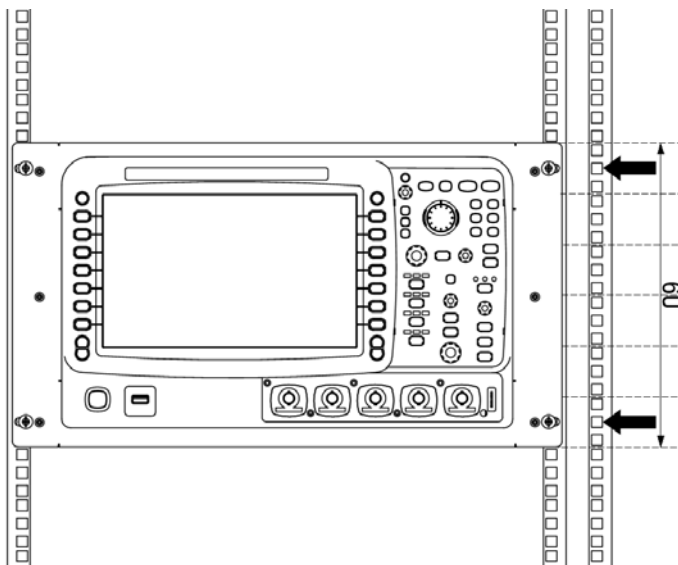
To Use the Built-in Help System

The help system of this oscilloscope provides instructions for all the function keys (including menu keys) at the front panel. Press **HELP** to open the help interface and press again to close the interface. The help interface mainly consists of two parts. The left is "Help Options" and you can use "Button" or "Index" to select. The right is "Help Display Area".
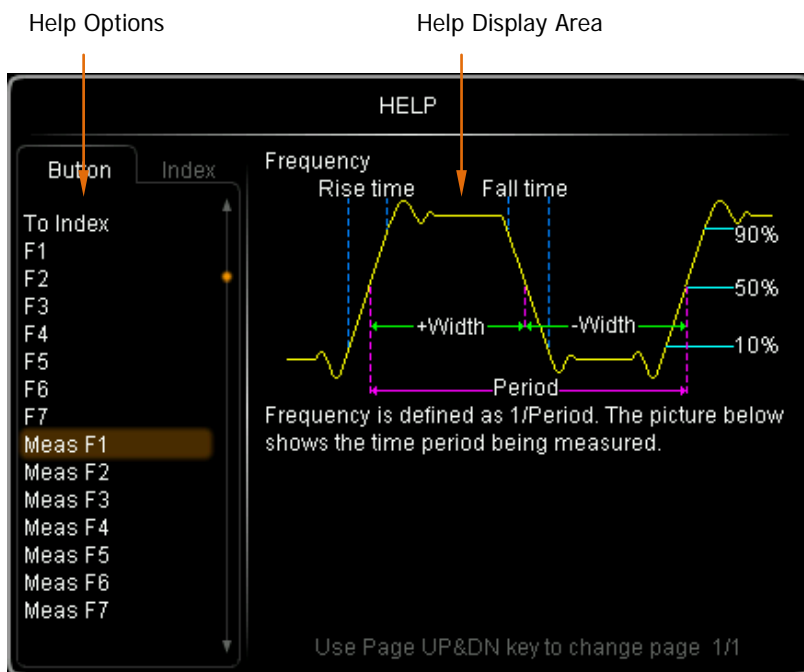
Figure 1-16 Help Information

Button:
Default mode. In this mode, you can press the buttons (except the power key ⊚, the knob ↻ and menu page up/down key ⌃/⌄) or rotate the Navigation knob ○ at the front panel directly to get the corresponding help information of that button in the "Help Display Area".
Use ↻ to select "To Index" and then press the knob to switch to **Index** mode.

Index:
In this mode, use ↻ to select the item that needs to get help (for example: "Band Width"). Press the knob to get the corresponding help information in the "Help Display Area".
Use ↻ to select "To Button" and then press the knob to switch to **Button** mode.

2 To Set the Vertical System

The contents of this chapter:

- To Enable the Channel
- Channel Coupling
- Bandwidth Limit
- Probe
- Input Impedance
- Waveform Invert
- Vertical Scale
- Vertical Expansion
- Amplitude Unit
- Channel Label
- Delay Calibration

To Enable the Channel

DS6000 provides four analog input channels (CH1-CH4) and each channel can be controlled independently.
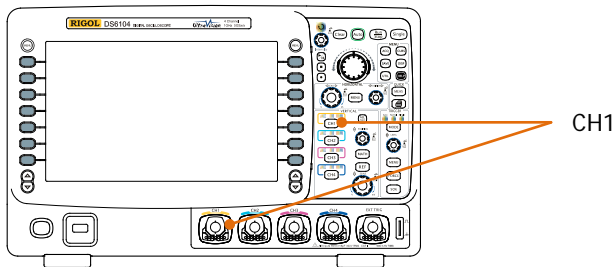
Connect a signal to the channel connector of any channel (for example, CH1) and then press **CH1** in the vertical control area (VERTICAL) at the front panel to enable CH1. At this point:

Panel:
The backlight of this key turns on. If the corresponding function in this menu has been turned on, the characters "AC", "50" or "BW" might also be illuminated. Note: the on/off state of the key light of "AC", "50" or "BW" is not limited by the on/off state of the channel.

Screen:
The channel setting menu is displayed at the right side of the screen and the channel label at the bottom of the screen (as shown in the figure below) is highlighted. The information displayed in the channel label is related to the current channel setting.

After the channel is turned on, modify the parameters such as the vertical scale, the horizontal time base and the trigger mode according to the input signal to make the waveform display easy to observe and measure.

Channel Coupling

Set the coupling mode to filter out the signals that is not needed. For example, the signal under test is a square waveform with DC offset.

- When the coupling mode is "DC": the DC and AC components of the signal under test can both pass the channel.
- When the coupling mode is "AC": the DC components of the signal under test are blocked.
- When the coupling mode is "GND": the DC and AC components of the signal under test are both blocked.

Press CH1 → **Coupling** and use ↻ to select the desired coupling mode (the default is DC). The current coupling mode is displayed in the channel label at the bottom of the screen. When "AC" is selected, the character "AC" above the CH1 channel key at the front panel will be illuminated. You can also press **Coupling** continuously to switch the coupling mode.

Bandwidth Limit

Set the bandwidth to reduce the display noise. For example, the signal under test is a pulse signal with high frequency oscillation.

- When the bandwidth limit is turned off, the high frequency components of the signal under test can pass the channel.
- Turn on the bandwidth limit and limit it to 20 MHz or 250 MHz, the high frequency components that exceed 20 MHz or 250 MHz are attenuated.

Press **CH1** → **BW Limit** and use ↻ to turn on/off the bandwidth limit (the default is off). When the bandwidth limit (20 MHz or 250 MHz) is turned on, the character "**B**" will be displayed in the channel label at the bottom of the screen. You can also press **BW Limit** continuously to switch between on and off of the bandwidth limit.

Probe

This oscilloscope supports normal passive probe and active differential probe and can automatically identify the type of the probe currently connected and the probe ratio. Press **CH1** → **Probe** to turn on the probe operation menu.

1. **ProbeType**
   Read the type of the probe currently connected as "Nor-Probe" or "DiffProbe". Note that when using a 50 Ω "DiffProbe", the **Input Impedance** of the channel is set to "50Ω" automatically.
   - Normal Probe: such as RP5600, RP3500 and RP6150 of **RIGOL**.
   - Differential Probe: such as RP7150 of **RIGOL**.

2. **Ratio**
   The oscilloscope can identify the probe ratio automatically. If the oscilloscope can not identify, press this softkey to select the corresponding probe ratio. The probe ratio can be the values listed in the table on the next page.

Table 2-1 Probe Attenuation Coefficient

Menu	Attenuation coefficient
0.01X	1:100
0.02X	1:50
0.05X	1:20
0.1X	1:10
0.2X	1:5
0.5X	1:2
1X	1:1
2X	2:1
5X	5:1
10X	10:1
20 X	20:1
50 X	50:1
100 X	100:1
200 X	200:1
500 X	500:1
1000 X	1000:1

When "DiffProbe" is inserted, the oscilloscope recognizes it automatically and the following menus are added.

3. Front-End

RP7150 active probe provides "Single-end" and "Difference" probe heads. Press this softkey to select the desired probe head.

4. Probe-Cal

Connect the differential probe to the channel input terminal (such as CH1) of the oscilloscope correctly, then connect the fast edge signal output from the **[Trig Out/Calibration]** connector at the rear panel of the instrument to the probe and CH2 respectively. At this point, press this softkey and the oscilloscope starts to calibrate the probe.

Note: if the probe is connected to CH1, CH3 or CH4, the fast edge signal should be connected to CH2; if the probe is connected to CH2, the fast edge signal should be connected to CH1.

5. Probe Info

Press this softkey to view various information about the probe, such as the manufacturer, model, serial number and the date of last calibration.

6. Bias Voltage

The oscilloscope provides bias voltage for differential probe. This bias voltage is used to adjust the signal under test that exceeds the input dynamic range of the probe amplifier to an appropriate range to ensure the completeness of the signal under test. Press this softkey and use the knob to adjust the bias voltage. The range is from -12V to +12V.

Input Impedance

To reduce the circuit load caused by the interaction of the oscilloscope and the circuit under test, this oscilloscope provides two input impedance modes: 1 MΩ (default) and 50 Ω.

- 1 MΩ: at this point, the input impedance of the oscilloscope is very high and the current flows into the oscilloscope from the circuit under test can be ignored.
- 50 Ω: match the oscilloscope with devices whose output impedance is 50 Ω.

Press **CH1** → **Input** to set the input impedance of the oscilloscope. When "50 Ω" is selected, the character "**Ω**" is displayed in the channel label at the bottom of the screen.

Note: after the probe is automatically identified, the input impedance will be automatically identified and you do not need to set it manually.

Waveform Invert

When waveform invert is turned on, the waveform display rotates 180 degree relative to the ground potential. When the waveform invert is turned off, the waveform display is normal. Press **CH1** → **Invert** to turn waveform invert on or off.

Vertical Scale

The vertical scale adjustment has two modes: "Coarse" and "Fine".

Press **CH1** → **Volts/Div** to select the desired mode. Turn the **VERTICAL** ⊚ **SCALE** to adjust the vertical scale. Turn clockwise to reduce the scale and turn counterclockwise to increase the scale.

The scale information (e.g. ⎓ 100mV) in the channel label at the bottom of the screen will change accordingly during the adjustment. The adjustable range of the vertical scale is related to the current probe ratio. By default, the probe attenuation ratio is 1X and the adjustable range of the vertical scale is from 2 mV/div to 5 V/div.

● 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...5 V/div.
● Fine adjustment: further adjust the vertical scale within a relatively smaller range to improve vertical resolution. If the amplitude of the input waveform is a little bit greater than the full scale and the amplitude would be a little bit lower if the next scale is used, fine adjustment can be used to improve the amplitude of waveform display to view the details of the signal.

Note: you can also press **VERTICAL** ⊚ **SCALE** to quickly switch between corse and fine adjustments.

Vertical Expansion

When using **VERTICAL** ⚙ **SCALE** to change the vertical scale of the analog channel, you can choose to expand or compress the vertical signal around the center of the screen or around the ground point of the signal.
Press **UTIL** → **System** → **VerticalExp** to select "Center" or "Ground" and the default is "Ground".
● Center: when the vertical scale is changed, the waveform will expand or compress around the center of the screen.
● Ground: when the vertical scale is changed, the waveform ground level will remain at the same point on the screen and the waveform will expand or compress around this point.

Amplitude Unit

Select the amplitude display unit for the current channel. The available units are W, A, V and U. when the unit is changed, the unit in the channel label will change accordingly.

Press **CH1** → **Unit** to select the desired unit and the default is V.

Channel Label

You can modify the labels used to mark the analog channels (CH1 to CH4) at the left side of the screen. The label is the number of the channel (e.g. <mark>1</mark>) by default and the length of the modified label can not exceed 4 characters.

Press **CH1** →**Label** to enter the label modification interface. As shown in the figure below:

Name Input Area   Keyboard   Upper-lower Case Switch
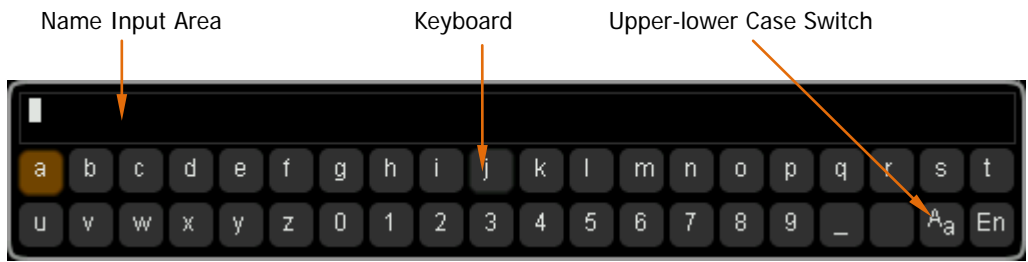
For example, change "<mark>1</mark>" to "<mark>Chn1</mark>".

Press **Keyboard** to select the "Keyboard" area. Select "Aa" using ↻ and press down ↻ to switch it to "^{a}A".   Select "C" using ↻ and press down ↻ to input the character. Use the same method to input "hn1".

To modify or delete the inputed character, press **Name** to select the "Name Input Area" and use ↻ to select the character to be modified or deleted. Reenter the desired character or press **Delete** to delete the selected character.

After finishing the input, press **OK** to finish the modification and the channel label will change to "<mark>Chn1</mark>".

Delay Calibration

When using the oscilloscope for actual measurement, the transmission delay of the probe cable may bring relatively larger error (zero point offset). For DS6000, users can set a delay time to calibrate the zero point offset of the corresponding channel. Zero point offset is defined as the offset of the crossing of the waveform and trigger level line relative to the trigger position as shown in the figure below.

Zero Point Offset
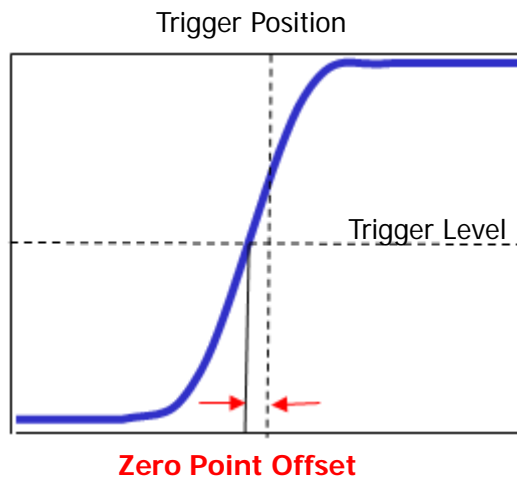
Press CH1 → **Delay-Cal** and use the knob to set the desired delay time. The range is from -100 ns to 100 ns.

Note: this parameter is related to the horizontal time base currently set. For example, when the horizontal time base is 5 μs, the step of delay time setting is 100 ns; when the horizontal time base is 1 μs, the step of delay time setting is 20 ns; when the horizontal time base is 500 ns, the step of delay time setting is 10 ns.

3 To Set the Horizontal System

The contents of this chapter:

- ■ Delayed Sweep
- ■ Time Base Mode
- ■ Horizontal Scale
- ■ Horizontal Reference

Delayed Sweep

Delayed sweep can be used to enlarge a length of waveform horizontally to view the details of the image.
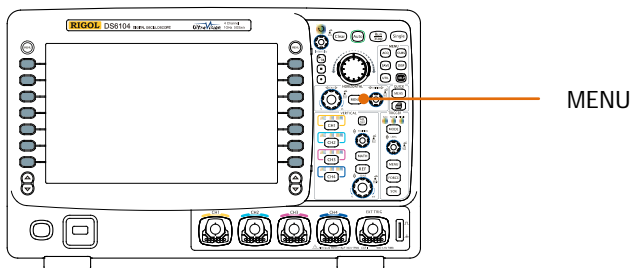
MENU

Press **MENU** in the horizontal control area (HORIZONTAL) and then press **Delayed** to turn the delayed sweep on or off. Note: to enable delayed sweep, the current time base mode must be "Y-T" and the "Pass/Fail test" must be disabled.

In delayed sweep mode, the screen is divided into two display areas as shown in the figure below.

The waveform before
enlargement

Main time
base

Delaye
Sweep
Time
Base

The waveform after enlargement
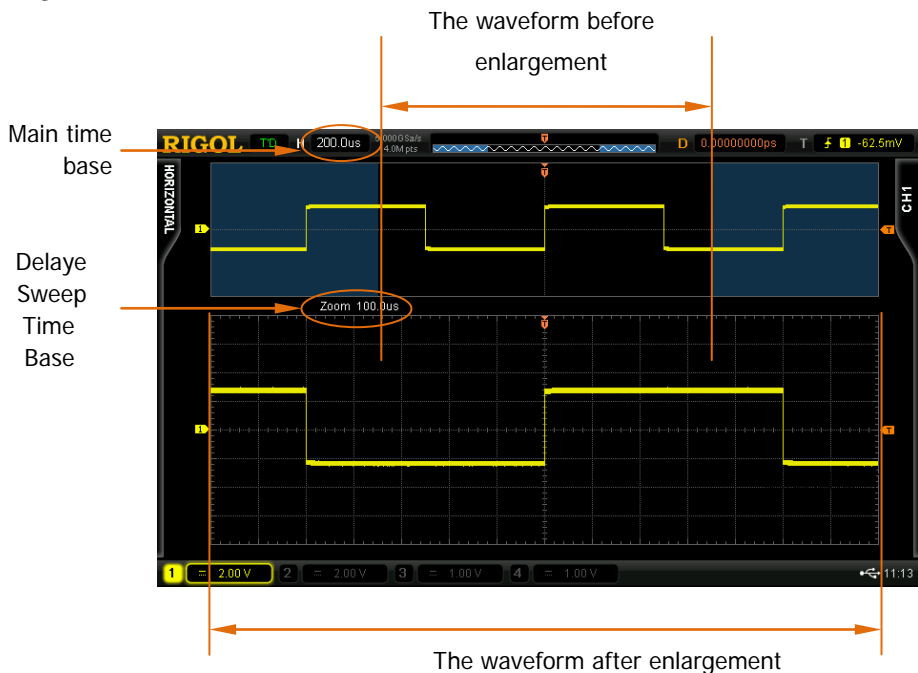
The waveform before enlargement:

The waveform in the area that has not been covered by the subtransparent blue in the upper part of the screen is the waveform before enlargement. You can turn **HORIZONTAL** ⊙ **POSITION** to move the area left and right or turn **HORIZONTAL** ⊙ **SCALE** to enlarge or reduce this area.

The waveform after enlargement:

The waveform in the lower part of the screen is the horizontally expanded waveform. Note: the delayed time base has increased its resolution relative to the main time base (as shown in the figure above). The delayed time base should be less than or equal to the main time base.

Tip

When the channel is currently CH1-CH4, you can enter the delayed sweep not only through MENU operation in the horizontal area but also by directly pressing **HORIZONTAL** ⊙ **SCALE** (delayed sweep shortcut key) to switch to the delayed sweep mode.

Time Base Mode

Press **MENU** in the horizontal control area at the front panel and then press **Time Base** to select the time base mode of the oscilloscope and the default is Y-T mode.

Y-T Mode

This mode is the main time base mode and is applicable to CH1-CH4 input channels. In this mode, the Y axis represents voltage and the X axis represents time. Note: only when this mode is enabled can "**Delayed Sweep**" be turned on.

X-Y Mode

In this mode, the oscilloscope automatically turns on all the four channels (CH1-CH4) and the screen is divided into two coordinate areas, where, X1 and Y1 track the voltages of CH2 and CH1 and X2 and Y2 track the voltages of CH4 and CH3.
The phase deviation of two signals with the same frequency can be easily measured via Lissajous method. The figure below shows the measurement schematic diagram of the phase deviation.
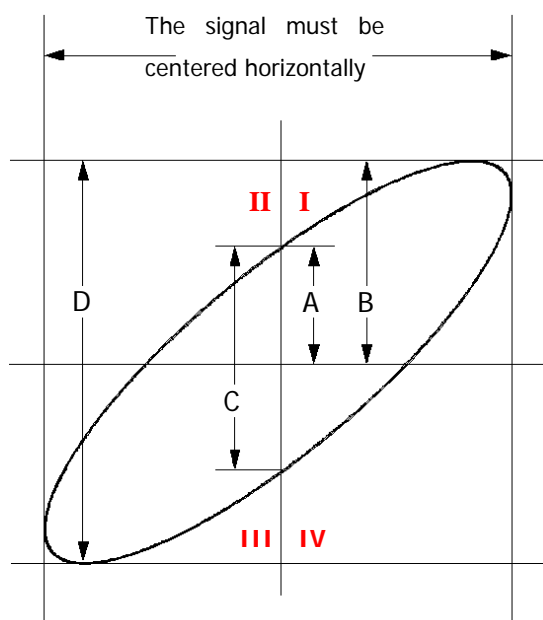
According to **sinθ=A/B** or **C/D** (where, θ is the phase deviation angle between the two channels and the definitions of A, B, C and D are as shown in the figure above), the phase deviation angle is obtained, that is:

$$\theta = \pm \text{arcsin (A/B) or} \pm \text{arcsin (C/D)}$$

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

X-Y function can be used to measure the phase deviation occurred when the test signal passes through a circuit network. Connect the oscilloscope to the circuit to monitor the input and output signals of the circuit.

Application example:

Measure the phase deviation of the input signals of two channels.

Method 1: Use Lissajous method

1. Connect a sine signal to CH1 and then connect a sine signal with the same frequency and amplitude but a 90° phase deviation to CH2.
2. Press **Auto**, turn on the X-Y mode and rotate **HORIZONTAL** ⊙ **SCALE** to adjust the sample rate properly to get a better Lissajous figure for better observation and measurement.
3. Adjust **VERTICAL** ⊙ **POSITION** of CH1 and CH2 to display the signals at the center of the coordinate region and adjust **VERTICAL** ⊙ **SCALE** of CH1 and CH2 to make the signals easy to observe. At this point, the circle as shown in the figure below should be got.
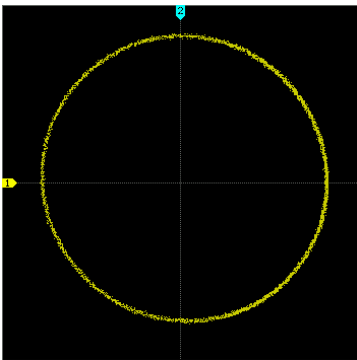
4. From the figure above, the distances from the points of intersection of the X/Y axis and the circle to the origin of the coordinates are approximately equal. Thus, the phase deviation angle **θ=±arcsin1=90°**.

Note:

- In Y-T mode, the oscilloscope could use arbitrary sample rate (within the guaranteed range) to capture waveform. The maximum sample rate of Y-T mode is 2.5 GSa/s. Generally, reducing the sample rate properly could get a Lissajous figure with relatively better display effect.
- When X-Y mode is turned on, "**Delayed**" will be turned off automatically.
- The following functions are not available in X-Y mode:
  Auto measure, cursor measure, math operation, delayed sweep, vector display type, **HORIZONTAL** ⚙ **POSITION**, trigger control, memory depth, acquisition mode, Pass/Fail test and waveform recording.

Method 2: Use the shortcut measurement function

Please refer to "Phase A→B↯" and "Phase A→B↳" measurement functions of "Delay and Phase" on page 6-17.

Roll Mode

In this mode, the waveform scrolls from the right to the left to update the display and the waveform horizontal position and trigger control are not available. The range of horizontal scale adjustment is from 200.0 ms to 1.000 ks.

Note: when Roll mode is turned on, "**Delayed Sweep**", "**Protocol Decoding**", "**Pass/Fail Test**", "**Measurement Range**", "**Waveform Recording**", "**To Set the Persistence Time**" and "**To Trigger the Oscilloscope**" are not available.

Slow Sweep

Another mode similar to the Roll mode. When the horizontal time base is set to 200 ms/div or slower, the instrument enters the "slow sweep" mode in which the instrument first acquires the data at the left of the trigger point and then waits for trigger event. After the trigger occurred, the instrument continues to finish the waveform at the right of the trigger point. When using the slow sweep mode to observe low frequency signal, it is recommended that the **Channel Coupling** is set to "DC".

Horizontal Scale

Being similar to "**Vertical Scale**", the horizontal scale adjustment has two modes: "Coarse" and "Fine".

Press MENU → **ScaleAdjust** in the horizontal control area (HORIZONTAL) at the front panel to select the desired mode. Turn **HORIZONTAL** ⊙ **SCALE** to adjust the horizontal scale. Turn clockwise to reduce the horizontal scale and turn counterclockwise to increase the scale.

The scale information (e.g. **H** 500.0us) at the upper left corner of the screen will change accordingly during the adjustment. The range of horizontal scale adjustment is from 500 ps to 1.000 ks.

● Coarse (take counterclockwise as an example): set the horizontal scale in 1-2-5 step namely 500 ps, 1 ns, 2 ns, 5 ns...1.000 ks.
● Fine: further adjust within smaller range.

Horizontal Reference

Horizontal reference is the reference position according to which the screen waveform expands and compresses horizontally when adjusting **HORIZONTAL** ⊙ **SCALE**. In Y-T mode, press **MENU** → **HorRef** in the horizontal control area (HORIZONTAL) to select the desired reference mode and the default is "Center".

1. **The Center of the Screen**
    When changing the horizontal time base, the waveform expands or compresses horizontally around the center of the screen.
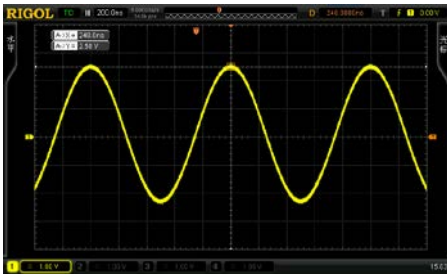
2. **Trigger Position**
    When changing the horizontal time base, the waveform expands or compresses horizontally around the trigger point.
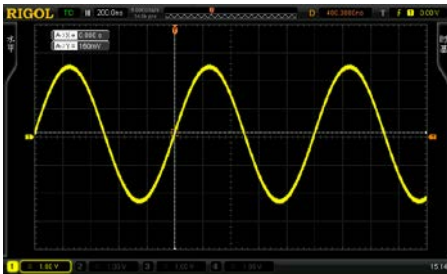
3. **User-defined**
    When changing the horizontal time base, the waveform expands or compresses horizontally around the user-defined reference position (after selecting "User", use the knob to set the reference position and the range is from -350 to 350). In the horizontal direction, the screen can display a maximum of 700 points with the leftmost as 350 and the rightmost as -350. For example: set the reference position to 150:
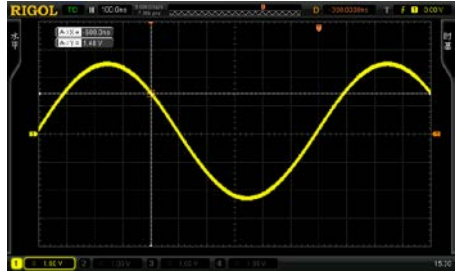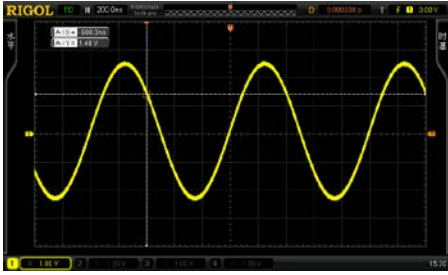
4 To Set the Sample System

The contents of this chapter:

- Acquisition Mode
- Sample Mode
- Sample Rate
- Memory Depth
- Antialiasing

Acquisition Mode

The acquisition mode is used to control how to generate waveform points from the sample points.

Press **ACQ** → **Acquisition** in the function menu at the front panel and use ↻ to select the desired acquisition mode (the default is normal), then press the knob to select this mode. You can press **Acquisition** continuously to switch the acquisition mode.

ACQ
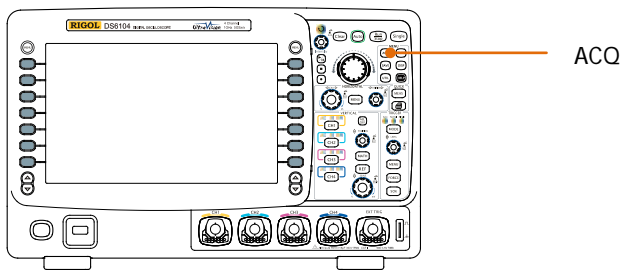
Normal

In this mode, the oscilloscope samples the signal at equal time interval to rebuild the waveform. For most of the waveforms, the best display effect is generated by using this 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 higher the number of averages is, the lower is the noise and the higher is the vertical resolution but the slower is the response of the displayed waveform to the waveform changes.

The available range of the number of averages is from 2 to 8192 and the default is 2. When "Average" mode is selected, press **Averages** and use ↻ to set the desired number of averages as the power function of 2.

The Waveform before Average:
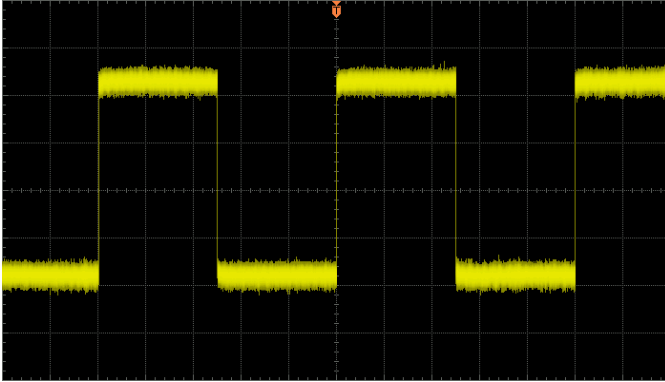
The Waveform after 256 Averages:
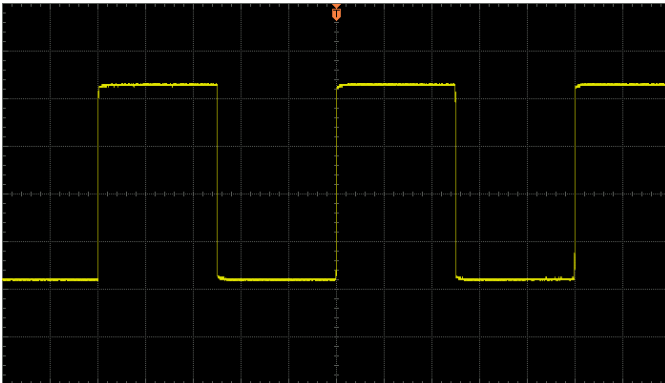
Peak Detect

In this mode, the oscilloscope acquires the maximum and minimum values of the sample interval signal to get the envelope or the narrow pulse of the signal that might be lost. Use this mode to avoid signal confusion at the expense of exaggerating the noise.

In this mode, the oscilloscope can display all the pulses whose pulse widths are at least as wide as the sample period.

High Resolution

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

Note: "Average" and "High Res" modes use different averaging methods. The former uses "Waveform Average" and the latter uses "Dot Average".

Sample Mode

Press ACQ → **Sampling** and press this softkey continuously to switch the sample mode (the default is real-time sample). Note: all the "**Acquisition Mode**" support real-time sample and equivalent sample.

Real-time Sample

In this mode, the oscilloscope samples and displays waveform within a trigger event. The maximum real-time sample rate of DS6000 is 5 GSa/s and the current sample rate is displayed under the **Sa Rate** menu.

Equivalent Sample

In this mode, the oscilloscope performs multiple trigger samples, samples at different positions of each trigger and uses the sample points from multiple triggers to rebuild the signal. The equivalent sample mode helps with detailed observation of the repeated periodic signals. The maximum equivalent sample rate of DS6000 is 100 GSa/s.

Tip

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

Sample Rate

The sample rate of this oscilloscope is up to 5 GSa/s. Note: the sample rate is displayed in the status bar at the upper side of the screen and in the **Sa Rate** menu and can be changed by adjusting the horizontal time base (s/div) through **HORIZONTAL** ⚙ **SCALE** or modifying the "**Mem Depth**".

