

RIGOL

User's Guide

MSO4000/DS4000 Series Digital Oscilloscope

Oct. 2013

RIGOL Technologies, Inc.

Guaranty and Declaration

Copyright

© 2013 RIGOL Technologies, Inc. All Rights Reserved.

Trademark Information

RIGOL is a registered trademark of RIGOL Technologies, Inc.

Publication Number

UGA15103-1110

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 material.
- **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 certifications are in progress.

Contact Us

If you have any problem or requirement when using our products or this manual, please contact **RIGOL**.

E-mail: service@rigol.com

Websites: 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 the power cord designed for the instrument and authorized for use within the local country where the product is being used.

Ground The Instrument.

The instrument is grounded through the Protective Earth lead of the power cord. To avoid electric shock, it is essential to connect the earth terminal of power cord to the Protective Earth terminal before any inputs or outputs.

Connect The Probe Properly.

Do not connect the ground 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 more information about ratings before connecting.

Use Proper Overvoltage Protection.

Make sure that no overvoltage (such as that produced by a thunderstorm) can reach the product, or else the operator might expose to danger of electrical shock.

Do Not Operate Without Covers.

Do not operate the instrument with covers or panels removed.

Change The Power Fuse.

If the power fuse needs to be changed, please return the instrument back to our factory and the **RIGOL** authorized operator will change it for you.

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 especially to circuits or accessories must be performed by **RIGOL** authorized personnel.

Keep Well Ventilation.

Inadequate ventilation may cause increased temperature or damage to the device. So please keep well ventilated and inspect the intake and fan regularly.

Do Not Operate In Wet Conditions.

In order to avoid short circuiting to the interior of the device or electric shock, please do not operate in a humid environment.

Do Not Operate in an Explosive Atmosphere.

In order to avoid damages to the device or personal injury, it is important to operate the device away from an explosive atmosphere.

Keep Product Surfaces Clean and Dry.

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

Electrostatic Prevention.

Minimize electrostatic discharge to avoid damage induced by static discharges. Always ground both the internal and external conductors of cables to neutralize static before making connections.

Handling Safety.

Please handle with care during transportation to avoid damage to buttons, knob interfaces and 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 conditions or practices that could result in injury or loss of life.



CAUTION

Caution statements indicate conditions or practices that could result in damage to this product or other property.

Terms on the Product. These terms may appear on the product:

DANGER indicates an injury or hazard may immediately happen.

WARNING indicates an injury or hazard may occur.

CAUTION indicates a potential damage to the instrument or other property might occur.

Symbols on the Product. These symbols may appear on the product:



**Hazardous
Voltage**



**Safety
Warning**



**Protective
Earth
Terminal**



**Chassis
Ground**



**Test
Ground**

Allgemeine Sicherheits Informationen

Überprüfen Sie die folgenden Sicherheitshinweise sorgfältig um Personenschäden oder Schäden am Gerät und an damit verbundenen weiteren Geräten zu vermeiden. Zur Vermeidung von Gefahren, nutzen Sie bitte das Gerät nur so, wie in diesem Handbuch angegeben.

Um Feuer oder Verletzungen zu vermeiden, verwenden Sie ein ordnungsgemäßes Netzkabel.

Verwenden Sie für dieses Gerät nur das für ihr Land zugelassene und genehmigte Netzkabel.

Erden des Gerätes.

Das Gerät ist durch den Schutzleiter im Netzkabel geerdet. Um Gefahren durch elektrischen Schlag zu vermeiden, ist es unerlässlich, die Erdung durchzuführen. Erst dann dürfen weitere Ein- oder Ausgänge verbunden werden.

Anschluss eines Tastkopfes.

Die Erdungsklemmen der Sonden sind auf dem gleichen Spannungspegel des Instruments geerdet. Schließen Sie die Erdungsklemmen an keine hohe Spannung an.

Beachten Sie alle Anschlüsse.

Zur Vermeidung von Feuer oder Stromschlag, beachten Sie alle Bemerkungen und Markierungen auf dem Instrument. Befolgen Sie die Bedienungsanleitung für weitere Informationen, bevor Sie weitere Anschlüsse an das Instrument legen.