The influence on the waveform when the sample rate is too low:

1. **Waveform Distortion:** when the sample rate is too low, some waveform details are lost and the waveform displayed is rather different from the actual signal.
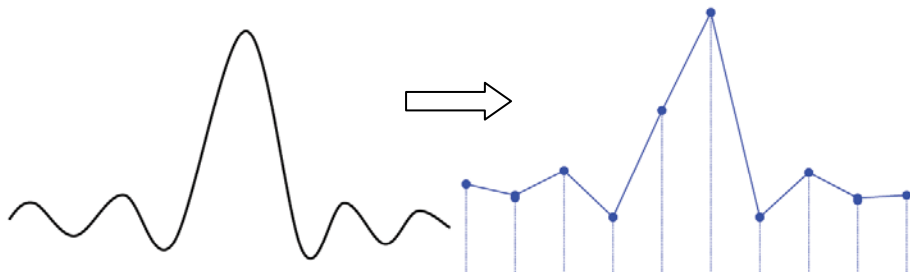
2. **Waveform Aliasing:** when the sample rate is lower than twice the actual signal frequency (Nyquist Frequency), the frequency of the waveform rebuilt by the sample data is lower than the actual signal frequency. The most common aliasing is the jitter on a fast edge.
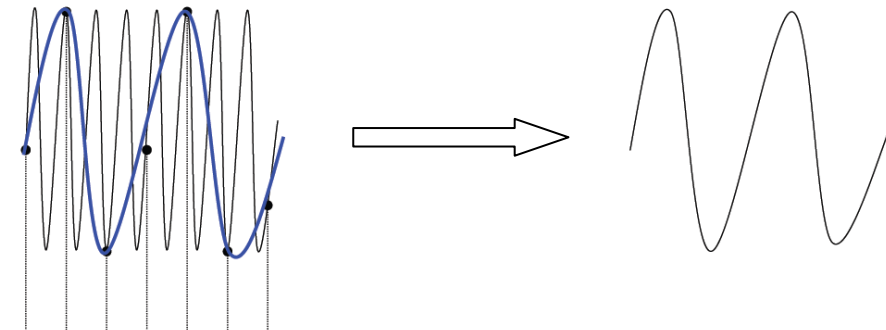
3. Waveform Leakage: when the sample rate is too low, the waveform rebuilt by the sample data does not reflect all the actual signal information.
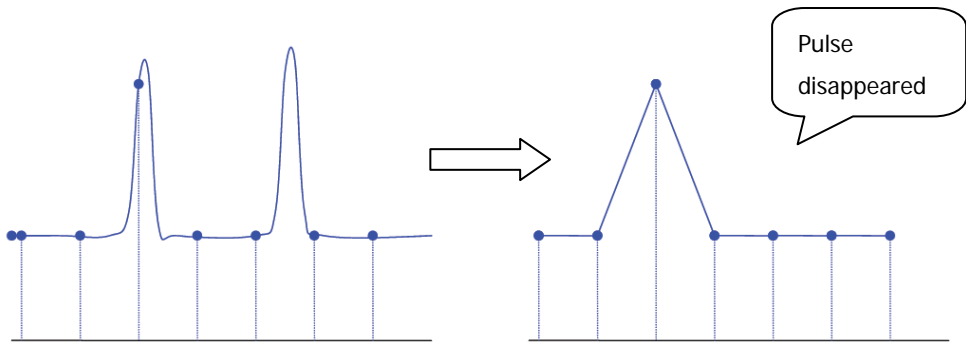
Memory Depth

Memory depth refers to the number of waveform points that the oscilloscope can store in a single trigger sample and it reflects the storage ability of the acquisition memory. DS6000 provides a memory depth up to 140 M points.

Trigger Point

T

|← Pre-sample →|← Delayed Sample →|

|← Memory Depth →|
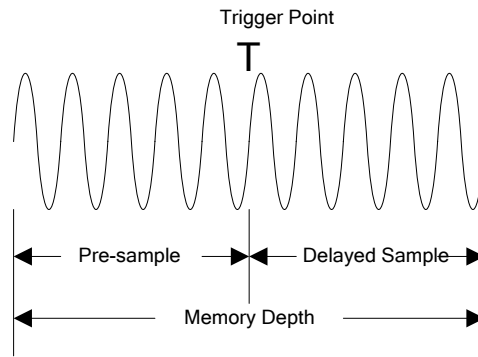
The relationship of memory depth, sample rate and waveform length fulfills the following equation:

Memory depth = sample rate (Sa/s) × waveform length (s/div × div)

Press **ACQ** → **Mem Depth**, use ↻ to select the desired memory depth (the default is auto) and then press the knob to select the option. You can also press **Mem Depth** continuously to switch the memory depth.

When single channel is turned on, the memory depths available include Auto, 14kPoints, 140kPoints, 1.4MPoints, 14MPoints and 140MPoints. In "Auto" mode, the oscilloscope selects the memory depth automatically according to the current sample rate.

When double channels are turned on (CH1 and CH2 or CH3 and CH4), the memory depths available include Auto, 7kPoints, 70kPoints, 700kPoints, 7MPoints and 70MPoints. In "Auto" mode, the oscilloscope selects the memory depth automatically according to the current sample rate.

Antialiasing

At slower sweep speeds, the sample rate is reduced and a proprietary display algorithm is used to minimize the likelihood of aliasing.

Press **ACQ** → **Anti-aliasing** to turn on or off the antialiasing function. By default, Antialiasing is disabled. The displayed waveforms will be more susceptible to aliasing when this function is disabled.

5 To Trigger the Oscilloscope

For trigger, you set certain trigger condition according to the requirement and when a waveform in the waveform stream meets this condition, the oscilloscope captures this waveform as well as the neighbouring part and displays them on the screen. When working, the digital oscilloscope continuously acquires waveform data even if a valid trigger does not occur. However, only when valid triggers occur, can the display become stable. The trigger circuit guarantees that each sweep or sample starts from the position of the trigger event in the input signal, namely each sweep and sample are synchronous and the waveforms acquired overlap to display stable waveform.

Trigger setting should be based on the features of the input signal, thus you need to have some knowledge of the signal under test to quickly capture the desired waveform. This oscilloscope provides abundant advanced trigger functions which help you to focus on the waveform details of interest.

The contents of this chapter:

- ■ Trigger Source
- ■ Trigger Mode
- ■ Trigger Coupling
- ■ Trigger Holdoff
- ■ Trigger Type
- ■ Trigger Ouput Connector

Trigger Source

Press **MENU** → **Source** in the trigger control area (TRIGGER) at the front panel to select the desired trigger source. Signals input from the analog channels CH1-CH4 and the **[EXT TRIG]** connector as well as the AC Line can all be used as the trigger source.

MENU
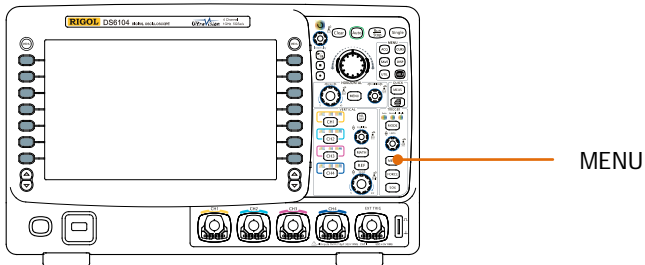
Analog channel input:
Signals input from analog channels CH1-CH4 can all be used as the trigger source. No matter whether the input from the channel selected is displayed, the channel can work normally.

External trigger input:
External trigger can be used to trigger on the fifth channel at the same time when all the four channels are acquiring data. The trigger signal (e.g. external clock, signal of the circuit to be tested) will be connected to **EXT** or **EXT/5** trigger source via the **[EXT TRIG]** connector. You can set the trigger conditions within the range of the trigger level from -0.8 V to +0.8 V.

AC Line:
The trigger signals are obtained from the AC power input of the oscilloscope. This kind of signals can be used to display the relationship between signal (such as illuminating device) and power (power supply device). For example, to stably trigger the waveform output from the transformer of a transformer substation, which is mainly used in related measurement of the power industry.

Trigger Mode

Trigger mode would influence the mode in which the oscilloscope searches for trigger event. The following is the schematic diagram of the acquisition memory from which you can see that the position of the trigger event is determined by the reference time point and the delay setting.

Acquisition memory
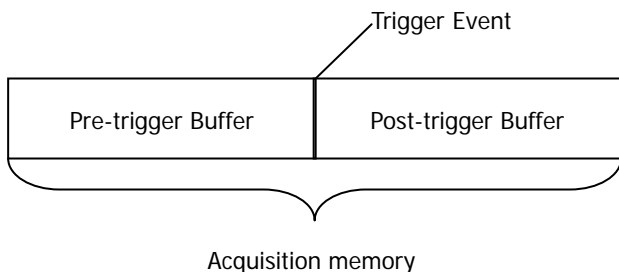
Pre-trigger/ Delayed trigger:
Acquire data before/after the trigger event. The trigger position is usually at the horizontal center of the screen. In full-screen display, you can view seven-grid pre-trigger and delayed trigger information. You can adjust the horizontal position of the waveform through **HORIZONTAL** ⚙ **POSITION** to view more pre-trigger information and delayed trigger information, through which to know the signal information before/after the trigger (e.g. capture the glitch generated by the circuit and analyze the pre-trigger data to find out the reasons for generating glitch).

Press **MODE** in the trigger control area (TRIGGER) at the front panel or through the **MENU** ➔ **Mode** menu to select the desired trigger mode. The corresponding status light of the mode currently selected turns on.
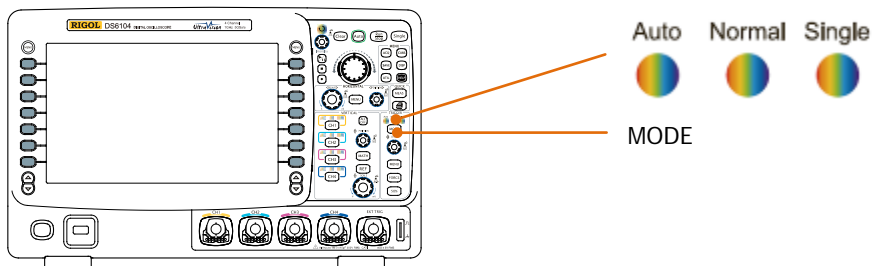
Auto:

No matter whether the trigger condition is met, there is always waveform display. A horizontal line is displayed when no signal is input.

When this mode is selected, the oscilloscope fills the pre-trigger buffer first and then search for a trigger while at the same time continues filling data. When the oscilloscope is searching for trigger, the data firstly filled into the pre-trigger buffer will overflow and be pushed out of the FIFO queue. When a trigger is found, the pre-trigger buffer would contain the data acquired before the trigger. If no trigger is found, the oscilloscope will trigger forcefully. If forceful trigger is invalid, the oscilloscope still displays waveform but the waveform is not stable, and if forceful trigger is valid, the oscilloscope will display stable waveform.

This trigger mode is applicable to low repetitive rate and unknown signal level. To display DC signals, this trigger mode must be used.

Note: when the horizontal time base is set to 50 ms/div or more, this trigger mode allows that there is no trigger signal.

Normal:

Display the waveform when the trigger condition is met or hold the original waveform and wait for the next trigger when the trigger condition is not met.

When this mode is selected, the oscilloscope fills the pre-trigger buffer first and then search for a trigger while at the same time continues filling data. When the oscilloscope is searching for trigger, the data firstly filled into the pre-trigger buffer will overflow and be pushed out of the FIFO queue. When a trigger is found, the oscilloscope will fill the post-trigger buffer and display the acquisition memory.

This trigger mode is applicable to signal with low repetitive rate and signal that does not require auto trigger.

Note: in this mode, press **FORCE** to generate a trigger signal forcefully.

Single:

After this mode is selected, the backlight of the **Single** key turns on, the oscilloscope waits for a trigger and displays the waveform when the trigger condition is met and then stops.

Note: in this mode, press **FORCE** to generate a trigger signal forcefully.

Trigger Coupling

Trigger coupling decides which kind of components will be transmitted to the trigger circuit. Note to distinguish it with "**Channel Coupling**".

- DC: allow the DC and AC components to pass the trigger path.
- AC: block all the DC components and attenuate signals lower than 8 Hz.
- LF reject: block the DC components and reject the low frequency components lower than 5 kHz.
- HF reject: reject the high frequency components higher than 50 kHz.

Press **MENU** → **Setting** → **Coupling** in the trigger control area (TRIGGER) at the front panel to select the desired coupling type (the default is DC).
Note: trigger coupling is only valid in edge trigger.

Trigger Holdoff

Trigger holdoff could trigger complicated waveforms (e.g. pulse sequence) stably. Holdoff time is the time the oscilloscope waits to re-enable the trigger circuit. During the holdoff time, the oscilloscope will not trigger.

Trigger position
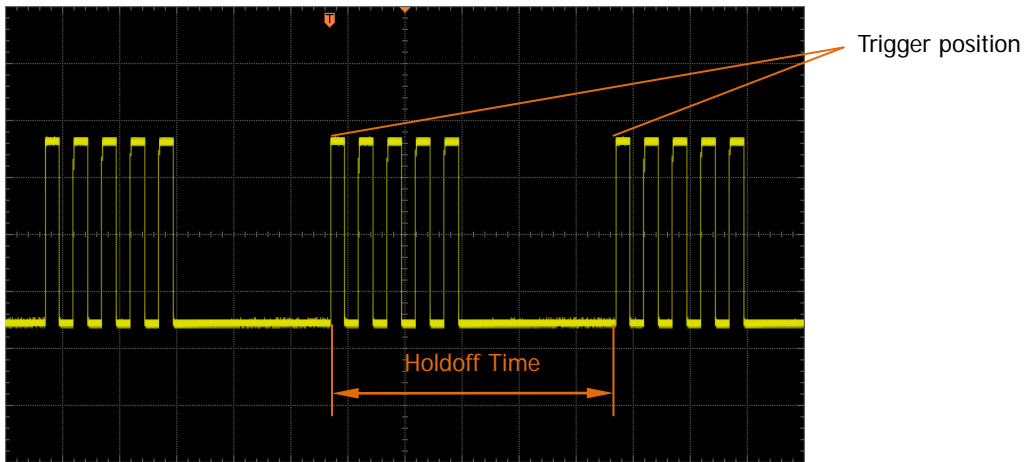
Holdoff Time

Press **MENU** → **Setting** → **Holdoff** in the trigger control area (TRIGGER) at the front panel and use ↻ to change the holdoff time (the default is 100 ns) until the waveform triggers stably. The adjustable range of the holdoff time is from 100 ns to 10 s.

Trigger Type

DS6000 has abundant trigger functions.
- Edge Trigger
- Pulse Trigger
- Slope Trigger
- Video Trigger
- Pattern Trigger
- RS232 Trigger
- I2C Trigger
- SPI Trigger
- CAN Trigger
- FlexRay Trigger
- USB Trigger

Edge Trigger

Trigger on the trigger threshold of the specified edge of the input signal.

Trigger Type:
Press **Type** to select "Edge". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:
Press **Source** to select CH1-CH4, EXT, EXT/5 or AC Line as the **Trigger Source**. The current trigger source is displayed at the upper right corner of the screen.
Note: select channel with signal input as trigger source to obtain stable trigger.

Edge Type:
Press **Slope** to select to trigger on which kind of edge of the input signal. The current edge type is displayed at the upper right corner of the screen.
- ⚡: trigger on the rising edge of the input signal when the voltage level meets the preset Trigger Level.
- ⚡: trigger on the falling edge of the input signal when the voltage level meets the preset Trigger Level.
- ⬡: trigger on the rising and falling edges of the input signal when the voltage level meets the preset Trigger Level.

Trigger Mode:
Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:
Press **Setting** to set the trigger parameters (trigger coupling and trigger holdoff) under this trigger type.

Trigger Level:
Use **TRIGGER** ⊙ **LEVEL** to modify the level. An orange trigger level line and the trigger mark " T " appear on the screen and move up and down with the turning of the knob, while at the same time, the trigger level value (e.g. Trig Level:760mV) at the lower left corner of the screen also changes accordingly. When stopping turning the knob, the trigger level line and the trigger mark disappear in about 2 seconds.

Pulse Trigger

Trigger on positive or negative pulse with specified width.

Trigger Type:
Press **Type** to select "Pulse". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:
Press **Source** to select CH1-CH4 or EXT as the **Trigger Source**. The current trigger source is displayed at the upper right corner of the screen.
Note: select channel with signal input as trigger source to obtain stable trigger.

Pulse Condition:
Press **When** to select the desired pulse condition.

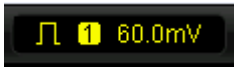● ▯⊦≻▯: trigger when the positive pulse width of the input signal is greater than the specified Pulse Width.

● ▭⊦▭: trigger when the positive pulse width of the input signal is lower than the specified Pulse Width.

● ▯⊦≺≻▯: trigger when the positive pulse width of the input signal is greater than the specified Lower Limit of Pulse Width and lower than the Upper Limit of Pulse Width.

● ▯⊦≻▯: trigger when the negative pulse width of the input signal is greater than the specified Pulse Width.

● ▭⊦▭: trigger when the negative pulse width of the input signal is lower than the specified Pulse Width.

● ▯⊦≺≻▯: trigger when the negative pulse width of the input signal is greater than the specified Lower Limit of Pulse Width and lower than the Upper Limit of Pulse Width.

Pulse Width Setting:

In this oscilloscope, positive pulse width is defined as the time difference between the two crossing points of trigger level and positive pulse. As shown in the figure below:
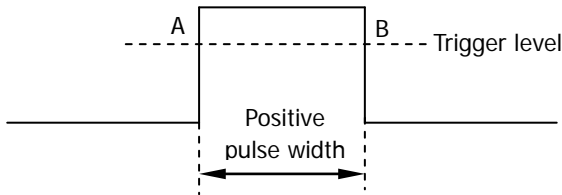
- When the **Pulse Condition** is set to ![icon], ![icon], ![icon] or ![icon], press **Setting** and use ![knob] to input the desired value. The range available is from 4 ns to 4 s.
- When the **Pulse Condition** is set to ![icon] or ![icon], press **Upper Limit** and **Lower Limit** respectively and use ![knob] to input the desired values. The range of the upper limit available is from 12 ns to 4 s. The range of the lower limit available is from 4 ns to 3.99 s. Note: the lower limit of the pulse width must be lower than the upper limit.

Trigger Mode:

Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:

Press **Setting** to set the trigger parameters (trigger holdoff) under this trigger type.

Trigger Level:

Use **TRIGGER** ⊚ **LEVEL** to modify the level. For details, please refer to the description of "**Trigger Level**" on page 5-8.

Slope Trigger

Trigger on the positive or negative slope with specified time.

Trigger Type:

Press **Type** to select "Slope". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:

Press **Source** to select CH1-CH4 as the **Trigger Source**. The current trigger source is displayed at the upper right corner of the screen.

Note: select channel with signal input as trigger source to obtain stable trigger.

Slope Condition:

Press **When** to select the desired slope condition.

- [icon] : trigger when the positive slope time of the input signal is greater than the specified Time.
- [icon] : trigger when the positive slope time of the input signal is lower than the specified Time.
- [icon] : trigger when the positive slope time of the input signal is greater than the specified Lower Limit of Time and lower than the specified Upper Limit of Time.
- [icon] : trigger when the negative slope time of the input signal is greater than the specified Time.
- [icon] : trigger when the negative slope time of the input signal is lower than the specified Time.
- [icon] : trigger when the negative slope time of the input signal is greater than the specified Lower Limit of Time and lower than the specified Upper Limit of Time.
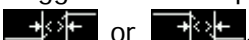
Time Setting:
In this oscilloscope, positive slope time is defined as the time difference between the two crossing points of trigger level line A and B with the positive edge. As shown in the figure below:
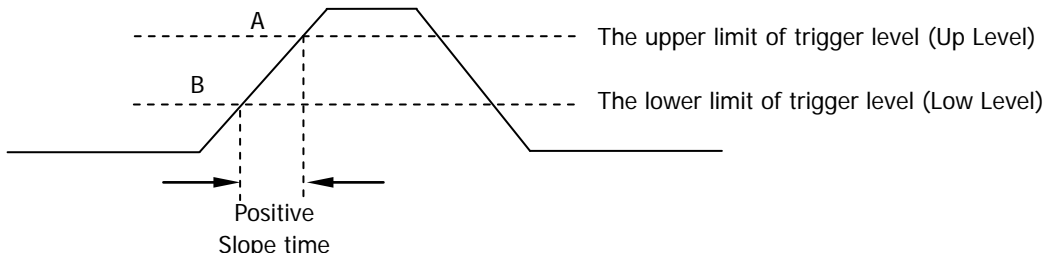
- When the **Slope Condition** is set to ▇▇, ▇▇, ▇▇ or ▇▇, press **Time** and use ↻ to input the desired value. The range available is from 10 ns to 1 s.
- When the **Slope Condition** is set to ▇▇ or ▇▇, press **Upper Limit** and **Lower Limit** respectively and use ↻ to input the desired values. The range of time upper limit available is from 20 ns to 1 s. The range of the time lower limit available is from 10 ns to 999 ms. Note: the time lower limit must be lower than the upper limit.

Vertical Window:
Press **Vertical** to select the desired vertical window.
You can select the boundaries of the trigger level and then use **TRIGGER** ◉ **LEVEL** to adjust the trigger level. During the adjustment, two orange trigger level lines and two trigger marks (▇ and ▇) appear on the screen and move up and down with the turning of the knob, while at the same time, the trigger level value and the slope value are displayed at the lower left corner of the screen. When stopping turning the knob, the trigger level lines and trigger marks disappear in about 2 seconds. When the **Slope Condition** is set to ▇▇ or ▇▇, the current trigger level and slope range will be displayed at the lower left corner of the screen.

Up Level: 200mV
Low Level: 166mV
Slew Rate: 34.0kV/s
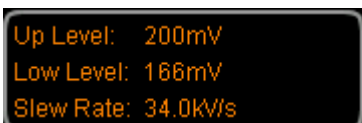
$$SlewRate = \frac{UpLevel - LowLevel}{TimeSettingValue}$$

Up Level: 148mV
Low Level: -362mV
Slew Rate: 255kV/s~510kV/s
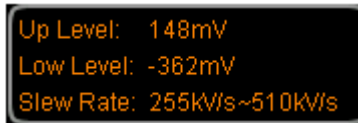
$$SlewRate = \frac{UpLevel - LowLevel}{TimeUpperLimit} \sim \frac{UpLevel - LowLevel}{TimeLowerLimit}$$

The adjustment mode of the trigger level is different when different vertical window is selected:

● ▱▱▱: only adjust the upper limit of the trigger level. During the adjustment, the "UP Level" and the "Slew Rate" change accordingly but the "Low Level" remains unchanged.

● ▱▱▱: only adjust the lower limit of the trigger level. During the adjustment, the "Low Level" and the "Slew Rate" change accordingly but the "UP Level" remains unchanged.

● ▱▱▱: adjust the upper and lower limits of the trigger level at the same time. During the adjustment, the "UP Level" and the "Low Level" change accordingly but the "Slew Rate" remains unchanged.

Trigger Mode:
Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:
Press **Setting** to set the trigger parameters (trigger holdoff) under this trigger type.

Video Trigger

Trigger on the standard video signal fields and lines of NTSC (National Television Standards Committee), PAL (Phase Alternating Line), SECAM (sequential color with memory) or HDTV (High Definition Television).

Trigger Type:
Press **Type** to select "Video". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:
Press **Source** to select CH1-CH4 as the **Trigger Source**. The current trigger source is displayed at the upper right corner of the screen.
Note: select channel with signal input as trigger source to obtain stable trigger.

Video Polarity:
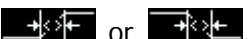Press **Polarity** to select the desired video polarity. The polarities available are normal polarity (█▌) and inverted polarity (▐█).

Sync:
Press **Sync** to select the desired sync type.
● All lines: trigger on all the horizontal sync pulses.
● Line number: for NTSC and PAL/SECAM video standards, trigger on the specified line in the odd or even field; for HDTV video standard, trigger on the specified line. Note that when this sync trigger mode is selected, you can modify the line number using ↻ in the **Line Num** menu with a step of 1. The range of the line number is from 1 to 525 (NTSC), 1 to 625 (PAL/SECAM), 1 to 525 (480P), 1 to 625 (576P), 1 to 750 (720P), 1 to 1125 (1080P) or 1 to 1125 (1080I).
● Odd field: trigger on the rising edge of the first ramp waveform pulse in the odd field.
● Even field: trigger on the rising edge of the first ramp waveform pulse in the even field.

Video Standard:
Press **Standard** to select the desired video standard.
● NTSC: the field frequency is 60 fields per second and the frame frequency is 30 frames per second. The TV sweep line is 525 with the even field goes first and the odd field follows behind.

- PAL: the frame frequency is 25 frames per second. The TV sweep line is 625 with the odd field goes first and the even field follows behind.
- SECAM: the frame frequency is 25 frames per second. The sweep line is 625 with interlacing sweep.
- HDTV: HDTV consists of 480P, 576P, 720P, 1080P and 1080I display formats. The specified video standards are as follows:

480P	the frame frequency is 60 frames per second; the TV sweep line is 525; line-by-line sweep; the line frequency is 31.5 kHz.
576P	the frame frequency is 60 frames per second; the TV sweep line is 625; line-by-line sweep.
720P	the frame frequency is 60 frames per second; the TV sweep line is 750; line-by-line sweep; the line frequency is 45 kHz.
1080P	the frame frequency is 60 frames per second; the TV sweep line is 1125; line-by-line sweep.
1080I	the field frequency is 50 to 60 fields per second; the frame frequency is 25 to 30 frames per second; the TV sweep line is 1125; interlacing sweep; the line frequency is 33.75 kHz.

Trigger Mode:
Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:
Press **Setting** to set the trigger parameters (trigger holdoff) under this trigger type.

Trigger Level:
Use **TRIGGER** ⊚ **LEVEL** to modify the level. For details, please refer to the description of "**Trigger Level**" on page 5-8.

Pattern Trigger

Identify the trigger condition through searching for the specified pattern. Pattern is the logic "AND" combination of each channel and the value of each channel can be H (High), L (Low) or X (Don't Care). One channel in the pattern can be specified with rising edge or falling edge and if the pattern settings of all the other channels are "True" (H or L), the oscilloscope will trigger on the specified edge. If no edge is specified, the oscilloscope will trigger on the last edge that makes the pattern as "True". If patterns of all the channels are set to "Don't Care", the oscilloscope would not trigger.

Trigger Type:
Press **Type** to select "Pattern". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Pat **1** 60.0mV
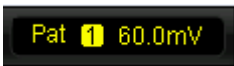
Channel Selection:
Press **Source** to select CH1-CH4 or EXT as the Trigger Source for H, L, X or edge condition. The current trigger source is displayed at the upper right corner of the screen.

Pattern Setting:
Press **Code** to set the pattern code of the current channel. At this point, the pattern setting area will be displayed at the bottom of the screen, see below.

Pat CH1 H XX CH4 X EXT
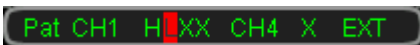
- **H**: set the pattern of the channel selected to "H", namely the voltage level is higher than the threshold level of the channel.
- **L**: set the pattern of the channel selected to "L", namely the voltage level is lower than the threshold level of the channel.
- **X**: set the pattern of the selected channel to "Don't Care", namely this channel is not a part of the pattern. When all the channels in the pattern are set to "Don't Care", the oscilloscope will not trigger.
- **⤒** or **⤓**: set the pattern to the rising or falling edge of the channel selected.

Note: in the pattern, you can only specify one rising edge or falling edge. If one edge item is currently defined and then another channel in the pattern is selected to define

another edge item, thus the former edge item defined will be replaced by X.

Pat CH1 H XX CH4 X EXT

Pat CH1 HX X CH4 X EXT

Trigger Mode:
Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:
Press **Setting** to set the trigger parameters (trigger holdoff) under this trigger type.

Trigger Level:
Use **TRIGGER** ⊚ **LEVEL** to modify the level. For details, please refer to the description of "**Trigger Level**" on page 5-8. Note that the trigger level of each channel needs to be set respectively.

RS232 Trigger

Trigger according to the start frame, error frame, check error or data. Below is the explanatory figure of RS232 protocol.

Trigger Type:
Press **Type** to select "RS232". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:
Press **Source** to select CH1-CH4 as the **Trigger Source**. The current trigger source is displayed at the upper right corner of the screen.

Polarity
Press **Polarity** to select the polarity of data transmission. It can be set to "Normal" or "invert" and the default is invert.

Trigger Condition:
Press **When** to select the desired trigger condition.
- Start: trigger on the start frame position.
- Error: trigger when error frame is detected. After this trigger condition is selected:
  --press **Stop Bit** to select "1 bit" or "2 bit";
  --press **Even-OddCheck** to select "None", "Even Checkout" or "Odd Checkout". The oscilloscope will determine error frame according to the preset parameters.
- Check Error: trigger when check error is detected. When this trigger condition is selected, press the **Even-OddCheck** softkey to select "Even Checkout" or "Odd Checkout". The oscilloscope will determine check error according to the preset parameters.
- Data: trigger on the last bit of the preset data bits and even-odd check bits.

When this trigger condition is selected:

--press **Data Bits** to select "5 bit", "6 bit", "7 bit" or "8 bit";

--press **Data** and input the data value according to the setting in **Data Bits** and the upper limit can be 31, 63, 127 or 255.

Baud Rate:

Press **Baud** to select the baud rate (equal to specifying a clock frequency) of the data transmission. The rates available include 2400 bps, 4800 bps, 9600 bps (default), 19200 bps, 38400 bps, 57600 bps, 115200 bps and user-defined. After "User" is selected, press **Setup** and use ↻ to set more specific value from 1 to 900000 with the adjustment step as 1 bps.

Trigger Mode:

Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:

Press **Setting** to set the trigger parameter (trigger holdoff) under this trigger type.

Trigger Level:

Use **TRIGGER** ⊚ **LEVEL** to modify the level. For details, please refer to the description of "**Trigger Level**" on page 5-8.

I2C Trigger

Trigger on the start condition, restart, stop and missing acknowledgement or on the read/write frame with specific device address and data value. When using I2C trigger, you need to specify the SCL and SDA data sources. The figure below shows the complete data transmission of I2C bus.

Trigger Type:
Press **Type** to select "I2C". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.
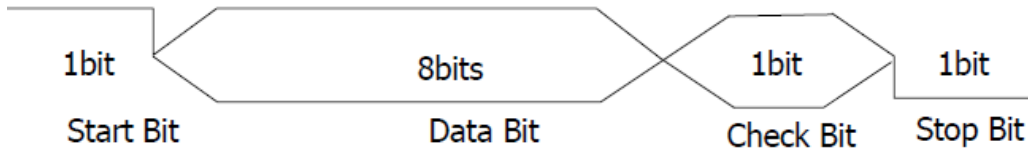

Source Selection:
Press **SCL** and **SDA** to specify the data sources of SCL and SDA respectively.
CH1-CH4 can be selected as the data source. The current trigger source is displayed at the upper right corner of the screen.

Trigger Condition:
Press **When** to select the desired trigger condition.
● Start: trigger when SDA data transitions from high to low while SCL is high.
● Restart: trigger when another start condition occurs before a stop condition.
● Stop: trigger when SDA data transitions from low to high while SCL is high.
● Missing ACK: trigger when the SDA data is high during any acknowledgement of SCL clock bit.
● Address: trigger on the clock (SCL) edge corresponding to the last byte data

(SDA) behind the preset address (Write, Read or R/W direction). After this trigger condition is selected:

--press **AddrBits** to select "7 bit" , "8 bit" or "10 bit";

--press **Address** to set the address value according to the setting in **AddrBits** and the ranges are from 0 to 127, 0 to 255 and from 0 to 1023 respectively;

--press **Direction** to select "Read", "Write" or "R/W" (note: when **AddrBits** is set to "8 bit", this setting is not available).

- Data: the trigger searches for the control byte value on the data line (SDA) following which there is a reading bit and an acknowledgement bit and then searches for the specified data value and qualifier. During this event, the oscilloscope will trigger on the clock edge of the acknowledgement bit behind the data byte. After this trigger condition is selected:

  --Press **ByteLength** to set the length of data and the range is from 1 to 5;

  --Press **CurrentBit** to select the desired data bit and the range is from 0 to (ByteLength×8-1);

  -- Press **Data** to set the data pattern of the current data bit to X, H or L.

  --Press **AllBits** to set the data pattern of all the data bits to the data pattern specified in **Data**.

- A&D: trigger when the "Address" and "Data" conditions are met at the same time. After this trigger condition is selected:

  --press **AddrBits** to select "7 bit" , "8 bit" or "10 bit";

  -- press   **Address** to set the address value according to the setting in **AddrBits** and the ranges are from 0 to 127, from 0 to 255 and from 0 to 1023 respectively;

  -- Press **ByteLength** to set the length of data and the range is from 1 to 5;

  -- Press **CurrentBit** to select the desired data bit and the range is from 0 to (ByteLength×8-1);

  -- Press **Data** to set the data pattern of the current data bit to X, H or L.

  -- Press **AllBits** to set the data pattern of all the data bits to the data pattern specified in **Data**.

  -- press **Direction** to select "Read", "Write" or "R/W" (note: when **AddrBits** is set to "8 bit", this setting is not available).

Trigger Mode:

Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.
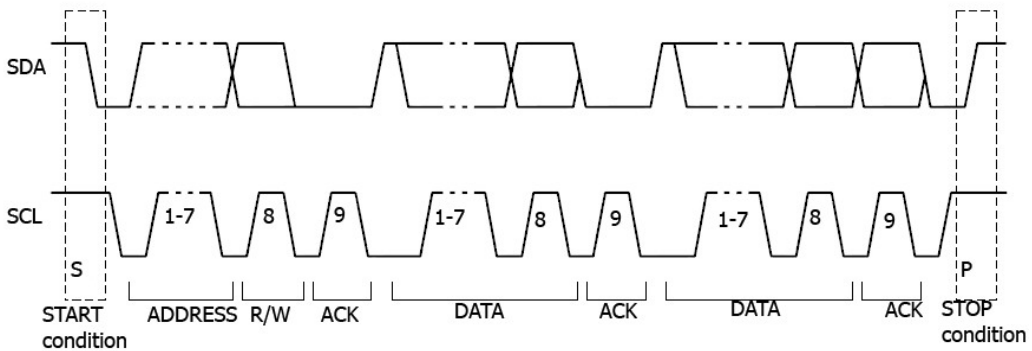
Trigger Setting:

Press **Setting** to set the trigger parameter (trigger holdoff) under this trigger type.

Trigger Level:

Use **TRIGGER** ⊛ **LEVEL** to modify the trigger level of SCL or SDA channel. For details, please refer to the description of "**Trigger Level**" on page 5-8.

SPI Trigger

Trigger on the data pattern with specified edge. When using SPI trigger, you need to specify the SCL, SDA and CS data sources. Below is the sequence chart of SPI bus data transmission.
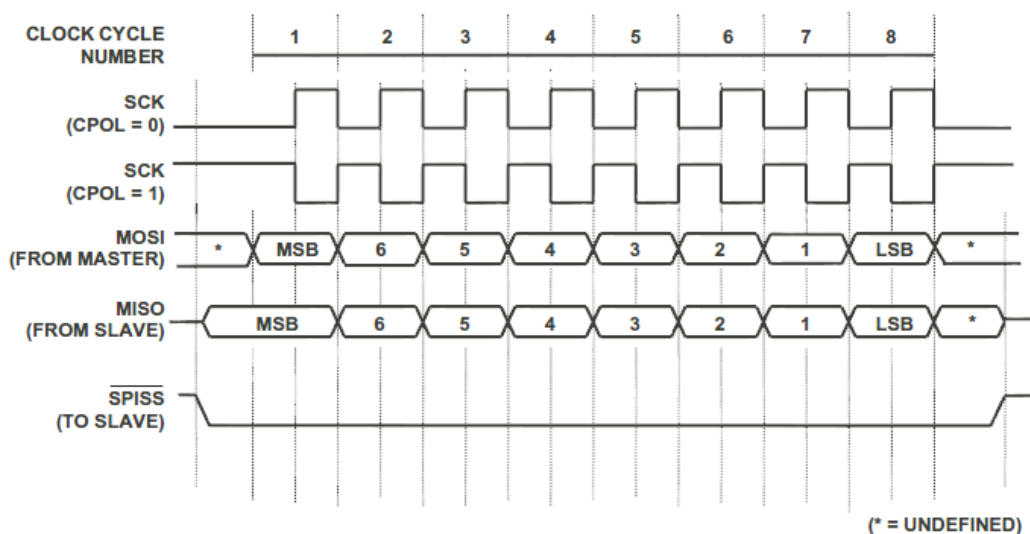
Trigger Type:
Press **Type** to select "SPI". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:
Press **SCL** and **SDA** to specify the data sources of SCL and SDA respectively. CH1-CH4 can be selected as the data source. The current trigger source is displayed at the upper right corner of the screen.

Trigger Condition:
Press **When** to select the desired trigger condition.
● CS: when this trigger condition is selected, press **CS** to specify the data source (CH1 to CH4) of CS. Press **Mode** to set the oscilloscope to trigger when the CS channel is "H" or "L".

● Timeout: when this trigger condition is selected, press **Timeout** to set the minimum time that the clock (SCL) signal must be idle before the oscilloscope will search for the data (SDA) on which to trigger. The range is from 100 ns to 1 s.

Data Line Setting:

Set to trigger after the data channel (SDA) has transmitted data with specified bits and length.

● Press **Data Bits** to set the number of bits of the serial data character string. It can be set to any integer between 4 and 32.
● Press **CurrentBit** to input the data of the current bit and the range is from 0 to (the specified value in **Data Bits** -1).
● Press **Data** to set the value of the current bit to H, L or X.
● Press **AllBits** to set all the bits to the value specified in **Data** immediately.

Clock Edge:

Press **ClockEdge** to select the desired clock edge.

● 🔼: sample the SDA data on the rising edge of the clock.
● 🔽: sample the SDA data on the falling edge of the clock.

Trigger Mode:

Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:

Press **Setting** to set the trigger parameter (trigger holdoff) under this trigger type.

Trigger Level:

Use **TRIGGER** ⊚ **LEVEL** to modify the level. For details, please refer to the description of "**Trigger Level**" on page 5-8.

CAN Trigger

Trigger on the specified frame type of the data frame. When using CAN trigger, you need to specify the signal source, signal rate and trigger signal type of the CAN signal. The figure below shows the standard and expanded formats of CAN bus data frame.
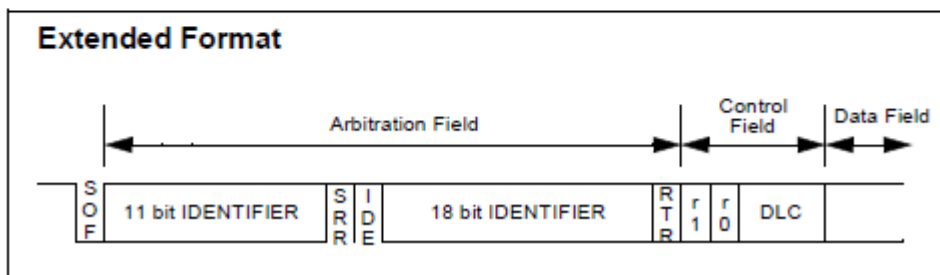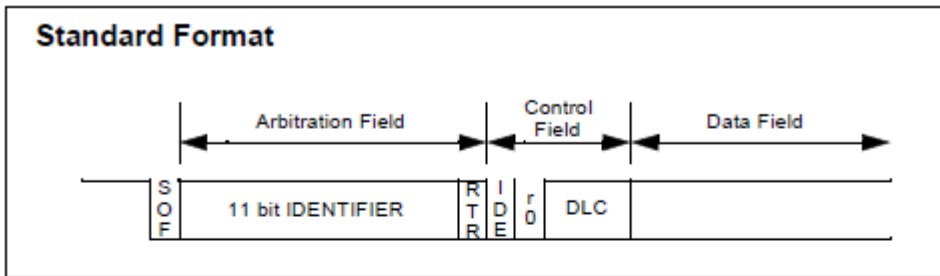
Trigger Type:
Press **Type** to select "CAN". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:
Press **Source** to select CH1, CH2, CH3 or CH4 as the **Trigger Source**. The current trigger source is displayed at the upper right corner of the screen.

Signal Type:
Press **Signal Type** to select the desired signal type.
● **Rx:** receiving signal from the CAN signal line.
● **Tx:** transmission signal from the CAN signal line.

- **CAN_H:** actual CAN_H bus signal.
- **CAN_L:** actual CAN_L bus signal.
- **Differential:** CAN differential bus signal connected to the analog channel by using the differential probe.

Trigger Condition:

Press **When** to select the desired trigger condition.

- **SOF:** trigger on the start frame position of the data frame.
- **EOF:** trigger on the end frame position of the data frame.
- **Frame Type:** after this type is selected, press **Frame Type** to select to trigger on "Data", "Remote", "Error" or "OverLoad".

    "Data" trigger:

    -- Press **ID Setup** and select "Specific ID" or "Random ID". When "Specific ID" is selected, press **ID Format** and select "Standard" or "Expand"; press **ID Data** and use ↻ to set the desired value. The range is from 0 to 2047 (standard ID format) or 0 to 536870911 (expanded ID format).

    -- Press **Byte Length** and use ↻ to set the desired value. The range is from 1 to 8.

    -- Press **CurrentBit** to select the desired data bit and the range is from 0 to (Bits×8-1).

    --Press **Data** to set the data pattern of the current bit to X, H or L.

    --Press **AllBits** to set the data pattern of all the data bits to the data pattern specified in **Data**.

    "Remote" trigger:

    --Press **ID Setup** and select "Specific ID" or "Random ID". When "Specified ID" is selected, press **ID Format** and select "Standard" or "Expand"; press **ID Data** and use ↻ to set the desired value. The range is from 0 to 2047 (standard ID format) or 0 to 536870911 (expanded ID format).

- **Frame Error:** after selecting this type, press **Error Type** to select to trigger on the "Bit Fill", "Answer Error", "Check Error", "Format Error" or "Random Error".

Baud:

Press **Baud Rate** to set the CAN baud to match the CAN bus signal and the values available are 10 kb/s (default), 20 kb/s, 33.3 kb/s, 50 kb/s, 62.5 kb/s, 83.3 kb/s, 100 kb/s, 125 kb/s, 250 kb/s, 500 kb/s, 800 kb/s, 1 Mb/s or user. After "User" is selected, press **User** and use ↻ to input the desired rate within 1 kb/s and 10.0 Mb/s (the default is 1 Mb/s).

Sample Point:

Sample point is the point within the bit time. The oscilloscope samples the bit level at this point. The sample point position is represented by the percentage that the "time from the start of the bit to the sample point" takes up in the "bit time".

Press **SamplePoint** and use ↻ to modify the parameter with the step as 1% and the range is from 5% to 95%.
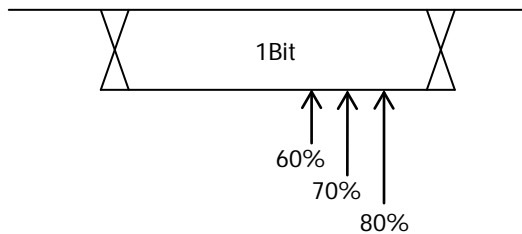
```
                          1Bit

                      60%
                         70%
                            80%
```

Trigger Mode:

Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:

Press **Setting** to set the trigger parameter (trigger holdoff) under this trigger type.

Trigger Level:

Use **TRIGGER** ◎ **LEVEL** to modify the level. For details, please refer to the description of "**Trigger level**" on page 5-8.

FlexRay Trigger

Trigger on the frame, symbol, error or TSS (Transmission Start Sequence) of FlexRay bus. FlexRay is a kind of differential serial bus configured with three continuous segments (namely packet header, payload and packet end). Its data transmission rate is up to 10 Mbps. Each frame contains a static and dynamic segment and ends with the bus idle time. The physical bus adopts unshielded twisted piar, but shielded twisted pair can be used to improve EMC performance. The figure below shows the frame format of FlaxRay protocol.

Trigger Type:
Press **Type** to select "FlexRay". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:
Press **Source** to specify data source for FlexRay bus. CH1, CH2, CH3 or CH4 can be selected as the trigger source. The current signal source is displayed at the upper right corner of the screen.

Baud:
Press **Baud** to set the signal rate to "2.5 Mb/s", "5 Mb/s" or "10 Mb/s".

Trigger Condition:

Press **When** to select the desired trigger condition.

- **FRAME:** trigger on the frame of FlexRay bus.
- **SYMBOL:** trigger on the CID (Channel Idle Delimiter), CAS (Collision Avoidance Symbol), MTS (Media Access Test Symbol) and WUP (Wakeup Pattern) of FlexRay bus.
- **ERROR:** trigger when error occurs to FlexRay bus, including header CRC error and frame CRC error.
- **TSS:** trigger on the transmission start sequence of FlexRay bus.

Note: as the occurrence probability of specified FlaxRay frame is very low, it is recommended to set the oscilloscope to "Normal" trigger mode to prevent the instrument from triggering automatically on the specified frame. The same goes for "Error" trigger condition. What's more, the trigger holdoff might need to be adjusted to view a specified error when multiple errors occur at the same time.

Trigger Mode:

Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:

Press **Setting** to set the trigger parameters (trigger holdoff) under this trigger type.

Trigger Level:

Use **TRIGGER** 🔘 **LEVEL** to modify the level. For details, please refer to the description of "**Trigger Level**" on page 5-8.

USB Trigger

Trigger on the SOP, EOP, RC, Suspended and Exit Suspended of the data packet on the differential USB data cable (D+ and D-). This trigger supports USB Low Speed and Full Speed. The figure below shows the USB data transmission protocol.

Trigger Type:
Press **Type** to select "USB". At this point, the trigger setting information as shown in the figure below is displayed at the upper right corner of the screen.

Source Selection:
Press **D+** and **D-** to specify data sources for D+ and D- data cables respectively. CH1-CH4 can be selected as the data source. The current trigger source is displayed at the upper right corner of the screen.

Signal Speed:
Press **Speed** to select "Low Speed" (1.5 Mb/s) or "Full Speed" (12 Mb/s).

Trigger Condition:
Press **When** to select the desired trigger condition.
- **SOP:** trigger at the sync bit at the beginning of the data packet (SOP).
- **EOP:** trigger at the end of the SEO portion of the data packet (EOP).
- **RC:** trigger when SEO is greater than 10 ms.
- **Suspended:** trigger when the idle time of the bus is greater than 3 ms.
- **ExitSuspended:** trigger when the bus exits from idle state for more than 10 ms.

Trigger Mode:
Press **Sweep** to select the **Trigger Mode** (page 5-3) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.
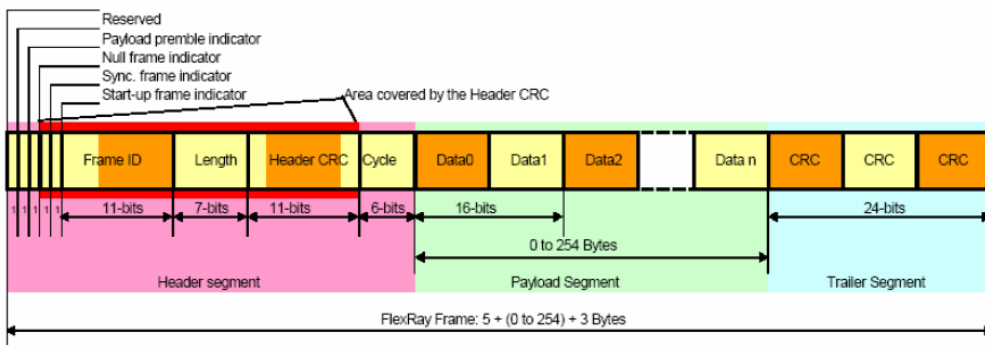
Trigger Setting:
Press **Setting** to set the trigger parameter (trigger holdoff) under this trigger type.

Trigger Level:
Use **TRIGGER** 🔘 **LEVEL** to modify the level. For details, please refer to the description of "**Trigger Level**" on page 5-8.

Trigger Ouput Connector

The trigger output connector at the rear panel can output trigger signals decided by the current setting.
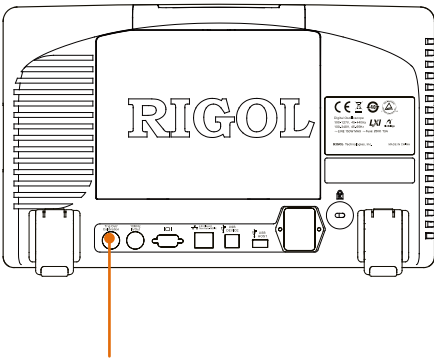
Trigger Output Connector

Press **UTIL** → **AuxOutput** to select "TrigOut". When the oscilloscope is triggered, it will output a trigger signal decided by the current setting through the **[Trig Out/Calibration]** interface.

6 To Make Measurements

After data is sampled and displayed, DS6000 can make math operation, cursor measurement and auto measurement.

The contents of this chapter:

- Math Operation
- Auto Measure
- Cursor Measurement

Math Operation

DS6000 can realize various math operations of waveforms between channels including addition (A+B), subtraction (A-B), multiplication (AxB), division (A÷B), FFT, logic operation and advanced operation. The results of math operation also allows further measurement (for details, please refer to "**Cursor Measurement**").
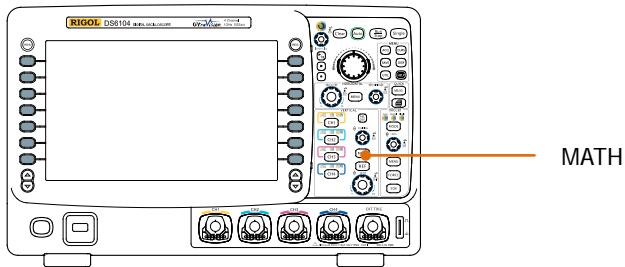
MATH

Press **MATH** → **Operate** in the vertical control area (VERTICAL) at the front panel to select the desired operation function. The result of the math operation is displayed on the waveform marked with "M" on the screen.

Addition

Add the waveform voltage values of signal source A and B point by point and display the results.

Press **MATH** → **Operate** to select "A+B":
● Press **Source A** and **Source B** to select the desired channels. The channels available are CH1, CH2, CH3 and CH4.
● Press **Invert** to turn the invert display of the operation results on or off.
● **VERTICAL** 🔘 **POSITION**, **VERTICAL** 🔘 **SCALE**, **HORIZONTAL** 🔘 **POSITION** and **HORIZONTAL** 🔘 **SCALE** can also be used to adjust the display of the operation results.

Subtraction

Subtract the waveform voltage values of signal source B from A point by point and display the results.

Press **MATH** → **Operate** to select "A-B":
- Press **Source A** and **Source B** to select the desired channels. The channels available are CH1, CH2, CH3 and CH4.
- Press **Invert** to turn the invert display of the operation results on or off.
- **VERTICAL** ◎ **POSITION**, **VERTICAL** ◎ **SCALE**, **HORIZONTAL** ◎ **POSITION** and **HORIZONTAL** ◎ **SCALE** can also be used to adjust the display of the operation results.

Multiplication

Multiply the waveform voltage values of signal source A and B point by point and display the results.

Press **MATH** → **Operate** to select "A x B":
- Press **Source A** and **Source B** to select the desired channels. The channels available are CH1, CH2, CH3 and CH4.
- Press **Invert** to turn the invert display of the operation results on or off.
- **VERTICAL** ◎ **POSITION**, **VERTICAL** ◎ **SCALE**, **HORIZONTAL** ◎ **POSITION** and **HORIZONTAL** ◎ **SCALE** can also be used to adjust the display of the operation results.

Division

Divide the waveform voltage values of signal source A by B point by point and display the results. It can be used to analyze the multiple relationships of waveforms in two channels. Note that when the voltage value of channel B is 0, the result of the division is treated as 0.

Press **MATH** → **Operate** to select "A ÷ B":
- Press **Source A** and **Source B** to select the desired channels. The channels available are CH1, CH2, CH3 and CH4.
- Press **Invert** to turn the invert display of the operation results on or off.
- **VERTICAL** ◎ **POSITION**, **VERTICAL** ◎ **SCALE**, **HORIZONTAL** ◎ **POSITION** and **HORIZONTAL** ◎ **SCALE** can also be used to adjust the display of the operation results.

FFT

FFT is used to quickly perform Fourier transform on specified signals and transform time domain signals to frequency domain signals. Use FFT to facilitate the following works:

● Measure harmonic components and distortion in the system
● Measure the characteristics of the noise in the DC power
● Analyze vibration

Press **MATH** → **Operate** to select "FFT" and set the parameters of FFT operation.
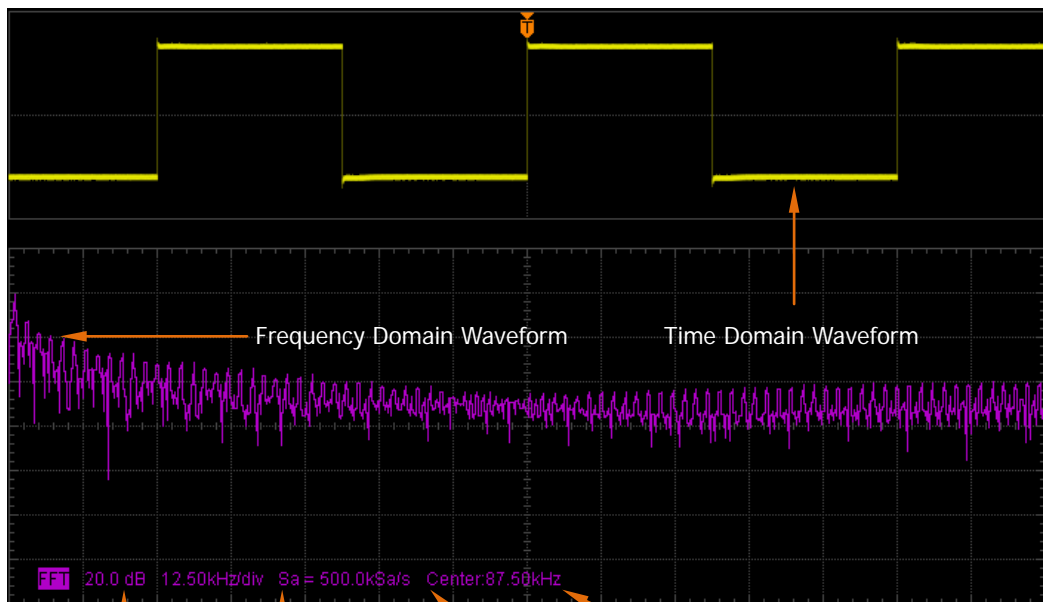
Vertical Amplitude   Horizontal Scale   FFT Sample Rate   Center Frequency

1. **Source Selection**
    Press **Source** to select the desired channel. The channels available are CH1, CH2, CH3 and CH4.

2. **Select Window Function**
    Spectrum leakage can be considerably decreased when a window function is used. DS6000 provides four kinds of FFT window functions with different

characteristics and are applicable to measure different waveforms. You need to select the window function according to different waveforms and characteristics. Press **Window** to select the desired window function and the default is "Rectangle".

Table 6-1 Window Function

Window	Characteristics	Waveforms Suitable for Measurement
Rectangle	The best frequency resolution The poorest amplitude resolution Similar to the situation when no window is multiplied	Transient or short pulse, the signal levels before and after the multiplication are basically the same; Sine waveforms with the same amplitude and rather similar frequencies; Wide band random noise with relatively slowly changing waveform spectrum.
Hanning	Better frequency resolution poorer amplitude resolution	Sine, periodic and narrow band random noise.
Hamming	A little bit better frequency resolution than Hanning	Transient or short pulse, the signal levels before and after the multiplication are rather different.
Blackman	The best amplitude resolution The poorest frequency resolution	Single frequency signal, search for higher order harmonics.

3. **Set the Display Mode**
    Press **Display** to select "Split" (default) or "Full Screen" display mode.
    Split: the source channel and the FFT operation results are displayed separately. The time domain and frequency domain signals are straightforward.
    Full Screen: the source channel and the FFT operation results are displayed in the same window to view the frequency spectrum more clearly and to perform more precise measurement.
    Note: in FFT mode and when MATH is the active channel, you can also press **HORIZONTAL** ⊙ **SCALE** to switch to "Split" or "Full Screen".

4. **Set the Vertical Scale**

    In FFT measurement, the unit of the horizontal axis changes from time to frequency. Use **HORIZONTAL** ⊚ **SCALE** to set the scale and position of the horizontal axis respectively.

    The unit of the vertical axis can be dB or Vrms which display the vertical amplitude in logarithmic mode and linear mode respectively. If you need to display the FFT frequency spectrum in a relatively larger dynamic range, dB is recommended. Press **Scale** to select the desired unit and the default is Vrms. Use **VERTICAL** ⊚ **SCALE** and **HORIZONTAL** ⊚ **SCALE** to set the vertical amplitude and position of the FFT frequency spectrum respectively.

    Tips

    - 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 Mode**" of the oscilloscope to "Average".

Logic Operation

Perform logic operation on the waveform voltage values of the specified source point by point and display the results. In operation, when the voltage value of the source channel is greater than the threshold of the corresponding channel, it is regarded as logic "1"; otherwise logic "0". The following common logic operation expressions are provided.

Table 6-2 Logic Operation

Operation	Description
AND	The results of logic AND operation of two binary bits are as follows: A B A AND B 0 0 0 0 1 0 1 0 0 1 1 1
OR	The results of logic OR operation of two binary bits are as follows: A B A OR B 0 0 0 0 1 1 1 0 1 1 1 1
NOT	The results of logic NOT operation of one binary bit are as follows: A NOT A 0 1 1 0
XOR	The results of logic XOR operation of two binary bits are as follows: A B A XOR B 0 0 0 0 1 1 1 0 1 1 1 0

Press **MATH** → **Operate** to select "Logic":

● Press **Expression** to select the desired operation expression and the default is AND.

● Press **Source A** and **Source B** to select the desired channels. The channels available are CH1, CH2, CH3 and CH4.

● Press **Invert** to turn the invert display of the operation results on or off.

● **VERTICAL** ◎ **POSITION**, **VERTICAL** ◎ **SCALE**, **HORIZONTAL** ◎ **POSITION** and **HORIZONTAL** ◎ **SCALE** can also be used to adjust the display of the operation results.

● Press **Threshold A** and use ↻ to set the threshold of source A in logic operation.

● Press **Threshold B** and use ↻ to set the threshold of source B in logic operation.

Advanced Operation

DS6000 provides advanced operation function that allows user-defined operation functions. Press **MATH** → **Operate** → "Advance" → **Expression** → "On", and the editing window as shown in the figure below is displayed on the screen.
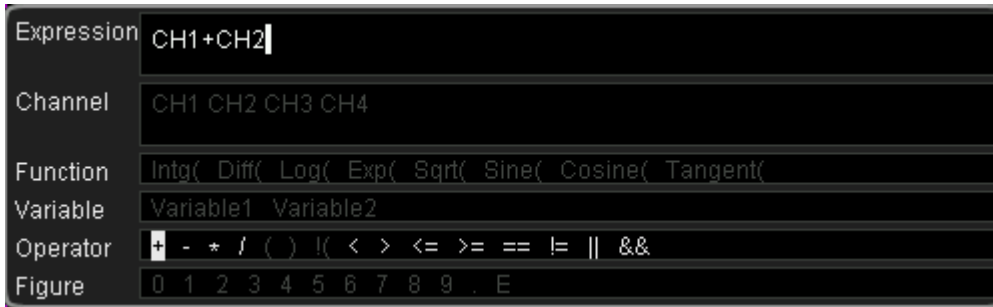
Turn ↻ to select any item (if they are currently available for selection) in "Channel", "Function", "Variable", "Operator" and "Figure", then press down the knob and the item selected will de displayed in the entry box on the right of "Expression".

During the expression edit, you can press **Delete** to delete the character currently at the left of the cursor and press **Clear** to delete all the characters in the entry box at any time.

After finishing the expression edit, press **Apply** and the oscilloscope will operate according to the expression you set and display the result. Note: after pressing **Apply** , the **Expression** menu will be automatically set to "OFF" but the preset expression will still be displayed at the bottom of the screen for your reference. You can also press **Invert** to turn the invert display of the operation results on or off.

The following are descriptions of the contents in the editing window:

1. **Expression**

   Here, it refers to the formulas formed by channel, function, variable, operator and figure. The length of the expression is limited to 64 bytes.

2. **Channel**

   You can select any of CH1, CH2, CH3 and CH4.

3. **Function**

   Please refer to the following table to get the functions of each function. Note: the left brackets "(" here are only used to facilitate your entry and they are not a part of the function name.

Table 6-3 Function

Name	Function
Intg(	Calculate the integral of the selected source. You can use integral to measure the area under a waveform or the pulse energy.
Diff(	Calculate the discrete time derivative of the selected source. You can use differentiate to measure the instantaneous slope of a waveform.
Log(	Calculate the natural logarithm of the selected source (use constant e, approximately 2.718282 as the base).
Exp(	Calculate the exponent of the selected source. For example: Exp(A) means calculating the Ath power of e.
Sqrt(	Calculate the square root of the selected source.
Sine(	Calculate the sine value of the selected source.
Cosine(	Calculate the cosine value of the selected source.
Tangent(	Calculate the tangent value of the selected source.

4. **Variable**

   Users can set Variable1 and Variable2 to desired values. Press **Variable** and turn on the setting menu as follows:
   - **Variable:** press this softkey to select the variable needed to be set as "Variable1" and "Variable2".
   - **Step:** press this softkey to set the step used when using the knob to modify

"Mantissa". The steps available are × 1, × 0.1, × 0.01, × 0.001 and × 0.0001.

- **Mantissa:** press this softkey to set the valid number of digits of the variable. After setting the "Step" or pressing this softkey, turn the knob to modify this parameter. The range available is from -9.9999 to 9.9999.
- **Exponent:** press this softkey to set the numeric values of the exponents with 10 as the base in the variables. The range is from -9 to 9.

For example, the following setting means set Variable1 to 6.1074×10^8.

Variable: Variable1

Mantissa: 6.1074

Exponent: 8

5. **Operator**

Please refer to the following table to get the functions of each operator.

Table 6-4 Operator

Operator	Function
+ - * /	Arithmetic operators: add, subtract, multiply and divide
( )	Parentheses: used to increase the priority of the operation enclosed in the parentheses
< > <= >= == !=	Relationship operators: lower than, greater than, lower than or equal to, greater than or equal to, equal to, not equal to
!(|| &&	Logic operator: NOT, OR, AND

Note: in division operation, when the divisor is 0, the result of the division is treated as 0.

6. **Figure**

Select from figure 0 to 9, decimal point and character E. Where, character E is used to refer to the nth power of 10. For example, 1.5E3 means 1.5×10^3.

Auto Measurement

DS6000 provide auto measurements of 24 waveform parameters and the statistic and analytical functions of the measurement results. What's more, you can also use the frequency counter to realize more precise frequency measurement.
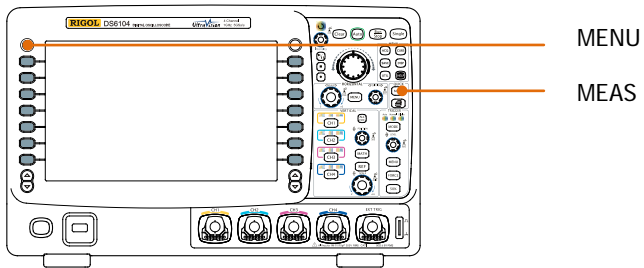
MENU

MEAS

Quick Measurement after Auto

When the oscilloscope is correctly connected and has detected input signal, press Auto enable the waveform auto setting function and open the following function menu.
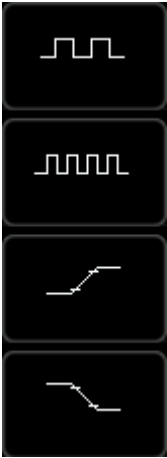
Single-period: measure the "Period" and "Frequency "of the current signal within a single period and display the measurement results at the bottom of the screen.

Multi-period: measure the "Period" and "Frequency "of the current signal within multiple periods and display the measurement results at the bottom of the screen.

Rise Time: measure the "Rise Time" of the current signal and display the measurement result at the bottom of the screen.

Fall Time: measure the "Fall Time" of the current signal and display the measurement result at the bottom of the screen.

One-key Measurement of 24 Parameters

Press **MENU** at the left of the screen to turn on the measurement menu of the 24 parameters and then press the corresponding menu softkey to quickly realize "One-key" measurement. The measurement result will be displayed at the bottom of the screen.

The icons of time and voltage parameters in the measurement items and the measurement results on the screen are always marked in the same color of the channel (**MEAS** → **Source**) currently used. But the delay and phase measurement items are always marked in green. For example,

Parameter Icons:

Measurement Results:

Note: if the measurement result is displayed as "*****", it means that there is no signal input in the current source or the measurement result is not within the valid range (too large or too small).