Verwenden Sie einen geeigneten Überspannungsschutz

Stellen Sie sicher, daß keinerlei Überspannung (wie z.B. durch Gewitter verursacht) das Gerät erreichen kann. Andernfalls besteht für den Anwender die Gefahr eines Stromschlages.

Nicht ohne Abdeckung einschalten.

Betreiben Sie das Gerät nicht mit entfernten Gehäuse-Abdeckungen.

Betreiben Sie das Gerät nicht geöffnet

Der Betrieb mit offenen oder entfernten Gehäuseteilen ist nicht zulässig. Nichts in entsprechende Öffnungen stecken (Lüfter z.B.)

Passende Sicherung verwenden

Setzen Sie nur die spezifikationsgemäßen Sicherungen ein.

Vermeiden Sie ungeschützte Verbindungen

Berühren Sie keine unisolierten Verbindungen oder Baugruppen, während das Gerät

in Betrieb ist.

Betreiben Sie das Gerät nicht im Fehlerfall

Wenn Sie am Gerät einen Defekt vermuten, sorgen Sie dafür, bevor Sie das Gerät wieder betreiben, dass eine Untersuchung durch qualifiziertes Kundendienstpersonal durchgeführt wird. Jedwede Wartung, Einstellarbeiten oder Austausch von Teilen am Gerät, sowie am Zubehör dürfen nur von **RIGOL** autorisiertem Personal durchgeführt werden.

Belüftung sicherstellen

Unzureichende Belüftung kann zu Temperaturanstiegen und somit zu thermischen Schäden am Gerät führen. Stellen Sie deswegen die Belüftung sicher und kontrollieren regelmäßig Lüfter und Belüftungsöffnungen.

Nicht in feuchter Umgebung betreiben

Zur Vermeidung von Kurzschluß im Geräteinneren und Stromschlag betreiben Sie das Gerät bitte niemals in feuchter Umgebung.

Nicht in explosiver Atmosphäre betreiben

Zur Vermeidung von Personen- und Sachschäden ist es unumgänglich, das Gerät ausschließlich fernab jedweder explosiven Atmosphäre zu betreiben.

Geräteoberflächen sauber und trocken halten

Um den Einfluß von Staub und Feuchtigkeit aus der Luft auszuschließen, halten Sie bitte die Geräteoberflächen sauber und trocken.

Schutz gegen elektrostatische Entladung (ESD)

Sorgen Sie für eine elektrostatisch geschützte Umgebung, um somit Schäden und Funktionsstörungen durch ESD zu vermeiden. Erden Sie vor dem Anschluß immer Innen- und Außenleiter der Verbindungsleitung, um statische Aufladung zu entladen.

Sicherer Transport

Transportieren Sie das Gerät sorgfältig (Verpackung!), um Schäden an Bedienelementen, Anschlüssen und anderen Teilen zu vermeiden.

Sicherheits Begriffe und Symbole

Begriffe in diesem Guide. Diese Begriffe können in diesem Handbuch auftauchen:



WARNING

Die Kennzeichnung WARNING beschreibt Gefahrenquellen die leibliche Schäden oder den Tod von Personen zur Folge haben können.



CAUTION

Die Kennzeichnung Caution (Vorsicht) beschreibt Gefahrenquellen die Schäden am Gerät hervorrufen können.

Begriffe auf dem Produkt. Diese Bedingungen können auf dem Produkt erscheinen:

DANGER weist auf eine Verletzung oder Gefährdung hin, die sofort geschehen kann.

WARNING weist auf eine Verletzung oder Gefährdung hin, die möglicherweise nicht sofort geschehen.

CAUTION bedeutet, dass eine mögliche Beschädigung des Instruments oder anderer Gegenstände auftreten kann.

Symbole auf dem Produkt. Diese Symbole können auf dem Produkt erscheinen:



**Gefährliches
pannung**



**Sicherheits-
Hinweis**



Schutz-erde



Gehäusemasse



Erde

Measurement Category

Measurement Category

The MSO4000/DS4000 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 limit capability of the equipment is made known to the user.