Time Parameters
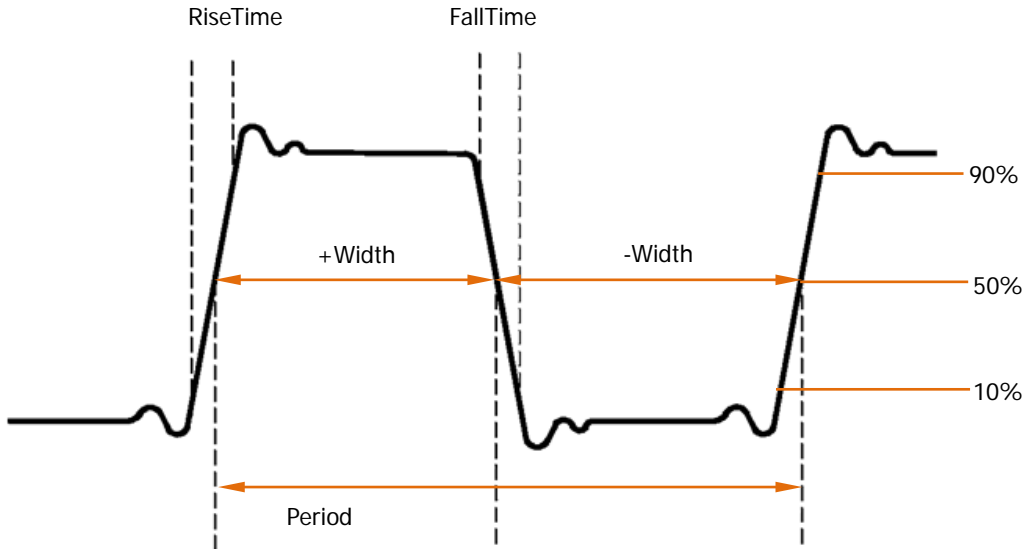
1. **Period:** defined as the time between the two crossing points of the middle threshold values of two continuous edges with the same polarity.
2. **Frequency:** defined as the reciprocal of period.
3. **Rise Time:** the time that the signal amplitude rises from 10% to 90%.
4. **Fall Time:** the time that the signal amplitude falls from 90% to 10%.
5. **+ Width:** the time from the 50% threshold of a rising edge to the 50% threshold of the next falling edge of a pulse.
6. **- Width:** the time from the 50% threshold of a falling edge to the 50% threshold of the next rising edge of a pulse.
7. **+ Duty:** the ratio of the positive pulse width to the period.
8. **- Duty:** the ratio of the negative pulse width to the period.

Delay and Phase
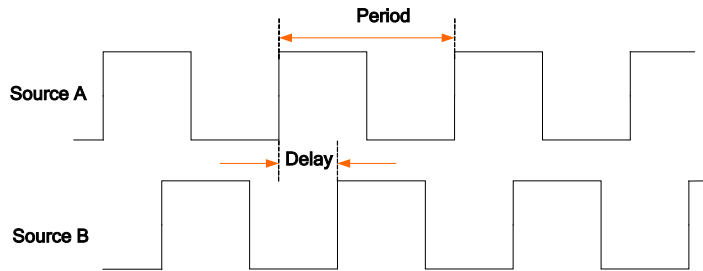
Source A and source B can be any of CH1, CH2, CH3 and CH4.

1. **Delay A→B⨍:** the time deviation between the rising edges of source A and source B. Negative delay means that the rising edge of source A appears later than that of source B.

2. **Delay A→B⨪:** the time deviation between the falling edges of source A and source B. Negative delay means that the falling edge of source A appears later than that of source B.

3. **Phase A→B⨍:** phase deviation in degree calculated according to "**Delay A→B⨍**" and the period of source A.

4. **Phase A→B⨪:** phase deviation in degree calculated according to "**Delay A→B⨪**" and the period of source A.

$$\text{Phase} = \frac{\text{Delay}}{\text{The Period of Source A}} \times 360°$$

Voltage Parameters
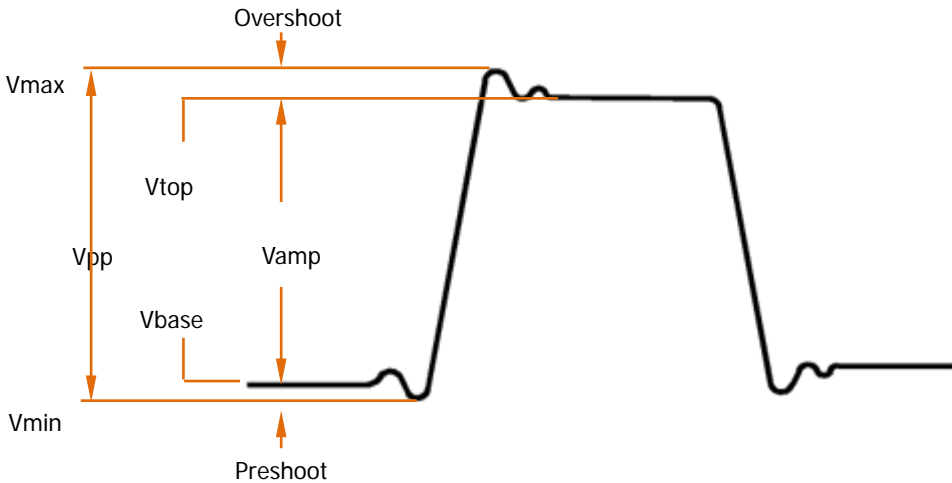
1. **Vmax:** the voltage value from the highest point of the waveform to the GND.
2. **Vmin:** the voltage value from the lowest point of the waveform to the GND.
3. **Vpp:** the voltage value from the highest point to the lowest point of the waveform.
4. **Vtop:** the voltage value from the flat top of the waveform to the GND.
5. **Vbase:** the voltage value from the flat base of the waveform to the GND.
6. **Vamp:** the voltage value from the top of the waveform to the base of the waveform.
7. **Vavg:** the arithmetic average value on the whole waveform or on the gating area.

$Average = \dfrac{\sum x_i}{n}$, where, x_i is the ith point being measured, n is the number of points being measured.

8. **Vrms**: the root mean square value on the whole waveform or the gating area.

$RMS = \sqrt{\dfrac{\sum\limits_{i=1}^{n} x_i^2}{n}}$, where, x_i is the ith point being measured, n is the number of points being measured.

9. Overshoot: the ratio of the difference of the maximum value and top value of the waveform to the amplitude value.

10. Preshoot: the ratio of the difference of the minimum value and base value of the waveform to the amplitude value.

Other Parameters

1. **Area:** the area of the whole waveform within the screen and the unit is voltage-second. The area meadured above the zero reference (namely the vertical offset) is positive and the area measured below the zero reference is negative. The area measured is the algebraic sum of the area of the whole waveform within the screen.

2. **Period Area:** the area of the first period of waveform on the screen and the unit is voltage-second. The area above the zero reference (namely the vertical offset) is positive and the area below the zero reference is negative. The area measured is the algeraic sum of the area of the whole period waveform. Note that when the waveform on the screen is less than a period, the period area measured is 0.

Frequency Counter Measurement

The hardware frequency counter supplied with this oscilloscope can make more precise measurement of the signal frequency.

Press **MEAS** → **Counter** to select anyone of CH1-CH4 as the source. The measurement result is displayed at the upper right corner of the screen and you can identify the current measurement source according to the color of the icon. The following figure is the result of the frequency measurement of the signal from CH1.

If "OFF" is selected, the frequency counter measurement function is turned off.

Measurement Setting

1. **Source Selection**
    Press **MEAS** → **Source** to select the desired channel for measurement
    (CH1-CH4 or MATH). The color of the parameter icon under **MENU** at the left of
    the screen will change with the source selected.

2. **Measurement Range**
    Press **MEAS** → **Meas.Range** → **Meas.Range** to select "Screen Region" or
    "Cursor Region" for measurement. When "Cursor Region" is selected, two cursor
    lines appear on the screen. At this point, press **Cursor A** and **Cursor B** and use
    ↻ to adjust the positions of the two cursor lines respectively so as to determine
    the measurement range. Or, press **CursorAB** and use ↻ to adjust the
    positions of cursor A and cursor B at the same time. Note: you can press ↻
    continuously to switch among the cursors.
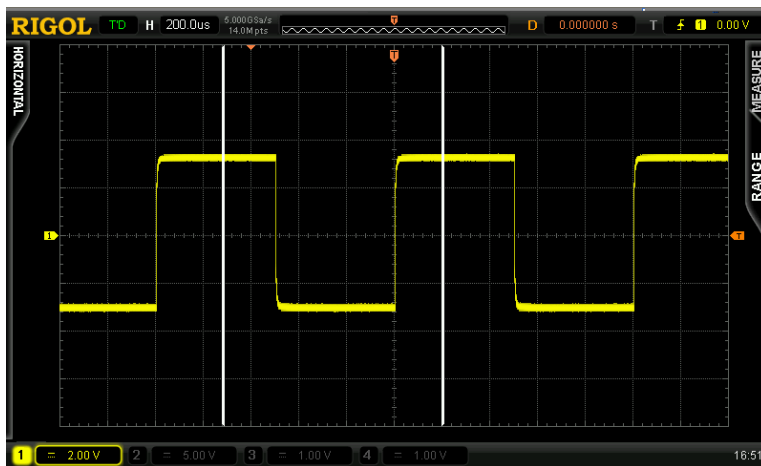
3. **Delayed Measurement Setting**
    Specify the source A and source B in the measurement items "**Delay A→B⌐**"
    and "**Delay A→B⌐**".
    Press **MEAS** → **Meas Setting** → **Type** → "Delay" and then press **Source A**
    and **Source B** to set the two channel sources (CH1-CH4) of the delayed
    measurement respectively.

4.   Phase Measurement Setting

Specify the source A and source B in the measurement items "**Phase A→B↲**"

and "**Phase A→B↳**".

Press **MEAS** → **Meas Setting** → **Type** → "Phase" and then press **Source A**

and **Source B** to set the two channel sources (CH1-CH4) of the phase

measurement respectively.

5.   Threshold Measurement Setting

Specify the vertical level (in percentage) being measured in the analog channel.

Measurement of all the time, delay and phase parameters will be influenced by

this setting.

Press **MEAS** → **Meas Setting** → **Type** → "Threshold" and then:

●   Press **Max** and use ↻ to set the maximum value of the measurement.
     Reducing the maximum value to the current "Mid" will automatically reduce
     the middle value to keep it lower than the maximum value. The default is
     90% and the range available is from 7% to 95%.

●   Press **Mid** and use ↻ to set the middle value of measurement. The middle
     value is limited by the settings of "Max" and "Min". The default is 50% and
     the range available is from 6% to 94%.

●   Press **Min** and use ↻ to set the minimum value of the measurement.
     Increasing the minimum value to the current "Mid" will automatically
     increase the middle value to keep it higher than the minimum value. The
     default is 10% and the range available is from 5% to 93%.

Clear the Measurement

If you have currently turned on one or more items in the 24 measurement parameters, you can "Delete" or "Recover" the 5 parameters ahead or "Delete" or "Recover" all the measurement items that have been turned on. Note that the 5 parameters ahead is determined according to the order in which you turned them on and they will not change as you delete one or more measurement items.

Press **MEAS** → **Clear** → **Item n** to "Delete" or "Recover" the specified measurement item. When one measurement item is deleted or recovered, the measurement result at the bottom of the screen will move one-item left or right.

Press **MEAS** → **Clear** → **All Items** to "Delete" or "Recover" all the measurement items that have been turned on.

Note: press and hold **MEAS** to quickly remove and recover all the measurement items currently enabled.

All Measurement

All measurement could measure all the time and voltage parameters (each
measurement source has 20 items, measurements can be performed on the five
measurement sources at the same time) of the current measurement source and
display the results on the screen. Press **MEAS** → **Display All** to turn the function on
or off. Press **All Measure Source** and use ↻ to select the channel to be measured
(CH1 to CH4 and MATH).

- When all measurement is turned on, "One-key" measurement is also valid.
- "**Clear**" will not clear all the measurement results.

All Measurement

One-key
Measurement

Statistic Function

Make statistic and display the current, average, minimum (or standard deviation) and maximum (or count) values of at most 5 measurement items that are turned on last.

● Press **Measure** → **Statistic** to turn the statistic function on or off. When the statistic function is enabled, press **StatisSel** to select "Extremum" or "Difference" measurement. When "Extremum" is selected, minimum and maximum values are displayed. When "Difference" is selected, standard deviation and count values are displayed.
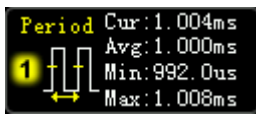
Extremum Measurement

Difference Measurement

● Press **MEAS** → **Reset Statistic** to clear the history data and make statistic again.

Measurement History

To view the history measurement data, press **MEAS** →**MeasHistory** → **MeasHistory** → "On". The history data can be displayed in two modes:

● Graph: display the results of multiple measurements of at most 5 measurement items that are turned on last in graph mode. The measurement points are connected using linear interpolation.

● Table: display the results of the last 10 measurements of at most 5 measurement items that are turned on last in table mode.

Measurement History (Graph):

Measurement History (Table):

Cursor Measurement

Cursors are the horizontal and vertical marks that can be used to measure the X axis values (usually Time) and Y axis values (usually Voltage) on a selected waveform. Please connect the signal to the oscilloscope and obtain stable display before using cursor measurement. All the "**Auto Measure**" parameters can be measured through cursor measurement.

Press the front panel function menu CURS → **Mode**, use ↻ to select the desired cursor mode (the default is OFF) and then press the knob. The modes available include Manual, Track and Auto.
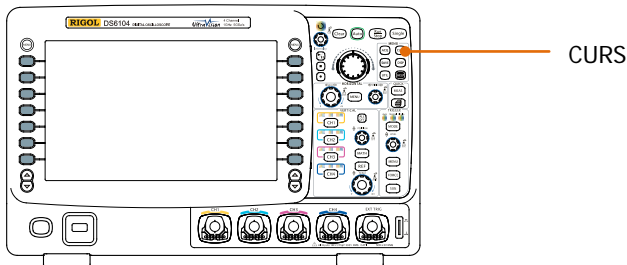
CURS

Manual Mode

In this mode, a pair of cursors will appear. You can adjust the cursors to measure the X (or Y), X increment (or Y increment), and reciprocal of the X increment on the selected source (CH1-CH4 or MATH).

Press **CURS** → **Mode** → "Manual" to turn the manual cursor function on and the measurement results will be displayed at the upper left corner of the screen in the following mode.

```
CurA = -398us
CurB = 402us
 △X = 800us
1/△X = 1.250kHz
```
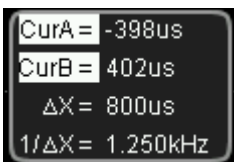
- The X or Y value at cursor A (CurA): X value takes the trigger position as reference and Y value takes the channel GND as reference.
- The X or Y value at cursor B (CurB): X value takes the trigger position as reference and Y value takes the channel GND as reference.
- The horizontal or vertical difference between cursor A and B (△X or △Y).
- The reciprocal of the horizontal difference between cursor A and B (1/△X).

If needed, please refer to the following steps to modify the parameters of manual cursor measurement.

1. **Select Cursor Type**
   Press **Type** to select "Time" or "Voltage". The time cursor is a pair of vertical dotted lines and is usually used to measure time parameters. The voltage cursor is a pair of horizontal dotted lines and is usually used to measure voltage parameters.

2. **Select Measurement Source**
   Press **Source** to select the waveform in the analog channels (CH1-CH4) or math operation results (MATH) for measurement (only channels enabled can be selected). If "None" is selected, no cursor will be displayed.

3. **Select X (Y) Axis Unit**

    When the cursor type is "Time", press **Time Unit** to select "s", "Hz", "°" or "%".

    ●   S: when this unit is selected, in the measurement results, the CurA, CurB
        and △X are in "s" and 1/△X is in "Hz".

    ●   Hz: when this unit is selected, in the measurement results, CurA, CurB and
        △X are in "Hz" and 1/△X is in "s".

    ●   °: when this unit is selected, in the measurement results, CurA, CurB and
        △X are in "°". At this point, CurA, CurB and △X will change to "0°", "360°"
        and "360°" respectively when you press **Set Cursor**, no matter where
        cursor A and B are currently located. At the same time, two cursor lines
        (unmovable) appear on the screen as the reference positions.

    ●   %: when this unit is selected, in the measurement results, CurA, CurB and
        △X are in "%". At this point, CurA, CurB and △X will change to "0%",
        "100%" and "100%" respectively when you press **Set Cursor**, no matter
        where cursor A and B are currently located. At the same time, two cursor
        lines (unmovable) appear on the screen as the reference positions.

    When the cursor type is "Voltage", press **Vertical Unit** to select "Source Unit" or
    "%".

    ●   Source Unit: when this unit is selected, in the measurement, the units of
        CurA, CurB and △Y will automatically be set to the unit of the current
        source.

    ●   %: when this unit is selected, in the measurement results, CurA, CurB and
        △Y are in "%". At this point, CurA, CurB and △Y will change to "0%",
        "100%" and "100%" when you press **Set Cursor**, no matter where cursor
        A and B are currently located. At the same time, two cursor lines
        (unmovable) appear on the screen as the reference positions.

4. **Adjust the Cursors Position (note that under the same menu page,
    you can also press ↻ continuously to switch the current cursor)**

    ●   Adjust cursor A: press **CursorA** and use ↻ to adjust the position of cursor
        A. During the adjustment, the measurement result will change accordingly.
        The adjustable range is limited within the screen.

    ●   Adjust cursor B: press **CursorB** and use ↻ to adjust the position of cursor
        B. During the adjustment, the measurement result will change accordingly.
        The adjustable range is limited within the screen.

    ●   Adjust cursor A and B at the same time: press **CursorAB** and use ↻ to

adjust the positions of cursor A and B at the same time. During the adjustment, the measurement results will change accordingly. The adjustable range is limited within the screen.

5.   Measurement Example

Use manual cursor measurement to measure the period (△X) of a square waveform and the result is 1 ms equaling the result from auto measurement.
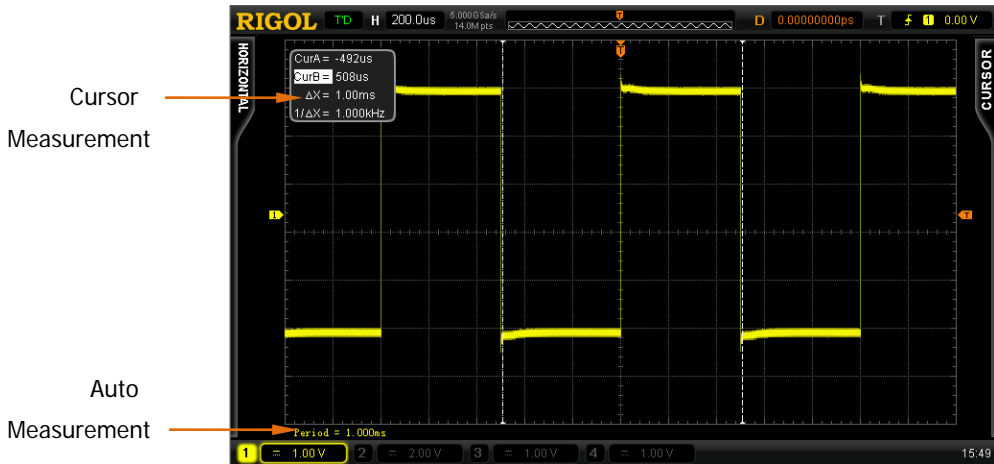
Track Mode

In this mode, one or two pairs of cursors will appear. You can adjust the two pairs of cursors to measure the X and Y values on two different sources respectively. The points being measured on cursor A and B are marked by an orange rectangle and rhombus respectively. When the cursors are moved horizontally, the marks will position on the waveform automatically. When the waveform is expanded or compressed horizontally, the marks will track the points being marked in the last adjustment of the cursors.

Press **CURS** → **Mode** → "Track" to turn on the cursor track function and the measurement results will be displayed at the upper left corner of the screen.

```
A->X = -2.88ms
A->Y = -2.40 V
B->X = 650us
B->Y = 2.60 V
 ΔX = 3.53ms
1/ΔX = 283.3 Hz
 ΔY = 5.00 V
```
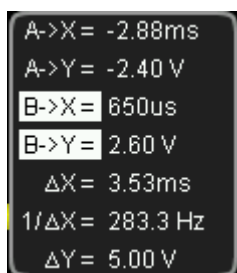
- The X value at cursor A (A->X): take the trigger position as reference and "s" or "Hz" (when measuring FFT waveform) as its unit.
- The Y value at cursor A (A->Y): take the GND as reference and use the same unit with the unit of the current source.
- The X value at cursor B (B->X): take the trigger position as reference and "s" or "Hz" (when measuring FFT waveform) as its unit.
- The Y value at cursor B (B->Y): take the GND as reference and use the same unit with the unit of the current source.
- The horizontal difference between cursor A and B ($\triangle$X).
- The reciprocal of the horizontal difference between cursor A and B (1/$\triangle$X).
- The vertical difference between cursor A and B ($\triangle$Y).

If needed, please refer to the following steps to modify track cursor measurement parameters.

1. **Select Measurement Source**

    Press **CursorA** to select the waveform in analog channels (CH1-CH4) or math operation results (MATH) as the measurement source of cursor A (only channels enabled are available). You can also select "None", namely do not use cursor A. Press **CursorB** to select the waveform in analog channels (CH1-CH4) or math operation results (MATH) as the measurement source of cursor B (only channels enabled are available). You can also select "None", namely do not use cursor B.

2. **Adjust Cursor Position (note that under the same menu page, you can also press ↻ continuously to switch the current cursor)**

    ●   Adjust cursor A: press **CursorA** and use ↻ to adjust the position of cursor A. During the adjustment, the measurement result will change accordingly. The adjustable range is limited within the screen.

    ●   Adjust cursor B: press **CursorB** and use ↻ to adjust the position of cursor B. During the adjustment, the measurement result will change accordingly. The adjustable range is limited within the screen.

    ●   Adjust cursor A and B at the same time: press **CursorAB** and use ↻ to adjust the positions of cursor A and B at the same time. During the adjustment, the measurement results will change accordingly. The adjustable range is limited within the screen.

    Note: the vertical cursor will track the marked point (namely jumps up and down with the transient moving of the waveform). Thus, the Y value might change even though you do not adjust the cursor.

3. **Measurement Example**

    Use cursor A and B to measure the waveforms in CH1 and CH2 respectively. You would discover that the cursors would track the marked points when the waveforms are expanded or compressed horizontally.

Cursor Track (Before Horizontal Expansion):
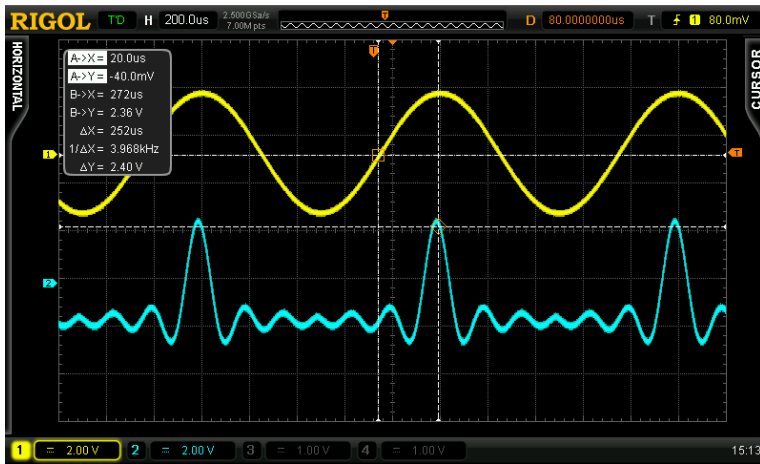
Cursor Track (After Horizontal Expansion):
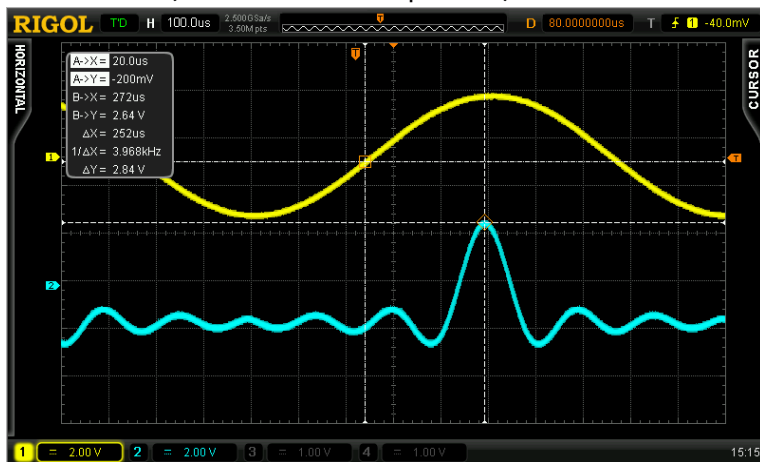
Auto Mode

In this mode, one or more cursors will appear. You can use auto cursor measurement to measure anyone of the 24 waveform parameters. Before using this mode, you need to at least turn on one auto measurement parameter and the number of cursors will change with the measurement parameters.

Press **CURS** → **Mode** → "Auto" and the cursors (the number of cursors correspond to the measurement parameter which is turned on firstly) will be displayed on the screen (different measurement parameters need different number of cursors). Note that no cursor will be displayed if no auto cursor measurement parameter is selected or there is no input of the measurement source. When the waveform is expanded or compressed horizontally, the cursor will move accordingly.

If multiple measurement parameters are turned on later, you can use **Meas.Para.** to switch among at most five measurement parameters that are turned on last and the measurement parameter currently selected is high-lighted at the bottom of the screen.

The following figure shows the example of auto measurement of the delay between two channels.
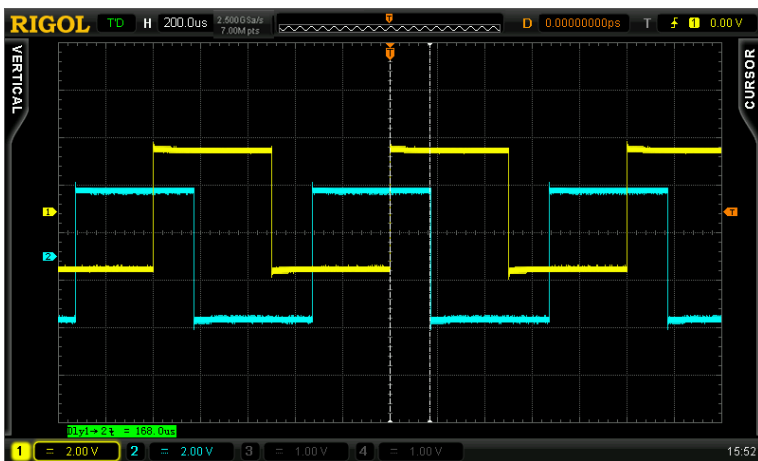
7 Protocol Decoding

Protocol analysis can be used by users to discover errors, debug hardware and accelerate development, so as to guarantee quick and high-quality accomplishment of projects. Protocol decoding is the basis of protocol analysis. Only protocol analyses with correct protocol decoding are acceptable and only correct protocol decoding can provide more error information. DS6000 provides two buses to make common protocol decoding (including: Parallel (standard), RS232 (option), SPI (option), I2C (option), CAN (option) and FlexRay (option)) of the analog channels (CH1-CH4). Because the decoding functions and setting method of the two buses are the same, this chapter only takes BUS1 for illustration.

To get the decoding option information, please refer to "**Appendix A: Options and Accessories**" on page 17-1.
When you have ordered the decoding option, please refer to "**Option Management**" on page 13-17 to activate the corresponding option.

The contents of this chapter:

■   Parallel Decoding
■   RS232 Decoding (Option)
■   I2C Decoding (Option)
■   SPI Decoding (Option)
■   CAN Decoding (Option)
■   FlexRay Decoding (Option)

Parallel Decoding

Parallel bus consists of clock line and data line. As shown in the figure below, CLK is the clock line, while Bit0 and Bit1 are the 0 bit and 1 bit on the data line respectively.
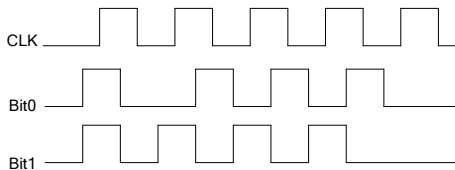
The oscilloscope will sample the channel data on the rising edge, falling edge or the rising&falling edges of the clock and judge whether each data point is logic "1" or logic "0" according to the preset threshold level.

Press **MATH** → **BUS1** → **Decode** to select "Parallel" and turn on the parallel decoding function menu.

1. **Clock Line Setting (CLK)**
    Press **CLKChannel** to select any channel (CH1-CH4) as the clock channel. If "None" is selected, no clock channel is set.
    Press **Slope** to select to sample the channel data on the rising edge (⬆), falling edge (⬇) or rising&falling edges (⨉⬛⨉). If clock channel is not selected, the instrument will sample when the channel data jumps in the decoding.

2. **Data Line Setting**
    ●   **Set the bus bits**
        Press **Bus Bits** to set the data width of the parallel bus namely the number of bits per frame. The default is 1 and the maximum is 20 bits (Bit0, Bit...Bit19).

    ●   **Specify data channel for each bit.**
        First, press **CurrentBit** to select the bit that need to specify channel. The default is 0 and the range available is always 1 smaller than the data width. For example, when the data width is 20, the range available is 0, 1...19.

Next, press **Channel** to specify a channel source for the bit currently selected in **CurrentBit**.

3. **Analog Channel Threshold Setting**
   To judge logic "1" and logic "0" of the buses, you need to set a threshold for each analog channel (CH1, CH2, CH3 and CH4). When the signal amplitude is greater than the preset value, it is considered as "1", otherwise "0".
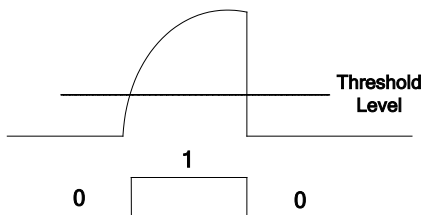
Press **Threshold** to turn the threshold setting menu on.

Channel	Select the channel that needs to set a threshold.
Set to TTL	Press this softkey to set the threshold of the specified channel to TTL level immediately.
Set to CMOS	Press this softkey to set the threshold of the specified channel to CMOS level immediately.
Set to ECL	Press this softkey to set the threshold of the specified channel to ECL level immediately.
Threshold	Press this softkey and use ↻ to set the threshold. The default is 0 V.

Note: when the threshold selected is beyond the specified range, it will be automatically limited within the range.

4. **Display-related Setting**
   Press **Format** to select the display format of the bus as Hex, Decimal, Binary or ASCII.
   Press **Offset** and use ↻ to adjust the vertical display position of the bus.
   Press **BusStatus** to turn the bus display on or off.

5. **Decoding Table**

The decoding table displays the decoded data and the corresponding line number and time in table format. It can be used to observe relatively longer decoded data to solve the problem that some data could not be seen clearly on the screen.

Press **Event Table** → **Event Table** to select "ON" (note that this operation is only available when **BusStatus** is set to "ON") to enter the decoding table interface as shown in the figure below. If a USB storage device is currently connected to the instrument, press **Export** to export the data table to the external USB storage device in CSV format.

RS232 Decoding (Option)

RS232 serial bus consists of the transmitting data line (TX) and the receiving data line (RX).
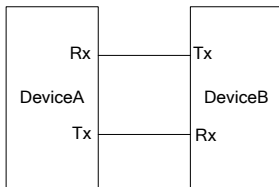
The industry standard of RS232 uses "Negative Logic", namely high level is logic "0" and low level is logic "1".
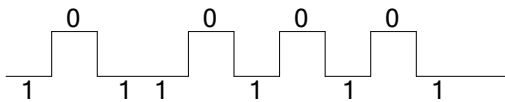
By default, RS232 uses LSB (Least Significant Bit) transmission sequence, namely the lowest bit of the data is transmitted first. While for MSB (Most Significant Bit), the highest bit of the data is transmitted first.

Endian (LSB)   Endian (MSB)
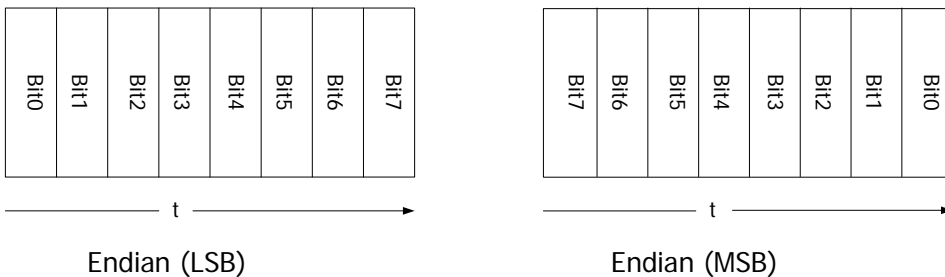
In RS232, baud rate is used to represent the transmitting rate (namely bits per second) of the data. The commonly used baud rates include 2400 bps, 4800 bps, 9600 bps, 19200 bps, 38400 bps, 57600 bps and 115200 bps.

In RS232, you need to set the start bit, the data bits, the check bit (optional) and the stop bit of each frame of data.

Start Bit	Data Bit	Check Bit	Stop Bit
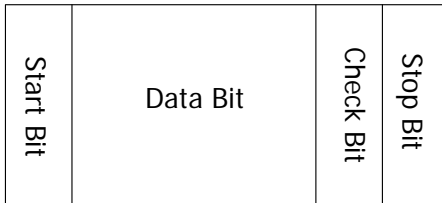
Start Bit: represent when the data starts outputting. Setting the **Polarity** is equivalent to specifying the "Start Bit".

Data Bits: represent the number of data bits actually contained in each frame of data.

Even-OddCheck: be used to check the correctness of the data transmission.

● Odd Checkout: the sum of the number of "1" in the data bit and check bit is an odd. For example, when 0x55 (01010101) is sent, a 1 needs to be filled in the check bit to make the number of 1 to be an odd.

● Even Checkout: the sum of the number of "1" in data bit and check bit is an even. For example, when 0x55 (01010101), a 0 should be filled in the check bit.

● None: if it is not set, there would not be check bit during the transmission.

Press **MATH** → **BUS1** → **Decode** to select "RS232" to turn the RS232 decoding function menu on.

1. **TX and RX Channel Setting**
   Press **TX** to select any channel (CH1-CH4) as the transmitting channel and when "OFF" is selected, no transmitting channel is set. Use the same method the set the **RX** channel. What's more, you need to set the thresholds of the input channels of **TX** and **RX**. Switch the menu page and press **TXThreshold** and **RXThreshold** respectively to input the desired threshold values.

2. **Polarity Setting**
   Press **Polarity** to select "Normal" or "Invert" and the default is invert. The oscilloscope will sample data on the rising or falling edge of the corresponding channel according to this setting.

3. **Endian Setting**
   Press **Endian** to select "LSB" or "MSB" and the default is "LSB".

4. **Baud Rate Setting**
   Press **Baud** to select the desired baud rate and the default is 9600 bps.

5. **Data Packet Setting**

    As mentioned before, in RS232, you need to set the start bit, the data bits, the check bit (optional) and the stop bit of each frame of data. "Start Bit" has been specified by the "Polarity Setting". The setting methods of other parameters are introduced below.

    ● Press **Data Bits** to set the data width of each frame of data. It can be set to 5, 6, 7, 8 or 9 and the default is 8.

    ● Press **Stop Bit** to set the stop bit after each frame of data. It can be set to 1 bit, 1.5 bits or 2 bits.

    ● Press **Even-OddCheck** to set the even-odd check mode of the data transmission. It can be set to None, Odd Checkout or Even Checkout.

    ● Press **Packet** to enable or disable the packet end of data transmmision. When packet end is enabled, several data blocks are combined according to the packet end.

    ● Press **PacketEnd** to set the packet end during the data transmission and it can be set to 00 (NULL), 0A (LF), 0D (CR), 20 (SP) or FF.

6. **Display-related Setting**

    Press **Format** to select the display format of the bus: Hex, Decimal, Binary or ASCII.

    Press **Offset** and use ↻ to adjust the vertical display position of the bus.

    Press **BusStatus** to turn the bus display on or off.

7. **Decoding Table**

    The decoding table displays the decoded data, the corresponding line number, time and error information on TX and RX data lines in table format. It can be used to observe relatively longer decoded data to solve the problem that some data could not be seen clearly on the screen. Note: when the **RX** channel is turned off, the information of this data line will not be displayed in the decoding table.

    Press **Event Table** → **Event Table** to select "ON" (note that this operation is only available when **BusStatus** is set to "ON") to enter the decoding table interface as shown in the figure below. If error occurs during the decoding, the corresponding error information is displayed. If a USB storage device is currently connected to the instrument, press **Export** to export the data table to the external USB storage device in CSV format.
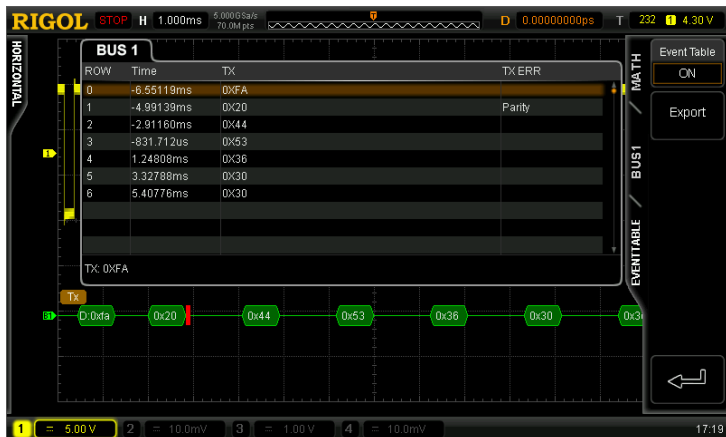
8. The Error Expression during Decoding

DS6000 makes full use of the resources such as color and view to express the results of the protocol decoding effectively and let users find the desired information quickly.

End Frame Error:

Errors generated when the end frame condition is not met. When the stop bit is set to 1.5, red error mark (note that the red mark is displayed in different modes according to the horizontal time base setting; when the horizontal time base is relatively smaller, [image] is displayed; otherwise, [image] is displayed) will be displayed if the stop bit is less than 1.5.
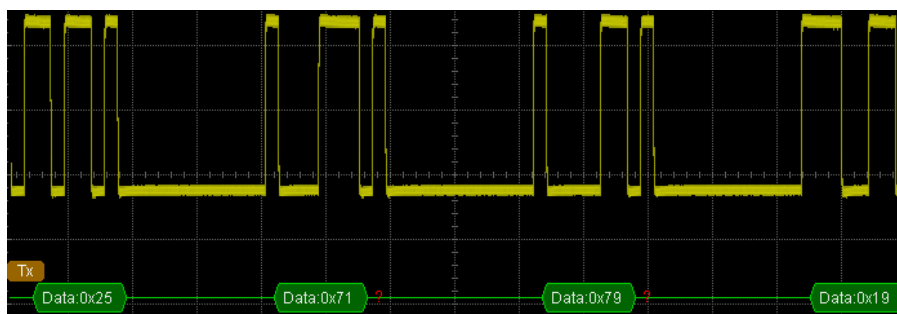
Check Error:

When check errors are detected during the decoding, the data will be filled with red batches. For example, when the transmitting terminal is set to none check and the decoder is set to odd check, the following errors appear.

(00000100, LSB)
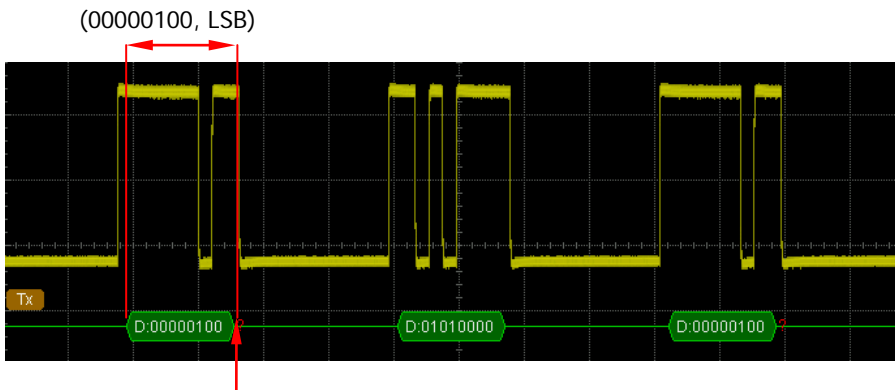
The check bit detected is 1

Wherein, there are odd number of 1 in the 8 bit data 01010010 and the parity bit should be 0. Howerer, parity error is generated as the parity bit on TX is 1.

After the decoder is set to none check, the parity bit will be 0 and the decoding becomes normal.
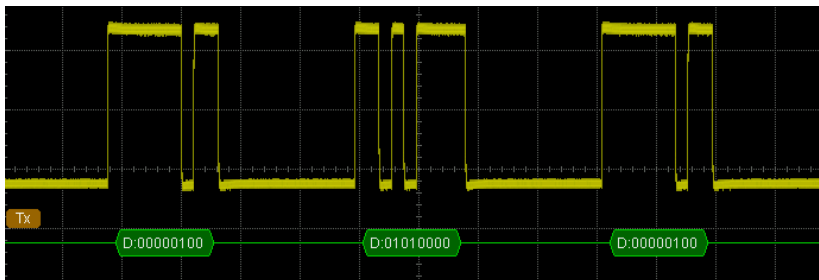
Note: two error marks will be displayed when end frame error and check error are detected at the same time.

I2C Decoding (Option)

I2C serial bus consists of the clock line (SCLK) and the data line (SDA).
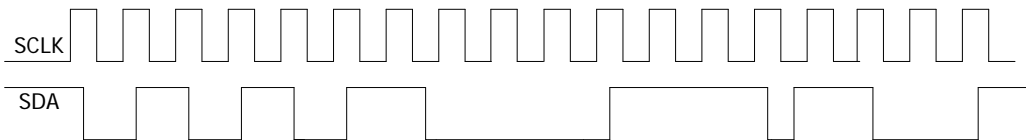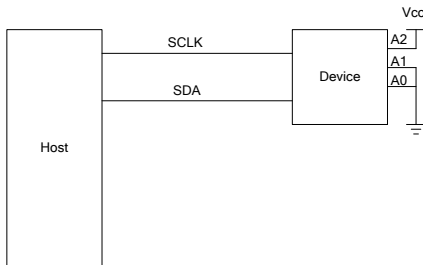
SCLK: sample the SDA on the clock rising edge or falling edge.
SDA: denote the data channel.

Press **MATH** → **BUS1** → **Decode** to select "I2C" and turn the I2C decoding function menu on.

1. **SCLK Setting**
    Press **SCLK** to select any channel (CH1-CH4) as the clock channel.
    Press **SCLKThreshold** to set the threshold of the clock channel.

2. **SDA Setting**
    Press **SDA** to select any channel (CH1-CH4) as the data channel.
    Press **SDAThreshold** to set the threshold of the data channel.

3. **Display-related Setting**
    Press **Format** to select the display format of the bus as Hex, Decimal, Binary or ASCII.
    Press **Offset** and use ↻ to adjust the vertical display position of the bus.
    Press **BusStatus** to turn the bus display on or off.

4. **Decoding Table**

The decoding table displays the decoded data, the corresponding line number, time, data direction, ID and ACK information in table format.

Press **Event Table** → **Event Table** to select "ON" (note that this operation is only available when **BusStatus** is set to "ON") to enter the decoding table interface as shown in the figure below. If a USB storage device is currently connected to the instrument, users can export the data table to the external USB storage device in CSV format.
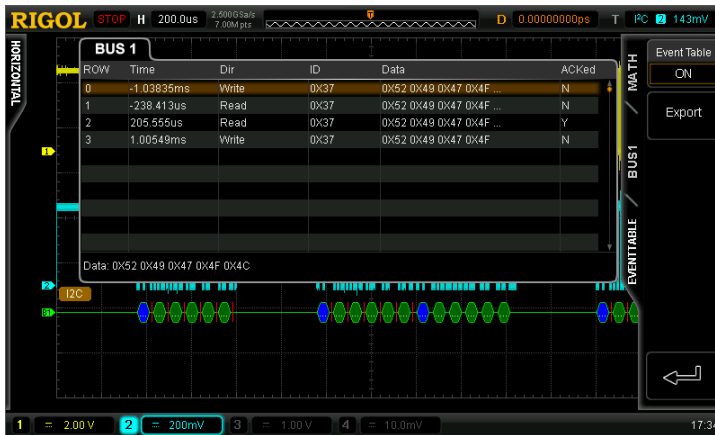
5. **Error Expressions during Decoding**

In I2C bus, the front part of each frame of data contains the address information and blue patches are used to represent address ID. In the ID, "Write" is used to represent writing address and "Read" is used to represent reading address.
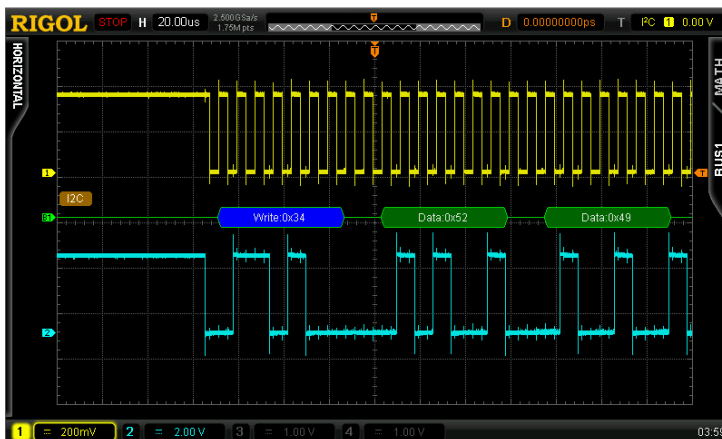
When the ACK (ACKnowledge Character) is not met, the red error marks as
shown in the figure below will be displayed.
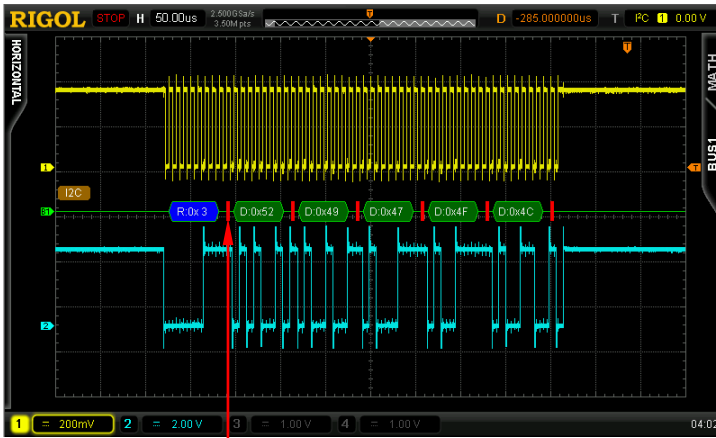
ACK=1

SPI Decoding (Option)

SPI serial bus consists of chip select line (SS), clock line (SCLK), MISO and MOSI.
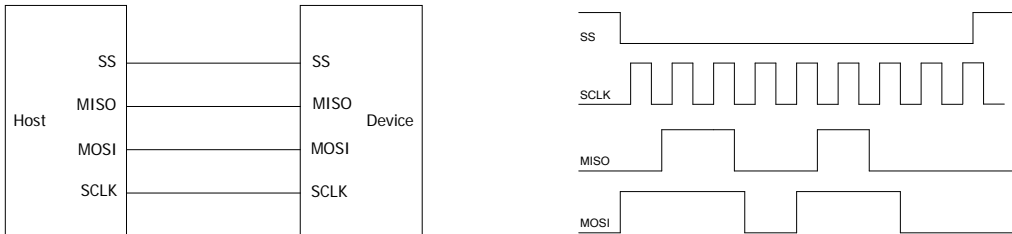
SS: set the polarity to "Low" or "High" (note that this function is not available when **Channel** is set to "OFF"). When SS selection is valid, the SPI bus samples data from MISO and MOSI at the jumping point of SCLK.

SCLK: sample data from MISO and MOSI on the clock rising edge or falling edge.

MISO: master input/slave output. Set the polarity to "Low" or "High".

MOSI: master output/slave input. Set the polarity to "Low" or "High".

Press **MATH** → **BUS1** → **Decode** to select "SPI" and turn the SPI decoding function menu on.

1. **SS Setting**

   Press **SS** to turn the chip select line setting interface on.
   - Press **Channel** to select any channel (CH1-CH4) as the chip select channel. When "OFF" is selected, the sample of MISO and MOSI occurs at the jumping point of SCLK.
   - Press **Polarity** to set the polarity of the chip select channel to "Low" or "High".
   - Press **Threshold** to set the threshold of the chip select channel.

2. **SCLK Setting**

   Press **SCLK** to turn on the clock line setting interface.
   - Press **Channel** to select any channel (CH1-CH4) as the clock channel.
   - Press **Slope** to set to sample MISO and MOSI on the rising edge or falling edge of SCLK.
   - Press **Threshold** to set the threshold of the clock channel.

3. MISO Setting

Press **MISO** to turn on the MISO data line setting interface.

- Press **Channel** to select any channel (CH1-CH4) as the MISO channel. When "OFF" is selected, this data line is not set.
- Press **Polarity** to set the polarity of the MISO data line to "Low" or "High".
- Press **Threshold** to set the threshold of the MISO data line.

4. MOSI Setting

Press **MOSI** to turn on the MOSI data line setting interface.

- Press **Channel** to select any channel (CH1-CH4) as the MOSI channel. When "OFF" is selected, this data line is not set.
- Press **Polarity** to set the polarity of the MOSI data line to "Low" or "High".
- Press **Threshold** to set the threshold of the MOSI data line.

5. Data Bits Setting

Press **Data Bits** to set the number of bits of each frame of data. The range available is from 4 to 32.

6. Endian Setting

Press **Endian** to select "LSB" or "MSB" and the default is "MSB".

7. Display-related Setting

Press **Format** to select the display format of the bus as Hex, Decimal, Binary or ASCII.

Press **Offset** and use ↻ to adjust the vertical display position of the bus.

Press **BusStatus** to turn the bus display on or off.

8. Decoding Table

The decoding table displays the decoded data, the corresponding line number, time and error information on the MOSI or MISO data line(s) in table format. It can be used to observe relatively longer decoded data to solve the problem that some data could not be seen clearly on the screen. Note: when the MISO channel is turned off, the information of this data line will not be displayed in the decoding table.

Press **Event Table** → **Event Table** to select "ON" (note that this operation is only available when **BusStatus** is set to "ON") to enter the decoding table

interface as shown in the figure below. If a USB storage device is currently connected to the instrument, users can export the data table to the external USB storage device in CSV format.
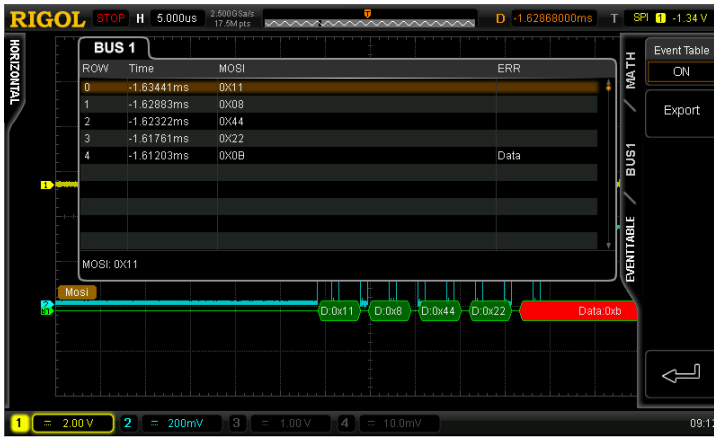
9. Error Expressions during Decoding

When the clock for a frame in SPI is not enough, the data is filled with red patches. For example, when Data Bits is set to 7, and SCLK slope is set to rising edge, deording error will be generated.
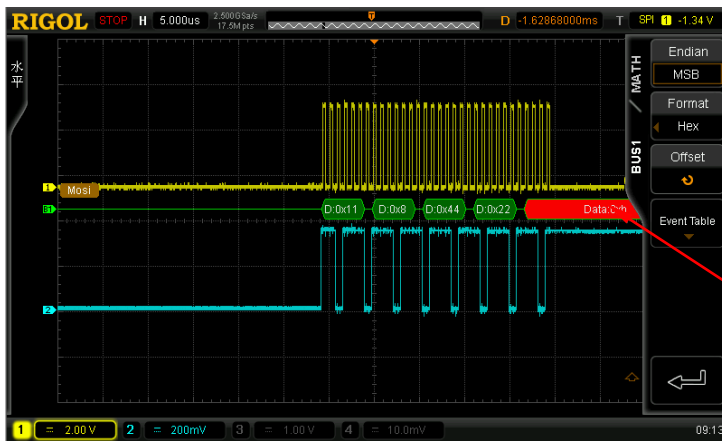
Not enough for 7 bits

CAN Decoding (Option)

Press **MATH** → **BUS1** → **Decode** and select "CAN" to open the CAN decoding function menu.

1. **Source**
   Press **Source** and select any channel (CH1-CH4) as the source channel.

2. **Signal Type**
   Press **Signal Type** to select the desired signal type.
   - **CAN_H:** the actual CAN_H bus signal.
   - **CAN_L:** the actual CAN_L bus signal.
   - **Differential:** the CAN differential bus signals connected to an analog channel using a differential probe. Connect the probe's positive lead to CAN_H and connect the negative lead to CAN_L.

3. **Baud**
   Press **Baud** to select a baud rate (100 kb/s, 125 kb/s, 250 kb/s, 400 kb/s, 500 kb/s, 800 kb/s, 1 Mb/s or User) that matches your CAN bus signal. When "User" is selected, press **Setup** and use ↻ to enter the desired rate, the range is from 10 kb/s to 1 Mb/s.

4. **Sample Point**
   The sample point is the point within the bit time. The oscilloscope samples the bit level at this point. The sample point position is represented by the percentage that the "time from the start of bit to the sample point" takes up in the "bit time".

   Press **Sample Point** and use ↻ to adjust this parameter with a step of 1%. The range is from 5% to 95%.
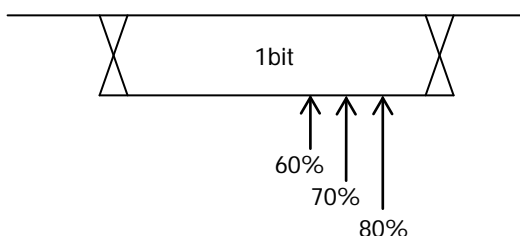
5. **Threshold**

   Refer to the introduction in **Parallel Decoding**.

6. **Display-related Setting**

   Press **Format** and set the bus display format to Hex, Decimal, Binary or ASCII.
   Press **Offset** and use ↻ to adjust the vertical display position of the bus.
   Press **BusStatus** to enable or disable bus display.

7. **Decoding Table**

   The decoding table displays the decoded data, the corresponding line number, time, frame ID, DLC, CRC and ACK information in table format.

   Press **Event Table → Event Table** to select "ON" (note that this operation is only available when **BusStatus** is set to "ON") to enter the decoding table interface as shown in the figure below. If a USB storage device is currently connected to the instrument, users can export the data table to the external USB storage device in CSV format.
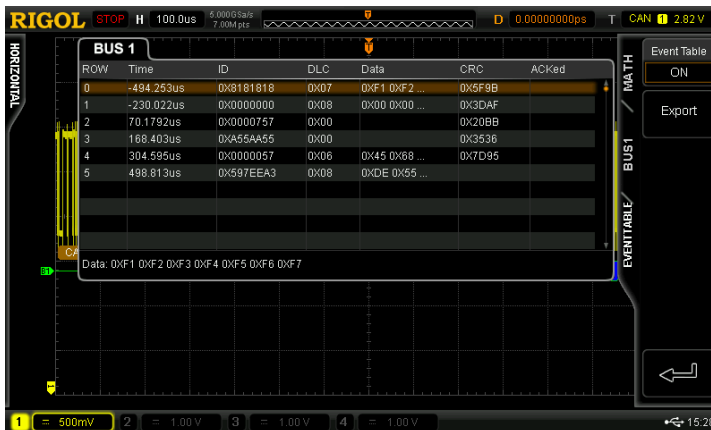
8. **Interpreting Decoded CAN Data**
   - Frame ID: displayed in hex digits in blue.
   - Data Length code (DLC): displayed in chartreuse patch.
   - Data Frame: displayed in green patches and in red patches if data frame is lost.
   - Cyclic Redundancy Check (CRC): displayed in blue-green patch when valid and red error mark is displayed when error occurs.
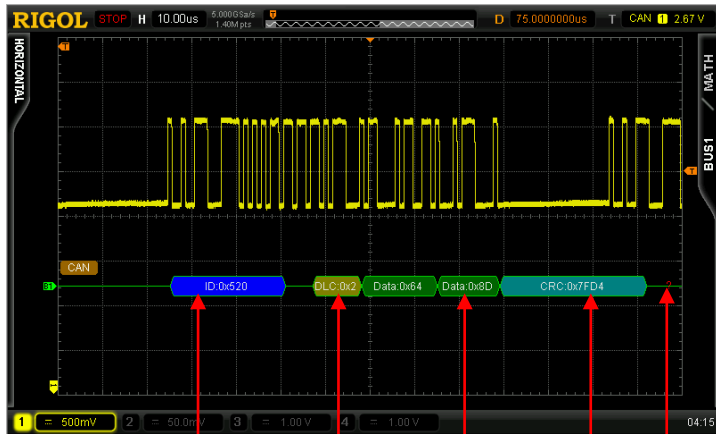
Address ID   Data Length   Data   CRC   Error

FlexRay Decoding (Option)

Press **MATH** → **BUS1** → **Decode** and select "FlexRay" to open the FlexRay decoding function menu.

1. **Source**
   Press **Source** to select any channel (CH1 to CH4) as the signal source channel.

2. **Signal Path**
   Press **Signal Path** to select the signal path (A or B) that matches the FlexRay bus signal.

3. **Signal Type**
   Press **Signal Type** to select the type of signal that matches the FlexRay bus. The signal types available include BP, BM and TX/RX.

4. **Baud**
   Press **Baud** to set the signal rate (2.5 Mb/s, 5Mb/s or 10 Mb/s) that matches the FlexRay bus signal.

5. **Sample Point**
   The sample point is the point within the bit time.The oscilloscope samples the bit level at this point. The sample point position is expressed by the percentage of the "time from the start of bit to the sample bit" takes up in "bit time".
   Press **Sample Point** and use ↻ to adjust this parameter with a step of 1% and the range is from 5% to 95%.
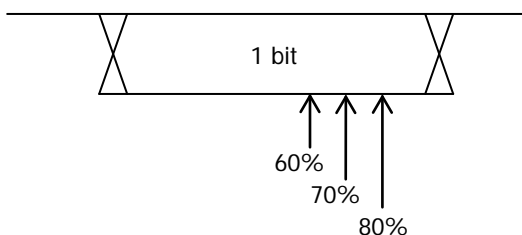
6. **Threshold**
   Refer to the introduction in **Parallel Decoding**.

7. Display-related Setting

Press **Format** to set the display format of the bus to Hex, Decimal, Binary or ASCII.

Press **Offset** and use ↻ to adjust the vertical display position of the bus.

Press **BusStatus** to enable or disable bus display.

8. Decoding Table

The decoding table lists the decoded data, the corresponding line number, time and error information in table format. It can be used to observe relatively longer decoded data to solve the problem that some data could not be seen clearly on the screen.

Press **Event Table** → **Event Table** to select "ON" (note that this operation is only available when **BusStatus** is set to "ON") to enter the decoding table interface as shown in the figure below. If a USB storage device is currently connected to the instrument, users can export the data table to the external USB storage device in CSV format.
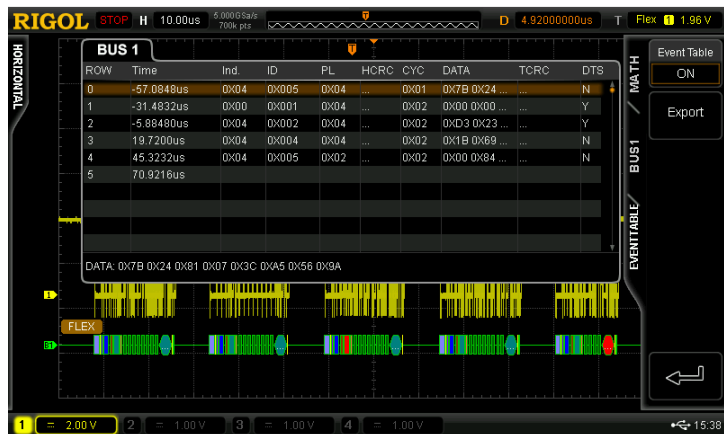
9. Explanation of the Decoded FlexRay Frame Data

The decoded FlexRay frame data is as shown in the figure below.
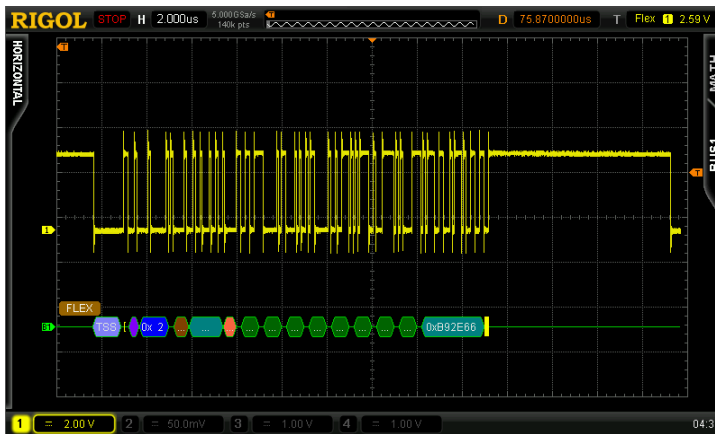
- TSS: transmission start sequence and is expressed by light purple patch.
- Frame Type: FlexRay frame can be NORMAL, SYNC, SUP or NULL. The frame type is always displayed on the screen in abbreviation format, such as "NOR" and is expressed by purple patch.
- Frame ID: decimal number and is expressed by blue patch.
- Effective Loading Length: decimal number and is expressed by brown patch.
- Head CRC: hexadecimal number and is expressed in blue-green patch. When CRC is invalid, it is expressed by red patch.
- Cycle Number: decimal number and is expressed by pink patch.
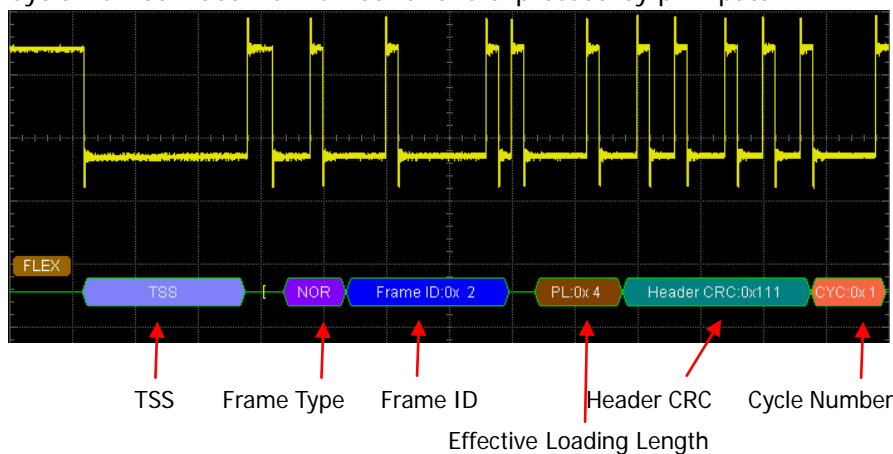
- Data: displayed in the format (Hex, Decimal, Binary or ASCII) specified in **Format** and expressed by green patch.
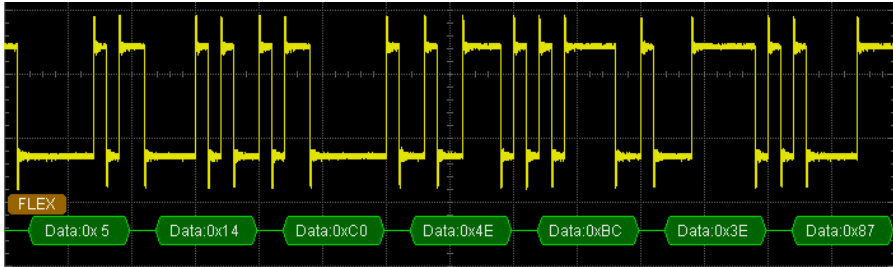
- End CRC: hexadecimal number and is expressed by blue-green patch. When CRC is invalid, it is expressed by red patch.
- DTS: dynamic end sequence and is expressed by purple patch.
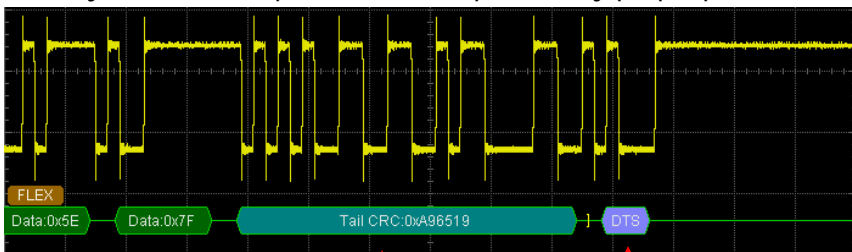
8 Reference Waveform

In actual testing process, the waveform being tested can be compared with the reference waveform module to judge the reasons for failures.

The contents of the chapter:

- To Enable REF Function
- To Set the Color
- To Select REF Source
- To Save to Internal Memory
- To Adjust REF Waveform Display
- To Export to Internal or External Memory
- To Import from Internal or External Memory

To Enable REF Function

Press **REF** in the vertical control area (VERTICAL) at the front panel to enable REF function. Note that when the time base is in X-Y mode, REF function can not be enabled.

REF
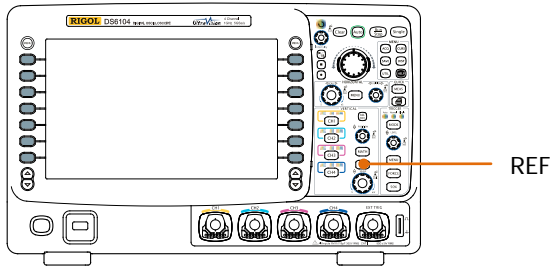
DS6000 provides 10 reference waveform channels. Press **Channel** and use ↻ to select to turn the desired reference channel on or off and a channel icon of the channel that is turned on will be display at the left side of the screen grid. For example, .
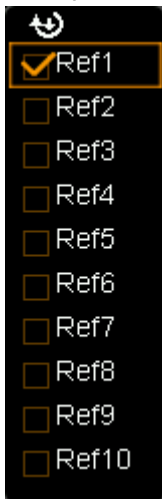
When REF function is enabled, you can select different color for each reference waveform, set the source of each reference channel, adjust the vertical scale and position of the reference waveform and save the reference waveform to internal or external memory as well as recall it when needed. For details, please refer to the introduction below.

To Set the Color

DS6000 series oscilloscope provides five colors (gray, green, light blue, magenta and orange) to mark the reference waveforms of different channels in order to distinguish them.

Press **Current** and use ↻ to select any of the reference channel (Ref1-Ref10) enabled. Then, press **Color** to specify a different color for the reference waveform of that channel. The corresponding icon at the left of the channel currently selected will be filled with the specified color, for example, R1.

To Select REF Source

Press **Current** and use ↻ to select any reference channel (Ref1-Ref10) that has been turned on and then press **Source** to specify a reference source (CH1-CH4 or MATH) for this channel. Note: only channels currently enabled can be selected.

To Save to Internal Memory

Press **Save** to save the waveform (screen region) in the specified source to internal memory as reference waveform and display it on the screen. Note that this operation only saves the reference waveform to the volatile memory and it will be lost after power-off.

To Adjust REF Waveform Display

To adjust the reference waveform in specified **Current**:

Press **REF** to set the oscilloscope to REF function state. Then, press ⬚ and use ↻ to adjust the vertical position of the reference waveform and press ⬚ and use ↻ to adjust the vertical scale of the reference waveform.

Press **Reset** and the reference waveform returns to the position where the source channel waveform is located when **Save** is pressed down.

To Export to Internal or External Memory

Users can also save the reference waveform to the Flash memory inside the instrument or to the external USB storage device. The file format of the reference waveform is "*.ref". At most 10 reference files (LocalREF0.ref to LocalREF9.ref) can be saved inside the instrument.

Press **Export** to enter file storage interface. Please refer to the relative descriptions in "**Store and Recall**" to save the reference waveform to internal or external memory.

To Import from Internal or External Memory

Users can also import the stored reference waveform in the Flash memory inside the instrument or in the external USB storage device to the internal memory.

Press **Import** to enter file recalling interface. Please refer to the relative descriptions in "**Store and Recall**" to import the reference waveform to the internal memory of the instrument.

9 Pass/Fail Test

Monitor the change of the signal by judging whether the input signal is within the mask created. The test result can be displayed on the screen as well as be prompted by the system sound or the pulse signal output from the **[Trig Out/Calibration]** connector at the rear panel.

The contents of this chapter:

■   To Enable Pass/Fail Test Function
■   To Select Source
■   Mask Range
■   Test and Output
■   To Save the Test Mask
■   To Load the Test Mask

To Enable Pass/Fail Test Function

Press **UTIL** → **Pass/Fail** → **Enable Test** to select "ON". Note: when the time base is in X-Y mode, the Pass/Fail test function can not be enabled.

UTIL
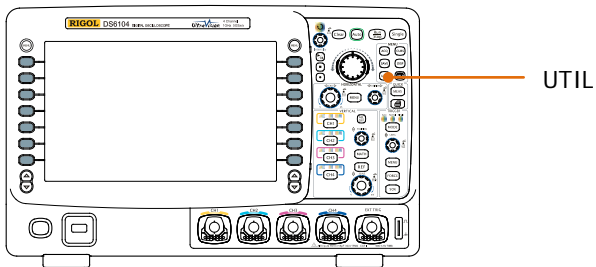
To start testing, press **Enable Test** and select "ON". Then, press **Operate**. Select "▶" to start testing and select "■" to stop testing. The figure below is the test interface.
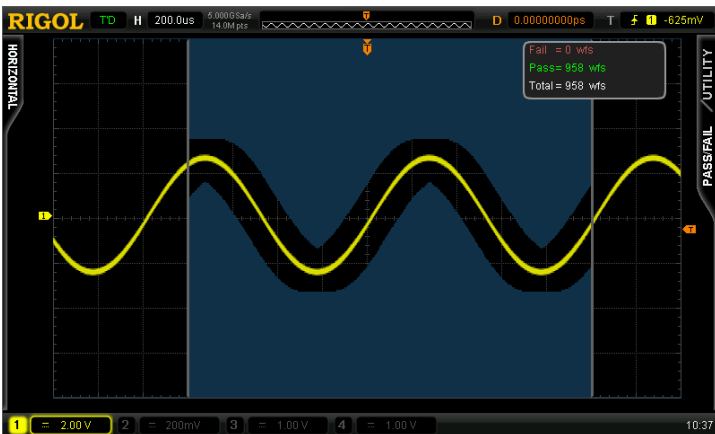
You can select the signal source, set the test mask range, create mask as well as save and load the test mask. For details, please refer to the following introduction.

To Select Source

Press **Source** to select the channel (channel enabled from CH1 to CH4) to be tested. During the test, the oscilloscope will judge whether each frame of waveform in the source complies with the current test mask and those waveforms pass through the mask area (blue area) is considered as failed.

Mask Range

Users can define their desired test masks.
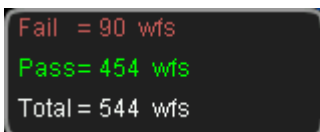
Press **MaskRange** → **MaskRange** to select "Screen Region" or "Cursor Region" for test. When "Cursor Region" is selected, two gray cursor lines appear on the screen. At this point, press **Cursor A** and **Cursor B**; then, use ↻ to adjust the positions of the two cursor lines respectively to determine the test range. Or, press **CursorAB** and use ↻ to adjust the positions of cursor A and curcor B at the same time. Note that you can press down ↻ continuously to switch the current cursor.

Press **X Mask** and **Y Mask** respectively and turn ↻, the mask lines appear on the screen. Press **Create Mask** to apply the mask currently created. The ranges of the horizontal and vertical adjustments are from 0.02 div to 4 div and 0.03 div to 4 div respectively.

Test and Output

Before the test, you can use the following method to set the output mode of the test results.
Press **Msg Display** to select "ON" or "OFF". When "ON" is selected, the test results will be displayed at the upper right corner of the screen.

Press **Stop On Outp** to select "ON" or "OFF".

- ON: when failed waveforms are detected, the oscilloscope will stop the test and enter the "STOP" state. At this point, the results of the test remain the same with the display on the screen (if display is turned on) and the **[Trig Out/Calibration]** (if enabled) connector at the rear panel only outputs one pulse.
- OFF: the oscilloscope will continue with the test even though failed waveforms are detected. The test results on the screen will update continuously and the **[Trig Out/Calibration]** connector at the rear panel will output a pulse each time a failed waveform is detected.

Press **Output** to select "Fail" or "Fail+ ".

- Fail: when failed waveforms are detected, there are display and output but the beeper does not sound.

- Fail+ : when failed waveforms are detected, there are display and output and the beeper sounds (not related to the on/off state of the sound).

Press **AuxOutput** to quickly turn "ON" or "OFF" the measurement output from the **[Trig Out/Calibration]** connector at the rear panel. You can also press UTIL → **AuxOutput** and select "PassFail" to set this output.

To Save the Test Mask

Users can save the current test mask to the Flash memory inside the instrument or external USB storage device. The file format of the test mask file is "*.pf". The internal memory can store at most 10 test mask file (LocalPF.pf).

Press **Save** to enter file saving interface. Please refer to the relative description in "**Store and Recall**" to save the test mask file to the internal or external memory.

To Load the Test Mask

Users can also load the test mask files (*.pf) stored in the Flash memory inside the instrument or the external USB storage device to the internal memory.

Press **Load** to enter the file loading interface. Please refer to the relative description in "**Store and Recall**" to load the test masks to the internal memory of the instrument.

10   Waveform Recording

Waveform recording can record the waveforms in the input channels (CH1-CH4). Waveform playback and waveform analysis can provide better waveform analysis effect.

The contents of this chapter:

■   Waveform Recording
■   Waveform Playback
■   Waveform Analysis

Waveform Recording

Waveforms from all the channels currently turned on will be recorded during waveform recording.

Navigation Knob (Playback)
UTIL
Play/Pause (Playback)
Stop Recording/Playback
Record (Record)
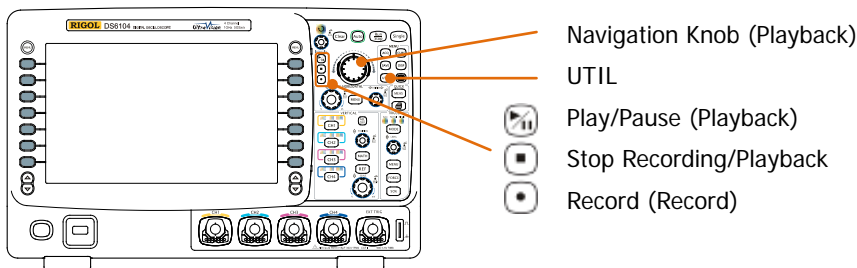
Press **UTIL** → **Record** → **Mode** and use ↻ to select "Record" to turn the waveform recording operation menu on.

1. **End Frame**
   Press **End Frame** and use ↻ to set the desired number of frames to be recorded. The number of frames available is related to the memory depth currently selected.

2. **Recording Operation**
   Waveform recoding can be realized via the shortcut buttons in the menu or at the front panel.

Menu	Front Panel
Press **Operate** and select "●" to start recoding.	Press ⊙, the backlight goes on (in red) and starts to flash, which indicates that the recording is started.
When the recording is finished, "●" changes to "■" automatically. You can also select "■" manually.	When the recoding is finished, ⊙ goes off automatically and the backlight of ⊙ goes on (in orange). You can also press ⊙ directly.

3. **Interval**

    Set the time interval between the frames in waveform recording and the range available is from 100 ns to 10 s.

4. **Maximum End Frame**

    The menu shows the maximum number of frames that can be recorded currently.

    As the capacity of the waveform memory is fixed, the more the number of points of each frame of waveform has, the less the number of waveform frames can be recorded. Thus, the maximum end frame of waveform recording is decided by the "Memory Depth". Please refer to the instruction in "**Memory Depth**" to select the desired memory depth.

    Table 10-1 Memory Depth and Maximum Number of Frames Recorded

Memory Depth	Maximum End Frame
Auto	200064
14k points	31986
140k points	1999
1.4M points	249
14M points	31
140M points	2

Waveform Playback

Waveform playback can play back the waveforms currently recorded. Press **UTIL** →
Record → **Mode** and use ↻ to select "Play back". At this point, the information as
shown in the figure below is displayed at the upper right corner of the screen,
indicating the specific frame currently displayed on the screen. During the playback,
this value would change continuously.

▶ Play = 19
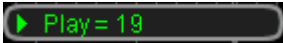
Please refer to the following descriptions to set the playback parameters.

1. **Play Mode**
   Press this softkey to set the playback mode to cycle or single.
   - ⟳: cycle playback. Play from the start frame to the end frame and then
     repeat until you stop it manually.
   - ▶━■: single playback. Play from the start frame to the end frame and
     then stop.

2. **Interval**
   Press this softkey and use ↻ to set the time interval of the playback. The range
   availbale is from 100 ns to 10 s and the default is 100 ns.

3. **Start Frame**
   Press this softkey and use ↻ to set the start frame of the playbak. The default
   is 1 and the maximum is the maximum number of frames recorded.

4. **Current Frame**
   Press this softkey and use ↻ or the Navigation knob (large scale adjustment)
   to set the current frame. By default, the current frame is the same with the end
   frame. The setting range of current frame is related to the start frame and end
   frame settings. During the setting, the screen will display the corresponding
   waveform of the current frame synchronously, namely manual playback.
   If you perform the playback operation after setting this parameter, this menu will
   be adjusted to the value corresponding to the "Start Frame" and will change
   continuously during the playback process.

5. **End Frame**

    Press this softkey and use ↻ to set the end frame of the playback. The default is the number of frames of the waveform to be played back.

6. **Playback Operation**

    Waveform playback can be realized through the shortcut buttons in the menu or at the front panel.

Menu	Front Panel
Press **Operate** and select "▶" to start playing back.	Press ⏯, the backlight goes on (in yellow), which indicates that the playback is started.
Press **Operate** and select "❚❚" to pause the playback.	Press ⏯ again to pause the playback.
Press **Operate** and select "■" to stop the playback.	Press ● to stop the playback.

Tip
In waveform playback, use **Run/Stop** to switch between play and pause. Each time **Single** is pressed, the **Current Frame** moves one frame forward.

Waveform Analysis

This function is used to analyze the recorded waveform. Press **UTIL** → **Record** → **Mode** and use ↻ to select "Analyze" to open the waveform analysis menu. At this point, the screen is divided into two display areas as shown in the figure below.

Settings and Results   Templet Display   Current Frame
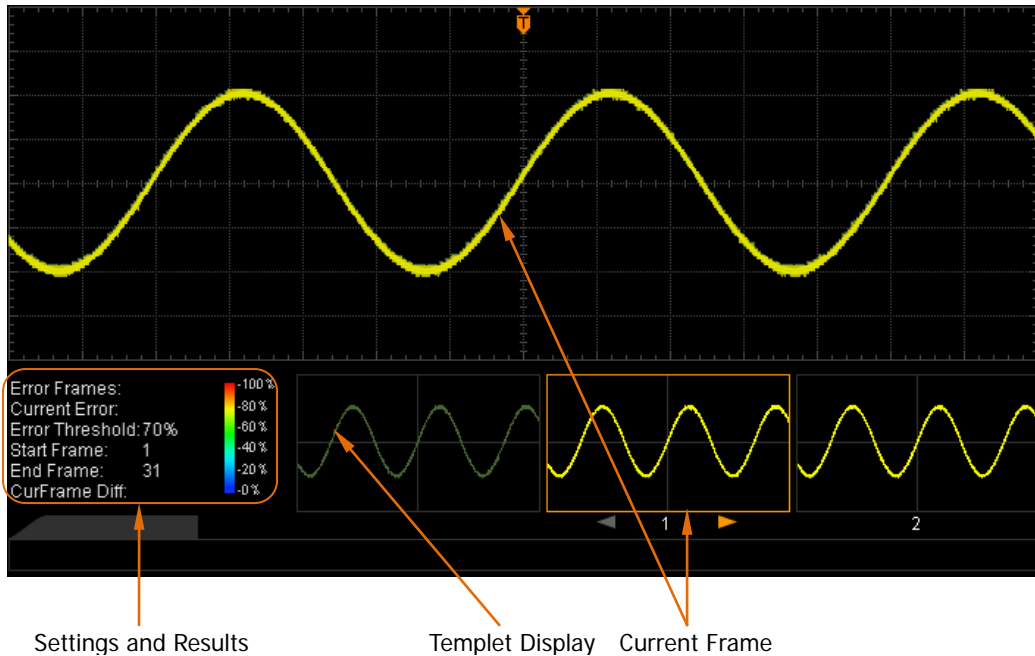
Please set the related parameters of waveform analysis by referring to the following explanations.

1. **Analysis**

    Press **Analyze** to select the desired analysis mode.
    ● Trace: Perform analysis on the basis of the templet selected by users. For detailed introduction, refer to **Analysis Based on Trace**
    ● Pass/Fail: perform analysis on the basis of the Pass/Fail mask created by users. For detailed introduction, refer to **Analysis Based on Pass/Fail Mask**.

2. **Source**

    Press **Source** to select the channel (CH1-CH4) to be analyzed. Note that only channels currently enabled can be selected.

3. **Start**

   Press **Satrt** to enable waveform analysis. Note that during the analysis process, the progress bar is displayed and the parameters can not be modified. After the analysis finishes, the analysis results of "Error Frames", "Current Error" and "CurFrame Diff" are displayed, at the same time, the first error frame is located, as shown in the figure below. At this point, using menu can locate the next error frame, the previous error frame as well as every frame in this waveform analysis.

   Error Frames: the total number of error frames discovered in this analysis. The number of error frames is related to the error frame threshold currently set.
   Current Error: the order number of the error frame lately located in all the error frames when locating every data frame in this analysis.
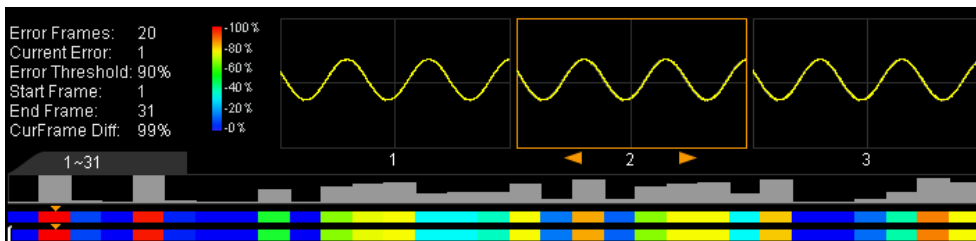   **CurFrame Diff:**
   During the analysis based on trace, the oscilloscope compares each frame of data with the templet to compute the difference value and normalizes each value using the maximum one, then, compares the normalized value with the threshold selected to judge whether the frame is an error frame. "CurFrame Diff" (relative difference) is the normalized value of the difference between current frame and templet.

   During the analysis based on pass/fail mask, the oscilloscope compares each frame with the mask to compute the difference value and recognizes the frame whose difference value is equal to or larger than the threshold selected as an error frame and the corresponding "CurFrame Diff" is 100%; otherwise, the frame is judged as correct and the "CurFrame Diff" is 0%. Note: there are only two "CurFrame Diff" values (100% and 0%) under the analysis based on pass/fail mask.

4. **Cancel**

   During the analysis, users can press **Cancel** to stop the analysis and press **Start**

again to restart the analysis.

5. **Previous**

   After the waveform analysis finishes, press **Previous** can locate the error frame before the current error frame. Pressing **Run/Stop** can also perform the operation.

6. **Next**

   After the waveform analysis finishes, press **Next** can locate the error frame following the current error frame. Pressing **Single** can also perform the operation.

7. **Current Frame**

   Press **Current Frame** to adjust the data frame currently displayed and the adjustable range is from 1 to the total number of frames in this waveform analysis.

   Note: you can use the navigation knob to view each frame of waveform of the analysis.

8. **Setup**

   Press **Setup** to open the detailed setting menu.

   ● Screen Start: press this key to set the start point of waveform analysis and the range is from 5 to 685. The start point must be lower than the "Screen End-10" currently set.

   ● Screen End: press this key to set the end point of waveform analysis and the range is from 15 to 695. The end point must be greater than the "Screen Start+10" currently set.

   ● Start Frame: press this key to set the start frame of waveform analysis and the default is the first frame.

   ● End Frame: press this key to set the end frame of waveform analysis and the default is the last frame.

   ● Threshold: press this key to set the threshold of waveform analysis and the range is from 1% to 99%. The threshold is used to judge whether the frame is an error frame. A frame can be recognize as an error one if the (relative) difference between the frame and the templet is equal to or larger than the threshold currently set.

Analysis Based on Trace

Press **Analyze** and select "Trace". Then, set the templet used in analysis based on trace through the method below.

1. **Trace**
    Press **Trace** to select the creation method of analysis templet.
    ● Current Frame: select the current frame as the analysis templet.
    ● Average: select the average of the current data frame as the analysis templet.

2. **Setup Template**
    Press **Setup Template** to configure the templet immediately. After the analysis starts, the oscilloscope compares each frame with the templet to compute the difference and determines whether error frame exists by comparing the difference with the threshold currently set.

3. **TemplateDisp**
    Press **TemplateDisp** to enable or disable template display.

Analysis Based on Pass/Fail Mask

Press **Analyze** and select "Pass/Fail". Then, set the templet used in analysis based on Pass/Fail mask through the method below.

Press **MaskRange** to open the following setting menus.

1. **Mask Range**

   Press this softkey to select "Screen Region" or "Cursor Region" for the analysis (the default is "Screen Region"). When "Cursor Region" is selected, two gray cursor lines appear on the screen. At this point, press **CursorA** and **CursorB**; then, use ↻ to adjust the positions of the two cursor lines respectively to determine the analysis range. Or, press **CursorAB** and use ↻ to adjust the positions of cursor A and curcor B at the same time. Note that you can press down ↻ continuously to switch the current cursor.

2. **X Mask**

   Press this softkey and use ↻ to adjust the horizontal threshold and the range is from 0.02 div to 4.00div. During the adjustment, two curves would be displayed to show the outline of the current mask, see the figure below.
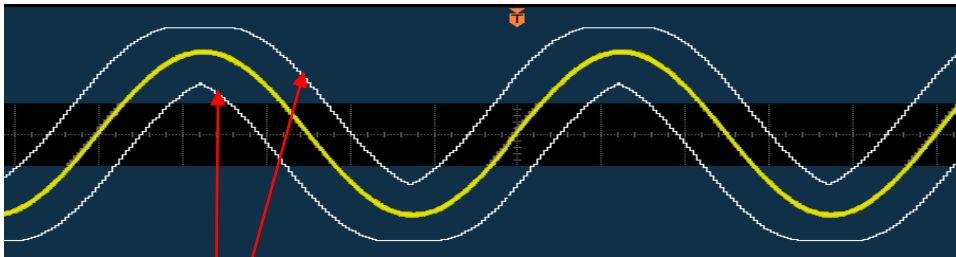
Current Mask Outline

3. **Y Mask**

   Press this softkey and use ↻ to adjust the vertical threshold and the range is from 0.03 div to 4.00 div. During the adjustment, two curves would be displayed to show the outline of the current mask.

4. **Create Mask**

   Mask is the templet used in waveform analysis. Press **Create Mask** to immediately apply the mask (X Mask and Y Mask) currently created.

Users can store the current test mask into the internal Flash memory or external USB storage device or load the test mask file (*.pf) stored in the internal Flash memory or external USB storage device into the internal memory.

Press **Save** to enter the file storage interface. Store the test mask file to internal or external memory by referring to the introduction in **Store and Recall**.

Press **Load** to enter the file load interface. Load the test mask into the internal memory of the instrument by referring to the introduction in **Store and Recall**.

11   Display Control

You can set the type, the persistence time and the brightness of waveform display as well as the grid type, grid brightness of the screen display and the menu display time.

The contents of this chapter:

- To Select the Display Type
- To Set the Persistence Time
- To set the Waveform Brightness
- To Set the Screen Grid
- To Set the Grid Brightness
- To Set the Menu Display

To Select the Display Type
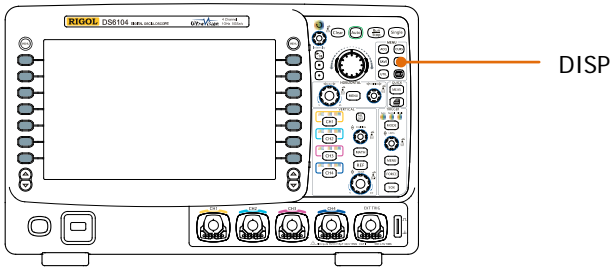
DISP

Press **DISP** → **Type** to set the waveform display mode to "Vectors" or "Dots".

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

To Set the Persistence Time

Press **DISP** → **Persis. Time** to set the persistence time of the oscilloscope to Vmin, specific values (from 50 ms to 20 s) or Infinite.

In the following part, a frequency sweep signal of the sine waveform is used to demonstrate the waveform effects in different persistence times.

1. **Vmin**
    Enable to view the waveform changing in high refresh rate.
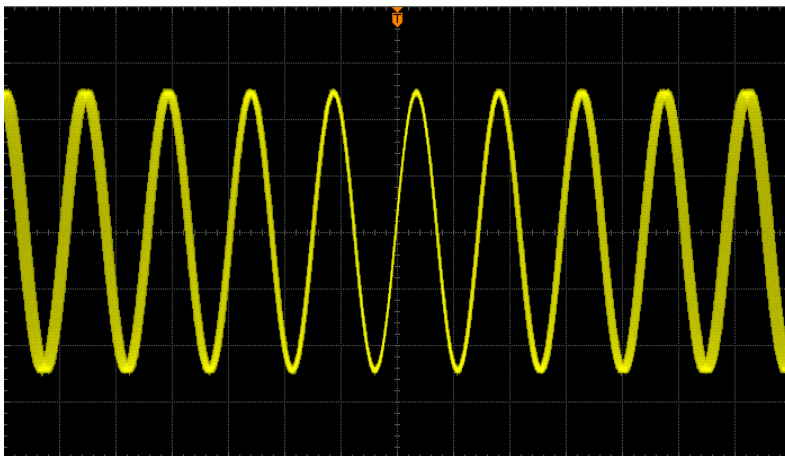
2. **Specific Values**
    Enable to observe glitch that changes relatively slowly or glitch with low occurrence probability. The persistence time can be set to 50 ms, 100 ms, 200 ms, 500 ms, 1 s, 2 s, 5 s, 10 s or 20 s.
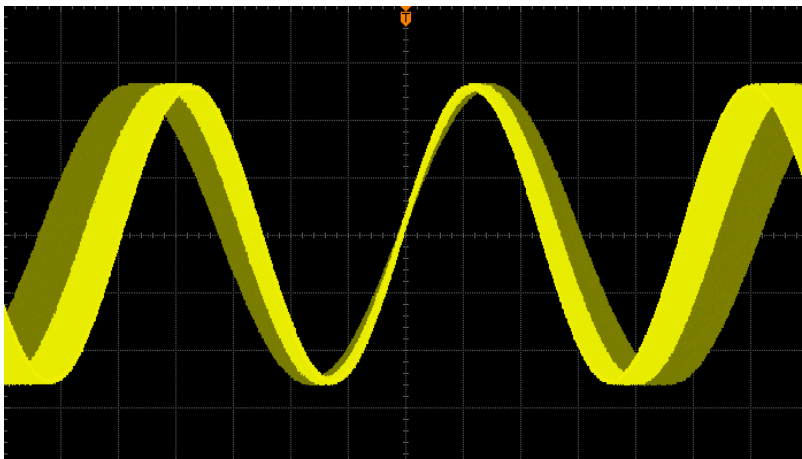
3.   Infinite

In this mode, the oscilloscope displays the waveform newly acquired without clearing the waveforms acquired formerly. The waveforms acquired formerly will be displayed in relatively low-brightness color and the waveform newly acquired will be displayed in normal brightness and color. Infinite persistence can be used to measure noise and jitter and to capture incidental events.
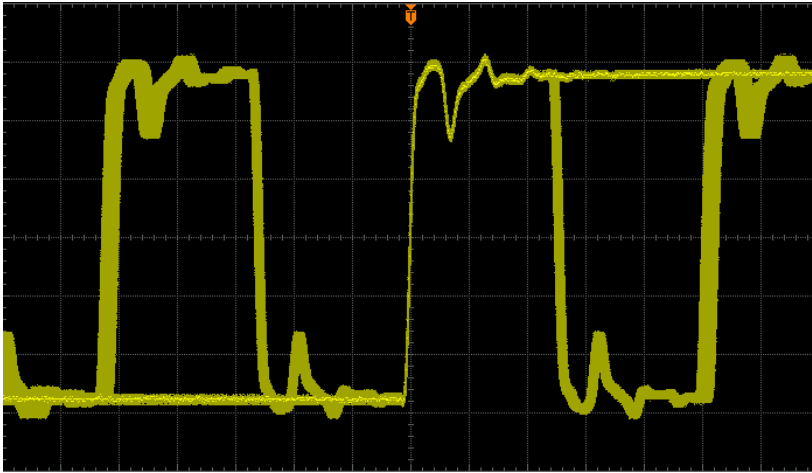
To set the Waveform Brightness

Press **DISP** → **WaveIntensity** or turn ⟳ when the menu is hidden to adjust the waveform brightness of the analog channel. The default is 50% and the range available is from 0% to 100%.

To Set the Screen Grid

Press **DISP** → **Grid** to set the screen grid type.

- ▦: turn the background grid and coordinate on.
- ⊞: turn the background grid off.
- ☐: turn the background grid and coordinate off.

To Set the Grid Brightness

Press **DISP** → **Brightness** to set the brightness of the screen grid. Turn ⟳ to adjust the grid brightness. The default is 50% and the range available is from 0% to 100%.

To Set the Menu Display

Press **DISP** → **Menu Display** to set the menu display time. The menu will hold for a specified period of time after the last button-pressing action and then hide. The display time can be set to 1 s, 2 s, 5 s, 10 s, 20 s or Infinite (the menu does not hide).

12   Store and Recall

Users can save the current settings, waveforms, and screen image in internal memory or external USB storage device (e.g. U disk) in various formats and recall the stored settings or waveforms when needed.

The contents of this chapter:

- ■   Storage System
- ■   Storage Type
- ■   Internal Store and Recall
- ■   External Store and Recall
- ■   Disk Management
- ■   Factory

Storage System

Press **SAVE** to enter the store and recall setting interface.

SAVE
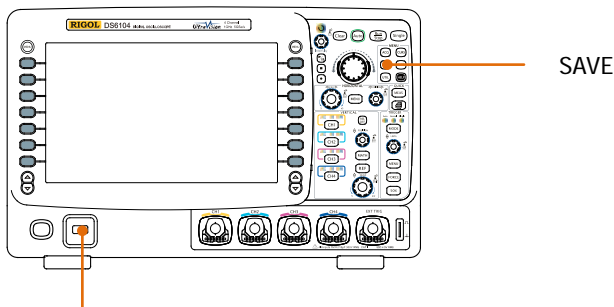
Insert the USB storage device

This oscilloscope provides two USB Host interfaces at the front and rear panels to connect USB storage device for external storage. The USB storage device connected is marked with "Disk E" (front panel) and "Disk D" (rear panel). The internal memory (Local Disk) of the instrument can store 10 trace files, 10 waveform files and 10 setting files as well as 10 reference waveform files and 10 test mask files of the Pass/Fail test. Below is the disk selecting interface.

Name	Size	Time
Up...		
Local Disk	1.00G	
Disk D	124.6M	
Disk E	124.6M	

Table 12-1 Probable Icons

Icon	Description	Icon	Description
	Return to the previous disk management interface		Waveform File
	Local Disk Memory		JPEG File
	External USB Storage Device		Pass/Fail Mask File
	Folder		PNG File
	Return to the previous folder		Waveform Recording File
	Unknown File		Reference Waveform File
	Bitmap File		Setting File
	CSV File		Trace File

Storage Type

Press **SAVE** → **Storage** to select the desired storage type. The default is "Traces". The store and recall descriptions of each type are as follows.

1. **Traces**
    Save the waveform data in internal or external memory in "*.trc" format. At most 10 trace files (from LocalTrace0.trc to LocalTrace9.trc) can be stored in the internal memory. The data of all the channels turned on can be saved in the same file. At recall, the data will be displayed on the screen directly.

2. **Waveforms**
    Save the waveform data in internal or external memory in "*.wfm" format. At most 10 waveform files (from LocalWfm0.wfm to LocalWfm9.wfm) can be stored in internal memory. The stored files contain the waveform data of the four analog channels and the main setting information of the oscilloscope and all the data can be recalled.

3. **Setups**
    Save the settings of the oscilloscope in internal or external memory in "*.stp" format. At most 10 setting files (from LocalSetup0.stp to LocalSetup9.stp) can be stored in internal memory. The stored settings can be recalled.

4. **Picture**
    Save the screen image in external memory in "*.bmp", "*.png", "*.jpeg" or "tiff" format. You can specify the file name and saving directory and save the corresponding parameter file (*.txt) under the same directory using the same file name. The recall of image and reference files is not supported.
    After seleting this type:
    Press **PicType** to select the desired storage format.
    Press **Para.Save** to enable or disable the parameter save function.

One-key Bitmap Saving

After a USB storage device is connected, press the print key ⌷ at the front panel to quickly save the current screen image under the root directory of the USB storage device in bitmap mode.

---

5. **CSV**

Save the waveform data displayed on the screen or of the specified channel in external memory in a single "*.csv" file. You can specify the file name and the saving directory and save the corresponding parameter file (*.txt) under the same directory using the same file name. The recall of CVS and parameter files is not supported.

After selecting this type:

Press **DataDepth** to select "Displayed" or "Maximum". After selecting "Maximum", press **Channel** to select the desired channel (note that only channels currently enabled can be selected).

Press **Para.Save** to enable or disable the parameter save function.

Internal Store and Recall

Internal store and recall support "Traces", "Waveforms" and "Setups" in **Storage**. In the following part, "Traces" are taken as an example for illustration.

1. Save the specified type of file in internal memory.
    1) Connect the signal to the oscilloscope and obtain stable display.
    2) Press **SAVE** → **Storage** to select "Traces" and press **Save** to turn on the interface as shown in Figure a. use ↻ to select " Local Disk" and press ↻ to turn on the local disk (Figure b).

 Figure a

    3) As shown in Figure b, the local disk can at most store 10 trace files. Use ↻ to select the desired storage position, **Save** is illuminated and then press the key to execute the saving operation. If the current position contains a file, the original file can be overwritten or be delected by pressing **Delete**. Use

    ↻ to select 🔼 Up... and then press down ↻ to return to the previous

    directory.

 Figure b

    Note: in internal storage, **New File** and **New Folder** are not available.

2. Load the specified type of file in the internal memory.

1) Press **SAVE** → **Storage** to select "Traces" and then press **Load** to turn on the interface as shown in Figure c. Use ↻ to select "Local Disk" and then press down ↻ to open the local disk (Figure d).

Figure c

2) As shown in Figure d, use ↻ to select the desired file to load and press **Load** to load the file selected.

Figure d

External Store and Recall

Before using external store and recall, make sure that a USB storage device is connected correctly. External storage supports all the types of files in **Storage** but in recall, "Picture" and "CSV" are not supported.

1. **Save the specified type of file in the external USB storage device.**
   1) Connect the signal to the oscilloscope and obtain stable display.
   2) Press **SAVE** → **Storage** to select "CSV" and press **Save** to turn on the interface as shown in Figure e. Use ↻ to select "Disk E" and press down ↻ to open the USB storage device (Figure f).

    Figure e

   3) As shown in Figure f, use ↻ to select the desired storage position. Enable to store under the root directory or in a certain folder under the root directory of the USB storage device.

    Figure f

   4) After the storage position is selected, press **New File** to open the interface as shown in Figure g. Refer to the descriptions in "**To Create a New File or Folder**" to create a new file name.

Figure g

5) Press **OK** to execute the saving operation.

2. Load the specified type of file in the external USB storage device.

1) Press **SAVE** → **Storage** to select "Traces" and then press **Load** to turn on the interface as shown in Figure h. Use ↺ to select "Disk E" and press down ↺ to open the USB storage device (Figure i).

Figure h

2) As shown in Figure i, use ↺ to select the desired file to load and then press **Load** to load the selected file.

Figure i

Disk Management

Press SAVE → **Disk.Manage** to turn on the disk management interface as shown in the figure below and use ↻ to select the desired disk. The disk currently selected is displayed in green and press down ↻ to open the selecte disk.

Name	Size	Time
📁 Up...		
🗂 Local Disk	1.00G	
⟷ Disk D	124.6M	
⟷ Disk E	124.6M	

Execute the following operations through the disk management menu:
- To Select File Type
- To Create a New File or Folder
- To Delete a File or Folder
- To Rename a File or Folder
- To Clear the Local Memory

To Select File Type

Except the file types in **Storage**, the oscilloscope can also display, save or read some files for advanced applications such as mask file of the Pass/Fail test (*.pf), waveform recording file (*.rec), update file (.rgl), parameter file (*.txt) and reference waveform file (*.ref).

Press **SAVE** → **Disk.Manage** → **File Type** to select the desired file type. The default is "*.*". Under the current directory, only files of which the suffixes of the file name match with the file type selected will be displayed in the current disk.

To Create a New File or Folder

This operation is only valid in external storage. Before using external store and recall, make sure that a USB storage device is connected correctly.

First, press **SAVE** → **Disk Manage** and use ↻ to select and open the external disk ("Disk D" or "Disk E").

Second, select the desired directory under which to create a new file or folder. The default is the root directory of the USB storage device.

Then, press **New File** or **New Folder** to turn on the interface as shown in the figure below.

This oscilloscope supports Chinese/English input method. The file name or folder name contains letters, numbers, underscores, spaces and Chinese characters and the length of the characters is limited to 64 bytes. The following part introduces how to input a file name or folder name using Chinese/English input method.

Operation Tip

During the name input, use the menu softkeys to select different operation areas, then turn ↻ to select the desired content and press down ↻ to input the content selected.

English Input Method

For example, create a file or folder using the name "Filename".

1. Press **Keyboard**.
   1) Use ↻ to select English input method "En" and uppercase input state "ᵃA".
   2) Use ↻ to input the letter "F". If the input is wrong, press **Delete** to delete the character input.
   3) Use ↻ to select lowercase input state "Aa".
   4) Use ↻ to input the remaining letters "ilename".

Name Input Area   Keyboard   Upper-lower Case Switch   Input Method Switch
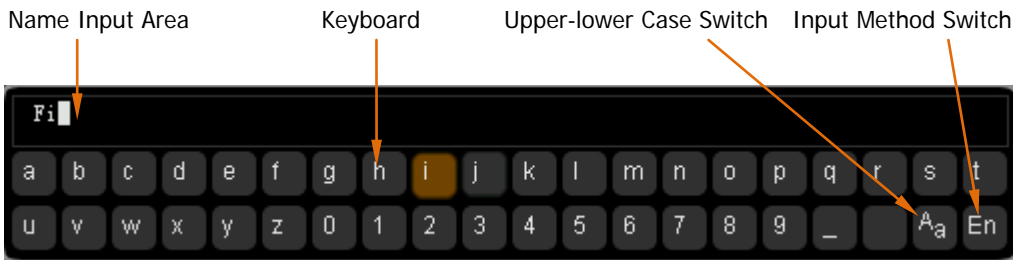
2. During the name input, you can press **Name** to select the "Name Input Area" and use ↻ to move the cursor, then press **Delete** to delete the characters on the left of the cursor one by one.
3. After finishing the input, press **OK** and the oscilloscope will create a folder or a specified type of file with this file name under the current directory.

Chinese Input Method

For example, create a file or folder using the name "文件名".

1. Press **Keyboard**.
   1) Use ↻ to select Chinese input method "中" and lowercase input state "Aa". Note that the "Chinese" menu is added in the menu items at the right of the screen.
   2) Use ↻ to input the pinyin "wen". If the input is wrong, press **Delete** to delete the pinyin input. After "wen" is input, a series of Chinese characters appear in the "Chinese Chraracter Selecting Area".
   3) Press **Chinese** and use ↻ to select and input "文".
   4) Use the same method to input "件" and "名".

Pinyin Input Area   Chinese Character Selecting Area   Input Method Switch

     Name Input Area   Keyboard   Upper-lower Case Switch

2. During name input, you can press **Name** to select the "Name Input Area" and then press **Delete** to delete the characters on the left of the cursor one by one.
3. After finishing the input, press **OK** and the oscilloscope will create a folder or a specified type of file with this file name under the current directory.

To Delete a File or Folder

Folder operation is valid only in external storage. Before using the external disk, make sure that a USB storage device is connected correctly.

1. **Delete a file in internal memory.**
    1) Press **SAVE** → **Disk.Manage** and use ↻ to select and open the local disk ("local Disk").
    2) Press **Type** to select the desired file type to delete (note that the file types of internal storage include "*.wfm, *.stp", "*.ref" and "*.pf").
    3) Use ↻ to select the desired file to delete.
    4) Press **Delete** to delete the file selected.

2. **Delete a file or folder in external memory.**
    Press **SAVE** → **Disk.Manage** and use ↻ to select and open the external disk ("Disk D" or "Disk E"). Use ↻ to select the file (or folder) to be deleted and then press **Delete** to delete the selected file (or folder).

To Rename a File or Folder

Rename operation is valid only in external storage. Before using the external disk, make sure that a USB storage device is connected correctly.

Press **SAVE** → **Disk.Manage** and use ↻ to select and open the external disk ("Disk D" or "Disk E"). Use ↻ to select the desired file or folder to rename and then press **Rename** to open the rename interface. For specific operations, please refer to the descriptions in "**To Create a New File or Folder**".

To Clear the Local Memory

You can clear all the files stored in the local memory with one operation.

Press **SAVE** → **Disk.Manage** and select "Local Disk", then turn to the next menu page and press **SecurityClear** → **OK** to delete all the files stored in the local memory. At the same time, the instrument will be resotred to the default setting.

Factory

Press **SAVE** → **Factory** to return the oscilloscope to its factory state. Refer to the table below.

Table 12-2 Factory

Parameter	Factory
Horizontal Setting (HORIZONTAL)	
Channel Setting (VERTICAL)	
Acquisition Setting (ACQ)	
Trigger Setting (TRIGGER)	
Display Setting (DISP)	
Cursor Setting (CURS)	
Storage Setting (SAVE)	
Utility Function Setting (UTIL)	
Math Operation Setting (MATH→Operation)	
Protocol Decoding (MATH→BUS1/BUS2)	
Reference Waveform Setting (REF)	
Horizontal Setting (HORIZONTAL)	
Horizontal Time Base	1 µs
Horizontal Offset	0 s
Delayed Sweep	OFF
Time Base Type	Y-T
Time Base Scale	Coarse
Horizontal Reference	The Center of the Screen
Channel Setting (VERTICAL)	
Vertical Scale	1 V
Vertical Offset	0 V
CH1 Switch	ON
CH2 Switch	OFF
CH3 Switch	OFF
CH4 Switch	OFF
Channel Coupling	DC
Bandwidth	OFF
Input Impedance	1 MΩ

Channel Invert	OFF
Vertical Scale	Coarse
Channel Unit	V
Acquisition Setting (ACQ)	
Acquisition Mode	Normal
Sampling Mode	Real Time
Memory Depth	Auto
Anti-Aliasing	OFF
Trigger Setting (TRIGGER)	
Trigger Type	Edge
Source	CH1
Slope	Rising Edge
Trigger Mode	Auto
Trigger Coupling	DC
Trigger Holdoff	100 ns
Display Setting (DISP)	
Display Type	Vectors
Persistence Time	Min
Waveform Intensity	50%
Screen Grid	▦▦
Brightness	50%
Menu Display	Infinite
Cursor Setting (CURS)	
Mode	OFF
Manual	
Cursor Type	Time
Source	CH1
Time Unit	s
CurA	-4*1 µs
CurB	4*1 µs
Track	
Cursor A	CH1
Cursor B	CH1
CurA	-4*1 µs
CurB	4*1 µs
Storage Setting (SAVE)	

Storage Type	Traces
Utility Function Setting (UTIL)	
I/O Setting	
Network Configuration Mode	DHCP, Auto IP
USB Device	Computer
Sound	
Sound	OFF
Pass/Fail Test	
Enable Test	OFF
Source	CH1
Operate	OFF
X Mask	0.24div
Y Mask	0.38div
Message Display	OFF
Stop On Output	OFF
Output	Fail
Waveform Recording	
Mode	OFF
Record	
End Frame	3998
Operate	OFF
Interval	100 ns
Max End Frame	3998
Playback	
Operate	OFF
Play Mode	Single
Interval	100 ns
Start Frame	1
Analyze	
Analysis Mode	Trace
Source	CH1
Trace Mode	Current Frame
Current Frame	1
Templet Display	ON
System Setting	

Vertical Expansion	Ground
Screen Saver	Default
Screen Saver Time	OFF
External Trigger Impedance	1 MΩ
Aux Output	TrigOut
Reference Clock	ClockOutput
Math Operation Setting (MATH→Operation)	
Operate	OFF
A+B	
Source A	CH1
Source B	CH1
Invert	OFF
Vertical Scale	2 V
A-B	
Source A	CH1
Source B	CH1
Invert	OFF
Vertical Scale	2 V
A*B	
Source A	CH1
Source B	CH1
Invert	OFF
Vertical Scale	2 U
A/B	
Source A	CH1
Source B	CH1
Invert	OFF
Vertical Scale	2 U
FFT	
Source	CH1
Window Function	Rectangle
Display	Split
Scale	dB
Vertical Scale	20.0 dB
Horizontal Scale	2.5 MHz/div

Sample Rate	100 MSa/s
Logic Operation	
Expression	AND
Source A	CH1
Source B	CH1
Invert	OFF
Vertical Scale	1 U
Threshold A	0 V
Threshold B	0 V
Advanced Operation	
Expression	OFF
Expression	CH1+CH2
Vertical Scale	2 V
Protocol Decoding (MATH→BUS1/BUS2)	
Decoding Type	Parallel
Bus Status	OFF
Format	Hex
Offset	0
Threshold	0
Parallel	
Clock Channel	None
Slope	Rising Edge
Bus Bits	1
Current Bit	0
Channel	CH1
RS232	
TX	CH1
RX	CH2
Polarity	Invert
Endian	LSB
Baud	9600 bps
Data Bits	8
Stop Bit	1
Even-Odd Check	None
Packet	OFF
Packet End	00 (NULL)

I2C	
SCLK	CH1
SDA	CH2
SPI	
SS Channel	OFF
SS Polarity	Low
SCLK Channel	CH1
SCLK Slope	Rising Edge
MISO Channel	CH2
MISO Channel	Low
MOSI Channel	OFF
MOSI Polarity	Low
Data Bits	8
Endian	MSB
Reference Waveform Setting (REF)	
Channel Setting	REF1
Current Channel	REF1
Source	CH1
Color	Gray

13   System Function Setting

The contents of this chapter:

- Remote Interface Configuration
- System-related
- To Print the Waveform

Remote Interface Configuration

DS6000 can communicate with PC via LAN, USB and GPIB (together with the USB to GPIB interface copnverter of **RIGOL**) buses. Please refer to the introduction below to configure the corresponding interface before using the remote interfaces.

LAN Setting

Press **UTIL** → **IO Setting** → **LAN Set** to open the LAN setting interface. You can view the network connection status and configure the network parameters.
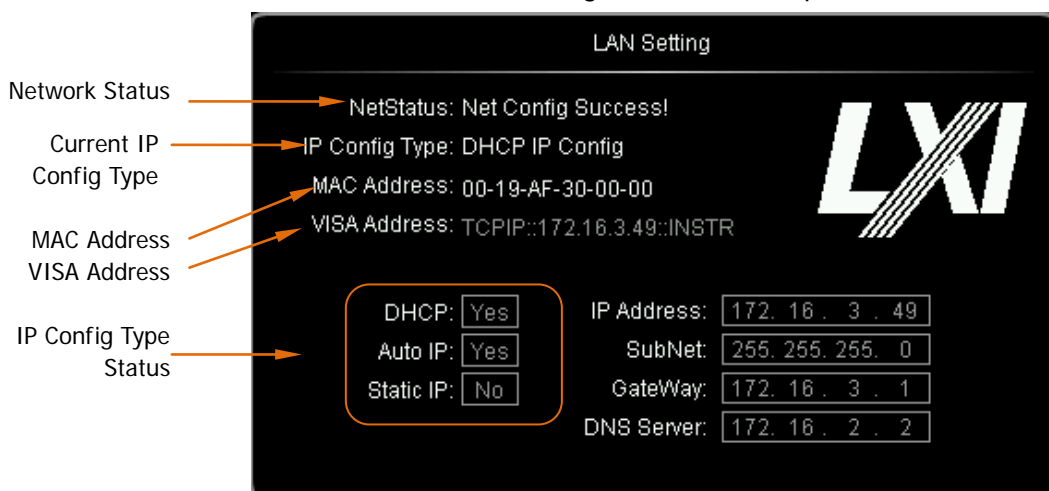
Network Status

Connect the oscilloscope to your local area network using the network cable. The network hole of the oscilloscope is at the rear panel. The oscilloscope will give different prompts according to the current network connection status.

- Net Config Success!
- Acquire IP…
- IP Conflict!
- Unconnected!
- DHCP Fail!
- Read Status Fail!

IP Configuration Type (DHCP)

The configuration type of the IP address can be DHCP, auto IP or static IP. In different IP configuration type, the configuration mode of the network parameters such as the IP address is different.

Press **Config Mode** and use ↻ to select "DHCP". Then press down ↻ to select this type. When DHPC type is valid, the DHCP server will assign the network parameters such as the IP address for the oscilloscope.
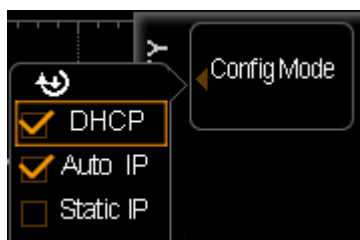

IP Configuration Type (Auto IP)

Press **Config Mode** and use ↻ to select "Auto IP". Then press down ↻ to select this type. When the auto IP type is valid, disable DHCP manually, **Gate** and **DNS** are added to the right side of the screen and users can define the gateway and DNS server address of the oscilloscope. In auto IP mode, the oscilloscope will get the IP address ranging from 169.254.0.1 to 169.254.255.254 and the subnet mask 255.255.0.0 automatically according to the current network configuration.

IP Configuration Type (Static IP)

Press **Config Mode** and use ↻ to select "Static IP". Then press down ↻ to select this type. When this type is valid, disable DHCP and auto IP manually, **IP Address**, **Mask**, **Gate** and **DNS** are added to the right of the screen. At this point, users can define their own network parameters (such as the IP address) of the oscilloscope.

1. **Set the IP Address**
   The format of the IP address is nnn.nnn.nnn.nnn, where, the range of the first

nnn is from 0 to 223 (except 127) and the ranges of the other three nnn are from 0 to 255. You are recommended to ask your network administrator for an IP address available.

Press **IP Address** and use ↻ to input the desired IP address. This setting will be saved in the non-volatile memory and if **Power On** is set to "Last", **DHCP** and **Auto IP** will be "Off" and the oscilloscope will load the preset IP address automatically at the next power-on.

2. **Set the Subnet Mask.**
    The format of the subnet mask is nnn.nnn.nnn.nnn, where, the range of the nnn is from 0 to 255. You are recommended to ask your network administrator for a subnet mask available.

    Press **Mask** and use ↻ to input the desired subnet mask. This setting will be saved in the non-volatile memory and if **Power On** is set to "Last", **DHCP** and **Auto IP** will be "Off" and the oscilloscope will load the preset subnet mask automatically at the next power-on.

Set the Gate

You can set this paramter In **Auto IP** and **Static IP** mode.
The format of the gate is nnn.nnn.nnn.nnn, where, the range of the first nnn is from 0 to 223 (except 127) and the ranges of the other three nnn are from 0 to 255. You are recommended to ask your network administrator for a gate address available.

Press **Gate** and use ↻ to input the desired gate address. This setting will be saved in the non-volatile memory and if **Power On** is set to "Last", **DHCP** and **Auto IP** will be "Off" and the oscilloscope will load the preset gate address automatically at the next power-on.

Set the Domain Name Server

You can set this paramter In **Auto IP** and **Static IP** mode.
The address format of the domain name server is nnn.nnn.nnn.nnn, where, the

range of the first nnn is from 0 to 223 (except 127) and the ranges of the other three nnn are from 0 to 255. You are recommended to ask your network administrator for an address available.

press **DNS** and use ↻ to input the desired address. Generally, users do not need to set the DNS, therefore this parameter setting can be ignored.

Tips

- When the three IP configuration types are all turned on, the priority of the parameter configuration from high to low is "DHCP", "Auto IP" and "Static IP".
- The three IP configuration types can not be all turned off at the same time.

Apply the Network Parameter Setting

Press **Apply** to validate the current network parameter setting.

Initialize the network Parameters

Press **Initialize** to return the network parameters to the default state.

MAC Address

For each oscilloscope, the MAC address is unique. When attributing the IP address for the oscilloscope, the MAC address is usually used to identify the instrument.

VISA Address

Display the VISA address currently used by the oscilloscope.

USB Device

This oscilloscope can communicate with a PC or PictBridge printer via the **USB Device** interface at the rear panel. You need to set the oscilloscope to make it match with different device types.

Press **UTIL** → **IO Setting** → **USB** and use ↻ to select the desired device type.

- Computer: under this type, the oscilloscope can communicate with a PC.
- PictBridge: under this type, the oscilloscope can communicate with a PictBridge printer.

To Set the GPIB Address

When using the GPIB method to control the oscilloscope, you need to use the USB-GPIB interface converter (need to order separately) to extend a GPIB interface for the oscilloscope.

To set the GPIB address, press **UTIL** → **IO Setting** → **GPIB** and use ↻ to input the desired address. The default is 1 and the range available is from 1 to 30.

System-related

Sound

When the sound is enabled, you can hear the sound of the beeper when you press a function key or a menu softkey or when the prompt message pops up.

Press **UTIL** → **Sound** to select ◀◁ (on) or ◀◁× (off). The default is off. When the sound is turned on, a trumpet icon ◣ will be displayed at the lower right corner of the screen.

Language

This oscilloscope supports multiple language menus, Chinese/English help and prompt message.

Press **UTIL** → **Language** and use ↻ to select the desired language. Then press down ↻ to select the language. The languages currently available are Simplified Chinese, Traditional Chinese and English.

System Information

Press **UTIL** → **System** → **System Info** to view the version information of your oscilloscope. The system information contains the following contents as shown in the figure below.

Power-off Recall

You can set the system configuration to be recalled when the oscilloscope is powered on again after power-off.

Press **UTIL** → **System** → **Power On** to select "Last" or "Default".
● Last: restore the instrument to the setting at the last power-off.
● Default: return to the factory setting of the system.

System Time

The system time is displayed at the lower right corner of the screen in "hh:mm (hour:minute)" format. When printing or storing a waveform, the file output will contain this time information.

Press UTIL → **System** → **System Time** → **System Time** to turn on the setting interface as follows.
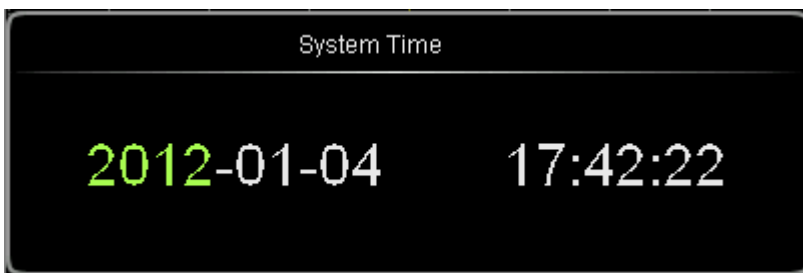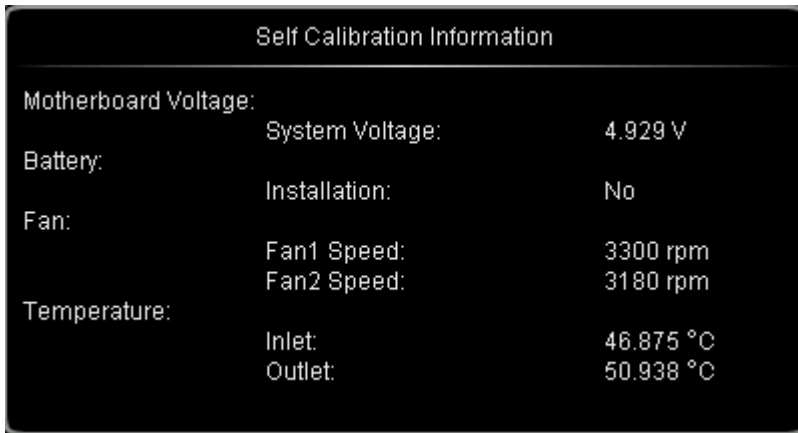
In the figure, the item in green (e.g. 2012) is the item that can be modified currently. Use ↻ to modify and press down ↻ to complete the input. The order of time modifications is year (2012) → month (01) → date (04) → hour (17) → minute (42)→ second (22). The setting range of each item conforms to the convention.

- Year: 1999 to 2099
- Month: 01-12
- Date: 01-31 (28, 29 or 30)
- Hour: 00 to 23
- Minute: 00 to 59
- Second: 00 to 59

Press **Apply** to put the current setting into effect. The time at the lower right corner of the screen will be updated.

Self-calibration Information

Press **UTIL** → **System** → **SelfCalInfo** to view the result of the last self-calibration of the oscilloscope. The self-calibration result usually contains the contents as shown in the figure below.

```
                    Self Calibration Information

Motherboard Voltage:
                    System Voltage:          4.929 V
Battery:
                    Installation:            No
Fan:
                    Fan1 Speed:              3300 rpm
                    Fan2 Speed:              3180 rpm
Temperature:
                    Inlet:                   46.875 °C
                    Outlet:                  50.938 °C
```

Note: if the oscilloscope is installed with battery, the self-calibration information will also display the electric quantity, voltage and current of the battery.

Screen

When the oscilloscope enters the idle state and holds for a certain period of time, the screen saver program will be enabled.

Press **UTIL** → **System** → **Screen** to turn on the screen saver setting menu. "Default" means using the icon **RIGOL** as the screen saver icon.

Press **Time** to select the time of the screen saver. When "OFF" is selected, the screen saver program is disabled. The time of the screen saver can be set to 1 min, 2 min, 5 min, 15 min, 30 min, 45 min, 60 min, 2 hour or 5 hour.

Error Information

If this menu is shown in gray, it means that the circuit board of the oscilloscope is working normally. Otherwise, an exclamation mark will be displayed in the status bar at the lower right corner of the screen, indicating that there is error information currently and at this point, press this key (press **UTIL** → **System** → **ErrorInfo**) to view.

The error information that might appear including: the temperature of the mainboard is too high, the fan stops rotating etc. If the above-mentioned information appears, users need to return the instruments back to our factory for repair. At most 16 error messages lately appeared can be viewed. The error information is stored in volatile memory and will be cleared automatically after power-off.

Self-calibration

Self-calibration program can quickly help the oscilloscope to reach the best working state and get the most precise measurement values. You can perform self-calibration at any time especially when the change range of the environment temperature is up to or more than 5 ℃. Make sure that the oscilloscope has been warmed up or operated for more than 30 minutes.

The self-calibration uses the fast edge signal output from the **[Trig Out/Calibration]** connector at the rear panel as the calibration signal. Please connect this signal to the four input channels and the external trigger input channel, and then press **UTIL** → **Self-Cal** and the self-calibration interface as shown in the figure below will be displayed on the screen.
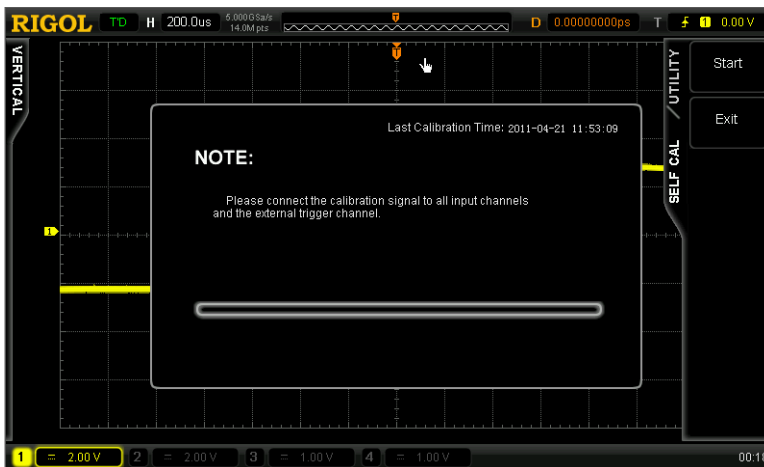
Press **Start** and the oscilloscope will start to execute the self-calibration program. Press **Exit** to give up the self-calibration operation at any time and return to the previous menu.

Note: most of the keys are disabled during the self-calibration.

Power Status

Users can set the power status of the oscilloscope after power-on. Connect the oscilloscope to the AC source using the power cord and turn on the power switch at the rear panel, the oscilloscope is powered on.

Press **UTIL** → **PowerStatus** to select "Default" or "Open".

- Default: after the oscilloscope is powered on, you need to press the power key at the front panel to start up the oscilloscope.
- Open: after the oscilloscope is powered on, it starts up directly. Note: at this point, you can still press the power key at the front panel to turn off the oscilloscope.

External Trigger Impedance

Users can set the input impedance of the **[EXT TRIG]** connector at the front panel.

Press **UTIL** → **Ext Trigger** to select "1 MΩ" (default) or "50 Ω".

Aux Output

Users can set the signal type output from the **[Trig Out/Calibration]** connector at the rear panel.

Press **UTIL** → **AuxOutput** to select the desired output type.

1. **TrigOut**
    After this type is selected, the oscilloscope output a signal that can reflect the current capture rate of the oscilloscope at each trigger.

2. **Fast**
    After this type is selected, the oscilloscope can output a fast edge signal which can be used in the self-calibration of the oscilloscope.

3. **GND**
    After this type is selected, the oscilloscope can output a ground level.

4. **PassFail**
    After this type is selected, the oscilloscope will output a pulse signal when failed waveforms are detected. Connect this signal to other control systems to conveniently view the test results.

Reference Clock

This oscilloscope can output the internal 10 MHz sample clock from the **[10MHz In/Out]** connector at the rear panel and accept an external 10 MHz clock to synchronize multiple oscilloscopes.
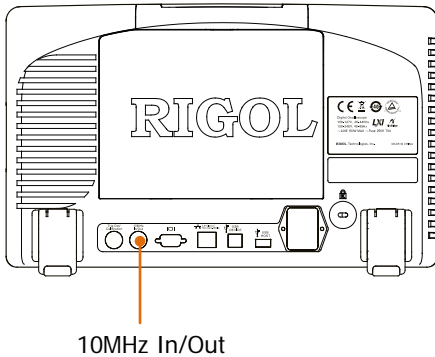
10MHz In/Out

Press **UTIL** → **RefClock** to select the desired clock type.

● ClockOutput: configure the **[10MHz In/Out]** connector as an output and output the internal 10 MHz clock.
● ClockInput: configure the **[10MHz In/Out]** connector as an input and the oscilloscope will accept an external 10 MHz clock input from this connector.

Option Management

This oscilloscope provides multiple options to fulfill your measurement requirements. Please contact your **RIGOL** sales representative or **RIGOL** technical support to order the corresponding options. You can view the options currently installed on the oscilloscope or activate the newly bought option serial number through the menu.

Press **UTIL** → **Options** → **Installed** to view the options currently installed in the oscilloscope and the related information. Press **Setup** to enter the serial number activation operation menu.

● Editor: press this key to open the serial number input interface as shown in the figure below. Use ↻ to select the characters on the virtual keyboard and press down the knob to input the character.

Serial Number Input Area   Character Selecting Area
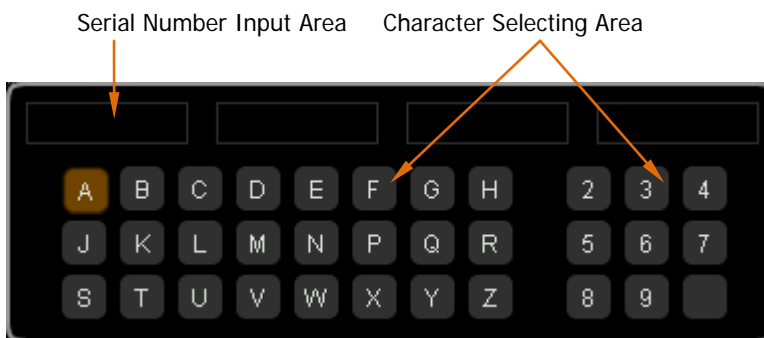
● Backspace: press this softkey to delete the characters in the "Serial Number Input Area" from the right to the left.
● Clear: press this softkey to clear all the characters in the "Serial Number Input Area".
● Apply: press this softkey and the oscilloscope will activate the corresponding option using the serial number currently input.

To Print the Waveform

To Connect the Print Device

DS6000 oscilloscope supports normal and PictBridge printers and you can set the print parameters to print the current screen image. The connection modes of different print devices are different. Use the USB Host interface to connect normal printer and use the USB Device interface to connect the PictBridge printer.
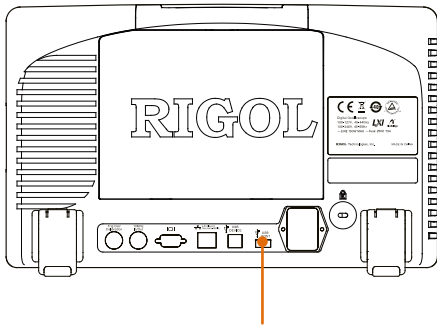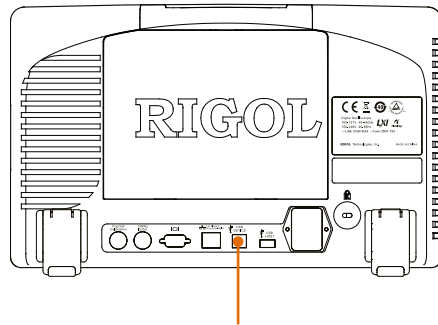
To Connect Normal Printer   To Connect PictBridge Printer

After the printer is connected, you need to set the USB device type and the print mode of the oscilloscope.

- Press **UTIL** → **IO Setting** → **USB Device** to select "PictBridge" (when PictBridge printer is connected).
- Press **UTIL** → **Print Set** → **Print Mode** to set the print mode to "Normal" or "PictBridge" according to the printer currently connected.

Normal Print

After "Normal" is selected, please refer to the following steps to set the print parameters to execute print.

1. **Print Range**
    Press this softkey to select "Screen" (default) or "Wave".
    - Screen: print the whole screen image.
    - Wave: only print the waveform and the basic information.

2. **Palette**
    Press this softkey to select "Grayscale" (default) or "Color".

3. **Invert**
    Press this softkey to turn the invert print on or off.

4. **Preview**
    Press this softkey to open the preview interface.

5. **Header**
    Press this key to set the header in print.

6. **Footer**
    Press this softkey to set the footer in print.

7. **Print**
    After setting the print parameters, press this softkey or press the shortcut key at the front panel directly to execute the print operation.

PictBridge Print

PictBridge is a new print standard. If your
oscilloscope and the printer both comply with the
PictBridge standard, you can connect the
oscilloscope to the printer using a USB cable to print
the screen image directly.
Devices complies with the PictBridge standard
always have the icon as shown in the figure on the
right.

After selecting the "PictBridge" print mode, please refer to the following steps to set
the print parameters and execute print.

1. **Status**

   A prompting frame will appear to show the current print status after you press
   this softkey.

2. **Print Range**

   Press this softkey to select "Screen" (default) or "Wave".
   - Screen: print the whole screen image.
   - Wave: only print the waveform and the basic information.

3. **Palette**

   Press this softkey to select "Grayscale" or "Color" (default).

4. **Preview**

   Press this softkey to turn on the preview interface.

5. **Size**

   Press this softkey to select the size of the paper of the print. You can select
   default, A2, A3, A4, A5, A6 or B5.

6. **Type**

   Press this softkey to select the picture type of the print. You can select default,
   Jpeg or Bmp.
   Note: the selection is unavailable when the printer does not support this
   function.

7. Quality
Press this softkey to select the print quality. You can select Default, Normal, Draft or Fine.
Note: the selection is unavailable when the printer does not support this function.

8. Date
Press this softkey to turn the date print on or off.
Note: this attribute is the built-in attribute of the PictBridge printer. This menu is invalid when the printer connected does not support this attribute.

9. Invert
Press this softkey to turn the invert print on or off.

10. Copies
Press this softkey and use ↻ to set the copies to be printed. The range is from 1 to 999.

11. Header
Press this softkey to set the header in print.

12. Footer
Press this softkey to set the footer in print.

13. Print
After setting the print parameters, press this softkey or press the ⊟ shortcut key at the front panel directly to execute the print operation.

14. Terminate
As the PictBridge printer is the master device, you need to terminate the print via the oscilloscope by pressing **Terminate**.
Note: this menu is only displayed during the print.

14 Remote Control

The oscilloscope can be controlled remotely mainly using the following two methods:

User-defined programming
Users can program and control the oscilloscope by using the SCPI (Standard Commands for Programmable Instruments) commands. For more information about the commands and programming, refer to the Programming Guide.

Use PC software provided by RIGOL or other manufacturers
Users can use the PC software **Ultra Sigma** (from **RIGOL**), **Measurement & Automation Explorer** (from National Instruments Corporation) or **Agilent IO Libraries Suite** (from Agilent Technologies, Inc.) to send commands to control the oscilloscope remotely.

This oscilloscope can communicate with PC through USB, LAN and GPIB (together with the USB to GPIB interface converter of **RIGOL**) instrument buses. This chapter will give a detailed introduction of how to use **Ultra Sigma** to control DS6000 remotely through various interfaces. As for how to get the **Ultra Sigma** software and detailed operation instruction, please contact **RIGOL**.

The contents of this chapter:

- Remote Control via USB
- Remote Control via LAN
- Remote Control via GPIB

Remote Control via USB

1. Connect the device

Connect the oscilloscope (USB DEVICE) and PC using a USB cable.

2. Install the USB driver

This oscilloscope is a USBTMC device. Assuming that your PC has already been installed with Ultra Sigma, after you connect the oscilloscope to the PC and turn both on for the first time (the oscilloscope will be automatically set to USB interface), the PC will display a New Hardware Wizard as shown in the figure below. Please install the "USB Test and Measurement Device" driver following the directions in the wizard. The steps are as follows.
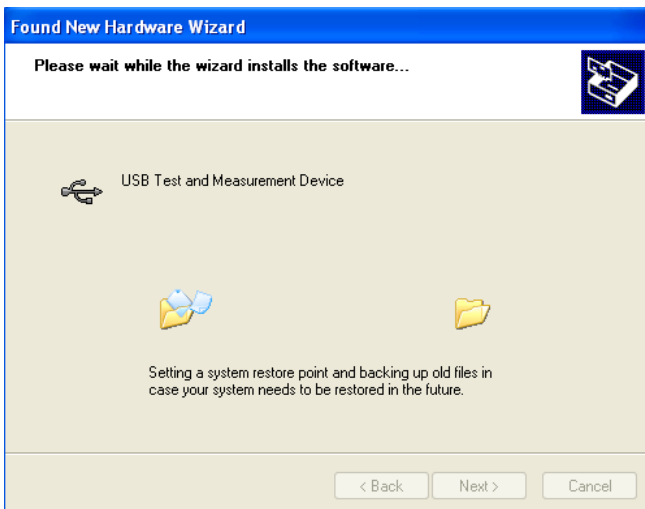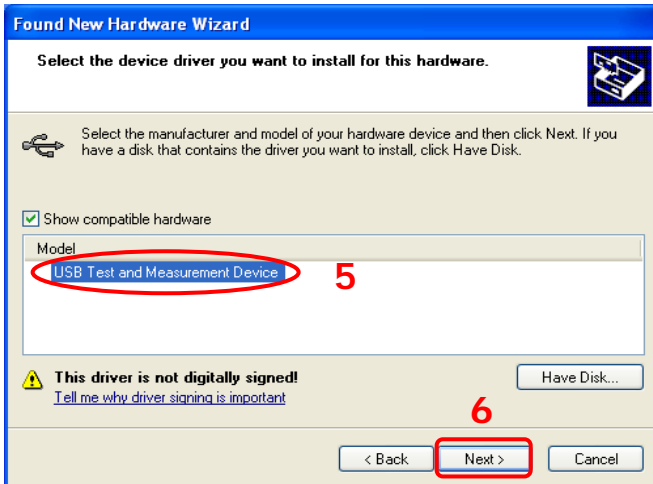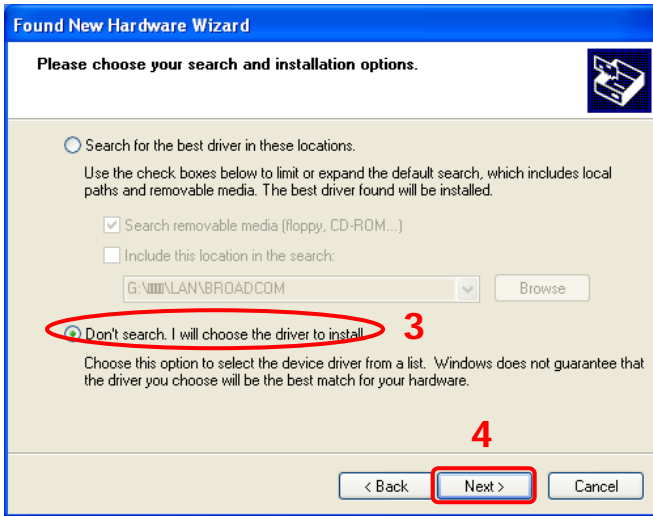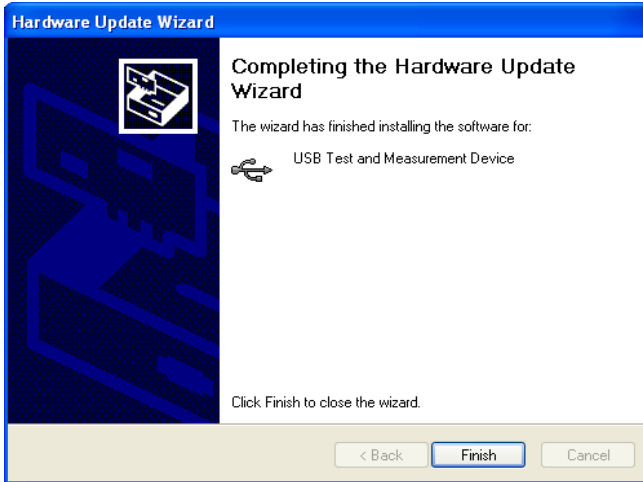
3. **Search device resource**

   Start up the Ultra Sigma and the software will automatically search for the oscilloscope resources currently connected to the PC. You can also click

   USB-TMC to search the resources. During the search, the status bar of the

   software is as shown in the figure below.

4. **View the device resource**

   The resources found will appear under the "RIGOL Online Resource" directory and the model number and USB interface information of the instrument will also be displayed as shown in the figure below.
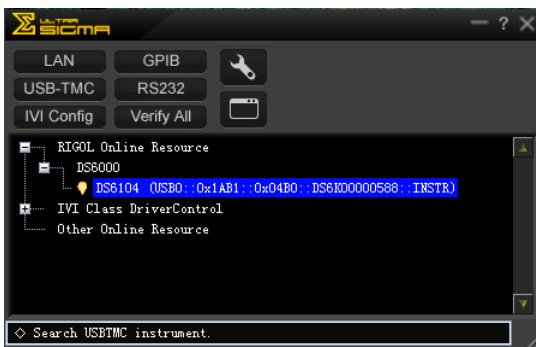
5. Communication test

Right click the resource name

"DS6104 (USB0::0x1AB1::0x04B0::DS6K00000588::INSTR)" to select "SCPI

Panel Control" to turn on the remote command control panel from which you can

send commands and read data as shown in the figure below.

Remote Control via LAN

1. **Connect the device**
   Connect the oscilloscope to your LAN using a network cable.

2. **Configure network parameters**
   Configure the network parameters of the oscilloscope according to the description in "LAN Setting".

3. **Search device resource**

   Start up the **Ultra Sigma** and click LAN . The window as shown in the

   figure below is displayed. Click Search (as shown in Figure (a)) and the

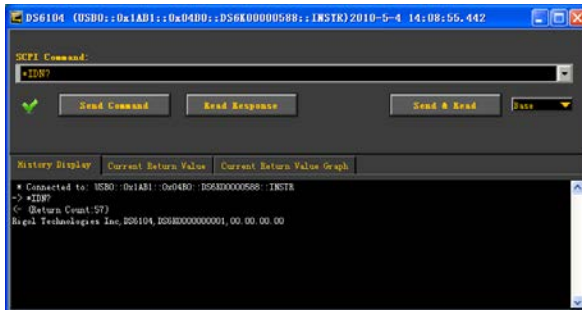   software searches for the oscilloscope resources currently connected to the LAN
   and the resources found are displayed at the right of the window. As shown in
   Figure (b), select the desired resource name and click OK to add it.
   Note that if you want to remove the unwanted resource, select the resource
   name and click Remove .

Figure (a)
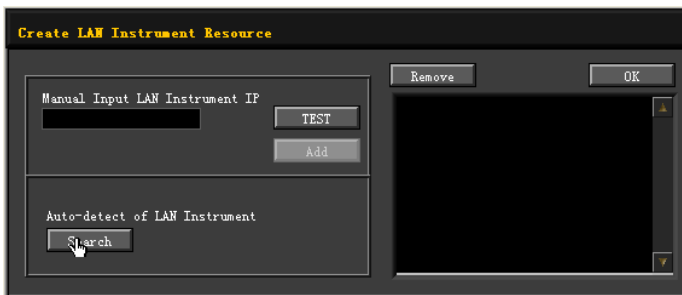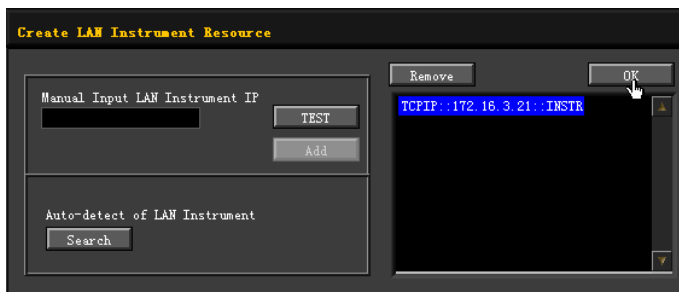

Figure (b)

4. View device resource

The resources found will appear under the "RIGOL Online Resource" directory as shown in the figure below.
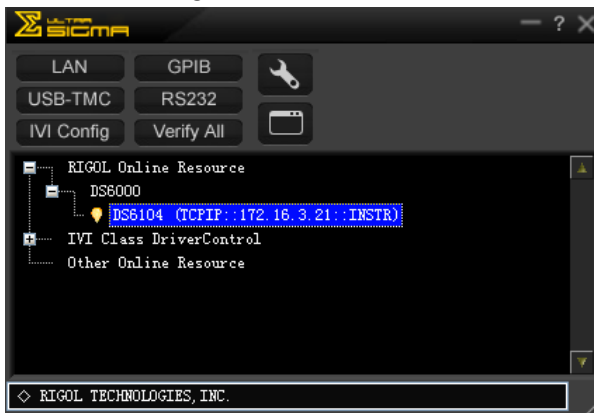
5. Communication test

Right click the resource name "DS6104 (TCPIP::172.16.3.21::INSTR)" to select "SCPI Panel Control" to turn on the remote command control panel from which you can send commands and read data as shown in the figure below.
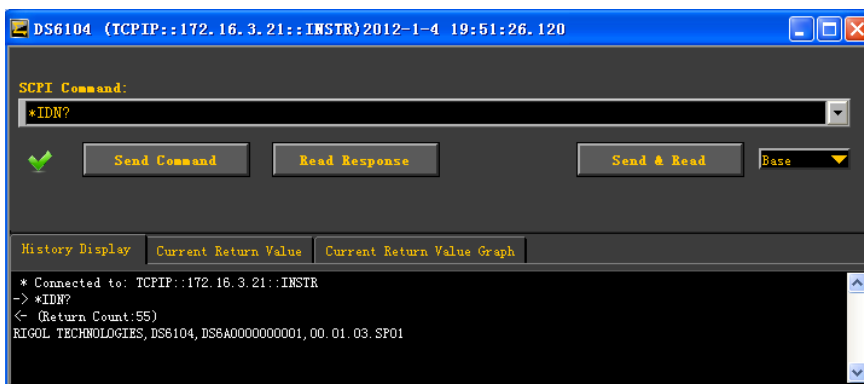
6. Load LXI webpage

As this oscilloscope conforms to LXI-C standards, you can load LXI webpage through Ultra Sigma (right-clicking the resources name and selecting LXI-Web; or directly input the IP address in the browser). Various important information about the oscilloscope including the model number, manufacturer, serial number, description, MAC address and IP address will be displayed on the webpage as shown in the figure below.

Remote Control via GPIB

1. **Connect the device**

   Use the USB to GPIB interface converter to extend a GPIB interface for the oscilloscope. Then connect the oscilloscope to your PC using a GPIB cable.

2. **Install the driver of GPIB card**

   Install the driver of the GPIB card which has been connected to the PC correctly.

3. **Set the GPIB address**

   Set the GPIB address of the oscilloscope according to the description in "To Set the GPIB Address".

4. **Search device resource**

   Start up the Ultra Sigma and click GPIB to open the panel as shown in the figure below. Click "Search" and the software will search the oscilloscope resources connected to the PC. The device resources will be displayed on the right side of the panel.
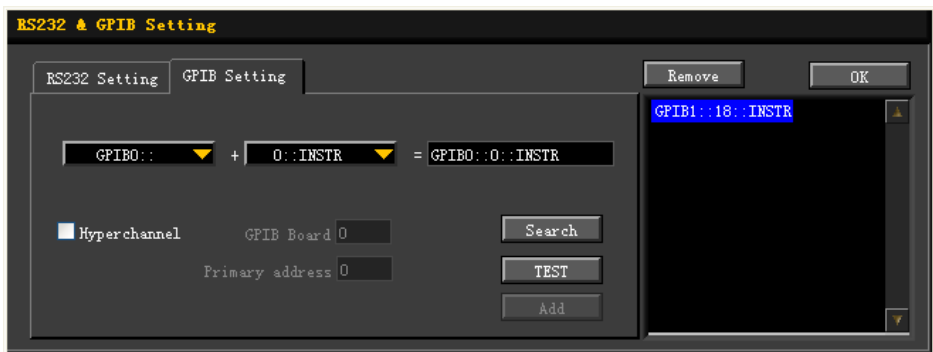
   If resources can not be found automatically:
   - Select the GPIB card address of the PC from the comboBox of "GPIB::" and select the GPIB address set in the oscilloscope from the comboBox of "::INSTR".
   - Click "Test" to check whether the GPIB communication works normally; if not, please follow the corresponding prompt messages to solve the problem.

5. View device resource

Click "OK" to return back to the main interface of Ultra Sigma. The resources found will appear under the "RIGOL Online Resource" directory.
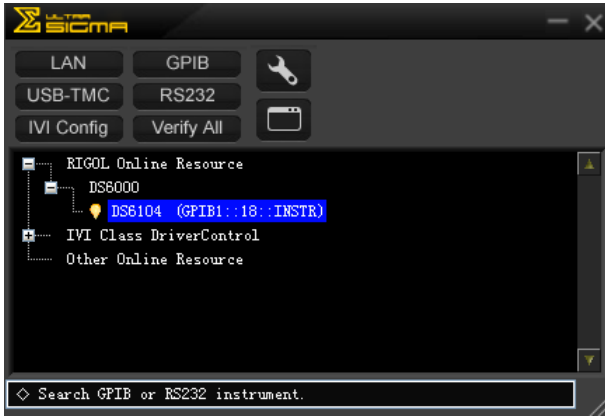
6. Communication Test

Right-click the resource name "DS6104 (GPIB0::18::INSTR)" to select "SCPI Panel Control" to turn on the remote command control panel from which you can send commands and read data as shown in the figure below.

15   Troubleshooting

The commonly encountered failures and their solutions are listed below. When you encounter those problems, please solve them following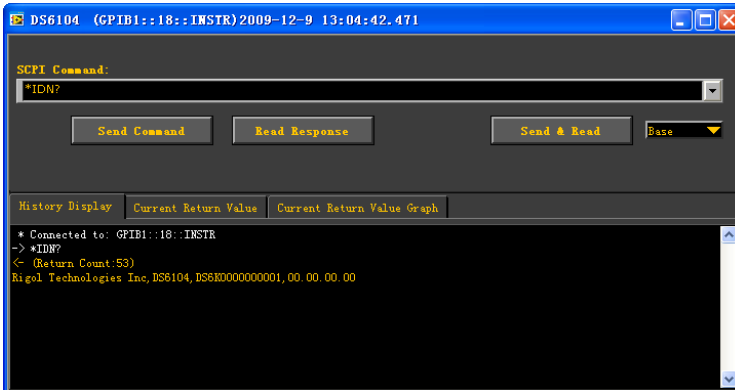 the corresponding steps. If the problem remains still, please contact **RIGOL** and provide your device information (acquisition method: UTIL → **System** → **System Info**).

1. **The screen is still dark (no display) after power on:**
   (1) Check whether the power is correctly connected or if the battery is correctly installed.
   (2) Check whether the power switch is really on.
   (3) Restart the instrument after finishing the above inspections.
   (4) If it still does not work correctly, please contact **RIGOL**.

2. **The signal is sampled but no waveform of the signal is displayed:**
   (1) Check whether the probe is correctly connected to the signal connecting wire.
   (2) Check whether the signal connecting wire is correctly connected to the BNC (namely channel connector).
   (3) Check whether the probe is correctly connected to the item under test.
   (4) Check whether there are signals generated from the item under test (you can connect the probe compensation signal to the problematic channel to determine which has problems, the channel or the item under test).
   (5) Resample the signal.

3. **The tested voltage amplitude is greater or lower than the actual value (note that this problem usually occurs when probe is used):**
   Check whether the attenuation coefficient of the channel complies with the attenuation ratio of the probe.

4. **There is waveform display but not stable:**
   (1) Check the trigger signal source: check whether the **Source** item at the front panel complies with the signal channel actually used.
   (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) Try to change the **Coupling** to "HF Reject"or "LF Reject" to filter out the high-frequency or low-frequency noise that disturbs the trigger.

(4) 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 the **Auto** button could automatically finish the above setting.

6. The display of waveform is ladder-like:

(1) The horizontal time base 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 ladder-like display. Set **Type** to "Dots" to solve the problem.

7. Fail to connect PC or PictBridge printer through USB:

Check the **IO Setting** menu in **UTIL** to make sure whether the setting in **USB Device** matches the currently connected device. If needed, restart the oscilloscope.

8. The USB storage device can not be recognized:

(1) Check whether the USB storage device can work normally.

(2) Make sure that the USB storage device being used is flash storage type. This oscilloscope does not support hardware storage type.

(3) Make sure whether the capacity of the USB storage device is too large. It is recommended that the capacity of the USB storage device being used with this oscilloscope is no larger than 4 GBytes.

(4) Restart the instrument and then insert the USB storage to check it.

(5) If the USB storage device still can not be used normally, please contact **RIGOL**.

16   Specifications

All the specifications are guaranteed except the parameters marked with "Typical" and the oscilloscope needs to operate for more than 30 minutes under the specified operation temperature.

Sample

Sample Mode	Real-time Sample, Equivalent Sample
Real Time Sample Rate	5 GSa/s (single-channel) 2.5 Gsa/s (dual-channel)
Equivalent Sample Rate	100 Gsa/s
Peak Detect	200 ps (single-channel) 400 ps (dual-channel)
Averaging	After all the channels finish N samples at the same time, N can be 2, 4, 8, 16, 32, 64, 128, 256, 512, 1024, 2048, 4096 or 8192.
High Resolution	12 bits of resolution when ≥5 μs/div @ 5 GSa/s (or ≥10 μs/div @ 2.5 GSa/s).
Memory Depth	single-channel: Auto, 14k pts, 140k pts, 1.4M pts, 14M pts and 140M pts are available dual-channel: Auto, 7k pts, 70k pts, 700k pts, 7M pts and 70M pts are available

Input

Number of Channels	DS6XX4: four channels DS6XX2: two channels
Input Coupling	DC, AC or GND
Input Impedance	(1 MΩ±1%) || (14 pF±3 pF) or 50 Ω±1.5%
Probe Attenuation Coefficient	0.01X to 1000X, in 1-2-5 step

Maximum Input Voltage (1MΩ)	Maximum Input Voltage of the Analog Channel
	CAT I 300 Vrms, CAT II 100 Vrms,
	Transient Overvoltage 1000V pk
	with RP2200 10:1 probe: CAT II 300 Vrms
	with RP3300 10:1 probe: CAT II 300 Vrms
	with RP3500 10:1 probe: CAT II 300 Vrms
	with RP5600 10:1 probe: CAT II 300 Vrms

Horizontal

Timebase Scale	DS606X: 1 ns to 1000 s
	DS610X: 500 ps to 1000 s
Timebase Accuracy	≤ ±(5 + 2 ×instrument age in years) ppm
Delay Range	Pre-trigger (negative delay): ≥1 screen width
	Post-trigger (positive delay): 1 s to 100,000 s
Timebase Mode	Y-T, X-Y, Roll, Delayed Sweep
Number of XYs	2 simultaneously
Waveform Capture Rate[1]	150,000 wfms (vector display); 180,000 wfms (dots display)

Vertical

Bandwidth (-3dB)	DS606X: DC to 600 MHz
	DS610X: DC to 1 GHz
Single-shot Bandwidth	DS606X: DC to 600 MHz
	DS610X: DC to 1 GHz (each channel)
Vertical Resolution	8bit, two channels sample at the same time
Vertical Scale	2 mV/div to 5 V/div (1 MΩ)
	2 mV/div to 1 V/div (50 Ω)
Offset Range	2 mV/div to 124 mV/div: ± 1.2 V (50 Ω)
	126 mV/div to 1 V/div: ± 12 V (50 Ω)
	2 mV/div to 225 mV/div: ± 2 V (1 MΩ)
	230 mV/div to 5 V/div: ± 40 V (1 MΩ)
Bandwidth Limit[2]	20 MHz or 250 MHz

Low Frequency Response (AC Coupling -3dB)	≤5 Hz (on BNC)
Calculated Rise Time[2]	DS606X: 600 ps DS610X: 400 ps
DC Gain Accuracy	±2% full scale
DC Offset Accuracy	200 mV/div to 5 V/div: 0.1 div ± 2 mV±0.5% offset value 1 mV/div to 195 mV/div: 0.1 div ± 2 mV±1.5% offset value
ESD Tolerance	±2 kV
Channel to Channel Isolation	DC to maximum band width: >40 dB

Trigger

Trigger Level Range	Internal	±6 div from center screen
	EXT	± 0.8 V
Trigger Mode	Auto, Normal, Single	
Holdoff Range	100 ns to 10 s	
High Frequency Rejection[2]	50 kHz	
Low Frequency Rejection[2]	5 kHz	
Edge Trigger		
Edge Type	Rising, Falling, Rising&Falling	
Pulse Trigger		
Pulse Condition	Positive Pulse Width (greater than, lower than, within specific interval) Negative Pulse Width (greater than, lower than, within specific interval)	
Pulse Width Range	4 ns to 4 s	
Slope Trigger		
Slope Condition	Positive Slope (greater than, lower than, within specific interval) Negative Slope (greater than, lower than, within specific	

	interval)
Time Setting	10 ns to 1 s
Video Trigger	
Signal Standard	Support standard NTSC, PAL and SECAM broadcasting standards; support 480P, 576P, 720P, 1080P and 1080I HDTV standards
Pattern Trigger	
Pattern Setting	H, L, X, Rising Edge, Falling Edge
RS232/UART Trigger	
Trigger Condition	Start, Error, Check Error, Data
Polarity	Normal, Invert
Baud Rate	2400 bps, 4800 bps, 9600 bps, 19200 bps, 38400 bps, 57600 bps, 115200 bps, User
Data Bits	5 bit, 6 bit, 7 bit, 8 bit
I2C Trigger	
Trigger Condition	Start, Restart, Stop, Missing ACK, Address, Data, A&D
Address Bits	7 bit, 8 bit, 10 bit
Address Range	0 to 127, 0 to 255, 0 to 1023
Byte Length	1 to 5
SPI Trigger	
Trigger Condition	CS, Timeout
Timeout Value	100 ns to 1 s
Data Bits	4 bit to 32 bit
Data	H, L, X
Clock Edge	Rising Edge, Falling Edge
CAN Trigger	
Signal Type	Rx, Tx, CAN_H, CAN_L, Differential
Trigger Condition	SOF, EOF, Frame Type
Baud Rate	10 kbps, 20 kbps, 33.3 kbps, 50 kbps, 62.5 kbps, 83.3 kbps, 100 kbps, 125 kbps, 250 kbps, 500 kbps, 800 kbps, 1 Mbps, User
Sample Point	5% to 95%
Frame Type	Data, Remote, Error, OverLoad
FlaxRay Trigger	
Baud Rate	2.5 Mb/s, 5 Mb/s, 10 Mb/s
Trigger Condition	Frame, Symbol, Error, TSS

USB Trigger

Signal Speed	Low Speed, Full Speed
Trigger condition	SOP, EOP, RC, Suspended, Exit Suspended

Measure

Cursor	Manual Mode	Voltage Deviation between Cursors (△V) Time Deviation between Cursors (△T) Reciprocal of △T (Hz) (1/△T)
	Track Mode	Voltage and Time Values of the Waveform Point
	Auto Mode	Allow to display cursors during auto measurement
Auto Measurement	Measurements of Maximum, Minimum, Peak-Peak Value, Top Value, Bottom Value, Amplitude, Average, Mean Square Root, Overshoot, Pre-shoot, Area, Period Area, Frequency, Period, Rise Time, Fall Time, Positive Pulse Width, Negative Pulse Width, Positive Duty Cycle, Negative Duty Cycle, Delay A→B⌇, Delay A→B⌇, Phase A→B⌇, Phase A→B⌇	
Number of Measurements	Display 5 measurements at the same time.	
Measurement Range	Screen or cursor	
Measurement Statistic	Average, Max, Min, Standard Deviation, Number of Measurements	
Frequency Counter	Hardware 6 bits frequency counter (channels available: DS606x, CH1/CH2; DS610x, CH1/CH2/CH3/CH4)	

Math Operation

Waveform Operation	A+B, A-B, A×B, A/B, FFT, Editable Advanced Operation, Logic Operation
FFT Window Function	Rectangle, Hanning, Blackman, Hamming
FFT Display	Split, Full Screen
FFT Vertical Scale	Vrms, dB
Logic Operation	AND, OR, NOT, XOR
Math Function	Intg, Diff, Log, Exp, Sqrt, Sine, Cosine, Tangent
Number of Buses for Decoding	2
Decoding Type	Parallel (standard), RS232/UART (option), I2C (option) , SPI (DS6XX4 option), CAN (option), FlaxRay (option)

Display

Display Type	10.1 inches (257 mm) TFT LCD display
Display Resolution	800 Horizontal ×RGB×480 Vertical Pixel
Display Color	160,000 Color
Persistence Time	Minimum, 50 ms, 100 ms, 200 ms, 500ms, 1 s, 2 s, 5 s, 10 s, 20 s, Infinite
Display Type	Dots, Vectors
Real-time Clock	Time and Date (user adjustable)

I/O

Standard Ports	Dual USB HOST, USB device, LAN, VGA Output, 10 MHz Input/Output, Aux output (TrigOut, Fast, GND, PassFail, Calibration)
Printer Compatibility	PictBridge

General Specifications

Probe Compensation Output		
Output Voltage[2]	About 3 V, peak-peak	
Frequency[2]	1kHz	
Power		
Power Voltage	100-127 V, 45-440 Hz	
	100-240 V, 45-65 Hz	
Power	Maximum 150W	
Fuse	3 A, T Degree, 250 V	
Environment		
Temperature Range	Operating: 0 ℃ to +50 ℃	
	Non-operating: -20 ℃ to +70 ℃	
Cooling Method	Fan cooling	
Humidity Range	Under +35 ℃: ≤90% Relative Humidity	
	+35 ℃ to +50 ℃: ≤60% Relative Humidity	
Altitude	Operating: under 3,000 meters	
	Non-operating: under 15,000 meters	
Mechanical Characteristics		
Size[3]	W×H×D = 399.0 mm×255.3 mm×123.8 mm	
Weight[4]	Package Excluded	5.345 ± 0.2 kg
	Package Included	10.8 ± 1 kg
Calibration Interval		
The recommended calibration interval is one year.		
Regulatory Information		
Electromagnetic Compatibility	2004/108/EC	
	Execution standard EN 61326-1:2006 EN 61326-2-1:2006	
Safety	UL 61010-1:2004；CAN/CSA-C22.2 NO. 61010-1-2004；	
	EN 61010-1:2001；IEC 61010-1:2001	

Note[1]: Maximum value. In single-channel mode, sine signal with 10 ns horizontal scale, 4 div input amplitude and 10 MHz frequency, edge trigger.

Note[2]: Typical.

Note[3]: Supporting legs and handle folded, knob height included, front panel cover excluded.

Note[4]: DS6104 model, standard configuration.

17   Appendix

Appendix A: Options and Accessories

	Description	Order Number
Model	DS6104 (1 GHz, 4-channel)	DS6104
	DS6102 (1 GHz, dual-channel)	DS6102
	DS6064 (600 MHz, 4-channel)	DS6064
	DS6062 (600 MHz, dual-channel)	DS6062
Standard Accessories	Power Cord conforming to the standard of the country	-
	Front Panel Cover	FPC-DS-6
	USB Cable	CB-USB-150
	2 or 4 Passive Probes (600 MHz)	RP5600
	1 or 2 Passive Probes (1.5 GHz)	RP6150 (for DS610X)
	Quick Guide	-
	Resource CD (User's Guide and Application Software)	-
Optional Accessories	Active Differential Probe (1.5 GHz)	RP7150
	Passive Probes (500 MHz)	RP3500
	11.1 V, 147 Wh Lithium Battery Set	BAT
	USB to GPIB Interface Converter	USB-GPIB
	Desk Mount Instrument Arm	ARM
	Rack Mount Kit	RM-DS-6
Decoding Options	RS232/UART Decoding Kit	SD-RS232-DS6
	I2C /SPI Decoding Kit	SD-I2C/SPI-DS6
	CAN Decoding Kit	SD-CAN-DS6
	FlexRay Decoding Kit	SD-FLEXRAY-DS6

Note: all the options or accessories can be ordered from you local **RIGOL** Office.

Appendix B: Warranty

RIGOL warrants that its products mainframe and accessories will be free from defects in materials and workmanship within the warranty period.

If a product is proven to be defective within the respective period, **RIGOL** guarantees the free replacement or repair of products which are approved defective. For detailed warranty description, please refer to **RIGOL** official website or the warranty card.

To get repair service and a complete copy of the warranty description, please contact with your nearest **RIGOL** sales and service office.

RIGOL does not provide any other warranty items except the one being provided by this summary and the warranty statement. The warranty items include but not being subjected to the hint guarantee items related to tradable characteristic and any particular purpose. **RIGOL** will not take any responsibility in cases regarding to indirect, particular and ensuing damage.

Appendix C: Any Question or Comment?

If you have any question or comment on our document, please mail to:
service@rigol.com

Index