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

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

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

Ventilation Requirement

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



WARNING

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

Working Environment

Temperature

Operating: 0°C to +50°C

Non-operating: -40°C to +70°C

Humidity

0°C to +30°C: ≤95% relative humidity

+30°C to +40°C: ≤75% relative humidity

+40°C to +50°C: ≤45% relative humidity



WARNING

To avoid short circuits 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 a thunderstorm) can reach the product, or else the operator might be exposed to the 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. An occasional and temporary conductive environment may be caused by condensation. For example, a general indoor environment.

Pollution degree 3: Conductive pollution occurs, or dry, non-conductive pollution occurs which becomes conductive due to condensation. For example, a 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 at places 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:

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 mild detergent or water). When cleaning the LCD, take care to avoid scratching it.



CAUTION

To avoid damage to the instrument, do not expose to corrosive liquids.



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 applicable European Union requirements 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.

MSO4000/DS4000 Series Overview

MSO4000/DS4000 is a multifunctional and high performance digital oscilloscope developed using of the **RIGOL** Ultra Vision chipset. MSO4000 series is a high-performance mixed-signal oscilloscope which is optimized for the debugging in embedded design and test areas that require simultaneous measurement of both analog and digital signals. The MSO4000 provides superb specifications, convenient functions, and easy-to-use design which help users fulfill their tasks (such as measurement and remote control) more quickly.

Main features:

- 500 MHz, 350 MHz, 200 MHz and 100 MHz bandwidth
- 2 +16 channel or 4 +16 channel MSO4000 series digital oscilloscope
- Dual-channel or 4-channel model DS4000 series digital oscilloscope
- Analog channel sample rates up to 4 GSa/s, 140 Mpts maximum memory depth (standard). Digital channel sample rates up to 1 GSa/s, 28 Mpts maximum memory depth (per channel)
- 110,000 wfms/s (digital channels are turned off) maximum waveform refresh rate or 85,000 wfms/s (digital channels are turned on, dots display) maximum waveform refresh rate
- Ultra Vision technology
- 9.0 inch, WVGA (800*480) 160,000 color TFT LCD, with ultra-wide screen, vivid picture, low power consumption and long service life
- Automatic probe identification
- Adjustable brightness of analog channel waveform
- Auto setting of waveform display (**AUTO**)
- Various trigger functions including multiple protocol triggers and supporting digital channels as trigger source
- Support standard parallel decoding, multiple serial decoding options and mixed decoding for analog and digital channels
- Auto measurements of 24 waveform parameters and measurement functions with statistics
- Real-time waveform recording, waveform playback, record open (constant on) and waveform analysis, supporting digital channels recording and playback
- Precise delayed sweep function
- Built-in FFT function

- Built-in logic analyzer 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 and printer
- Conforms to LXI-C instrument standards which enable quick, economic and efficient creation and reconfiguration of test system
- Supports remote command control
- Embedded help enables easier information access
- Supports multiple languages and Chinese/English input
- Supports one-key measurement, storage and print
- Convenient digital channel grouping and operating functions

Document Overview

[1 Quick Start](#)

Provides information about preparations before using the instrument and a brief introduction of the instrument.

[2 To Set the Vertical System](#)

Introduces the functions of the vertical system of the oscilloscope.

[3 To Set the Horizontal System](#)

Introduces the functions of the horizontal system of the oscilloscope.

[4 To Set the Sample System](#)

Introduces the functions of the sample system of the oscilloscope.

[5 To Trigger the Oscilloscope](#)

Introduces the trigger mode, trigger coupling, trigger holdoff, external trigger and various trigger types of the oscilloscope.

[6 To Make Measurements](#)

Introduces how to make math operation, cursor and auto measurements.

[7 Logic Analyzer](#)

Introduces how to use the logic analyzer function.

[8 Protocol Decoding](#)

Introduces how to decode the input signal using those common protocols.

[9 Reference Waveform](#)

Introduces how to compare the input waveform with the reference waveform.

[10 Pass/Fail Test](#)

Introduces how to monitor the input signal using the Pass/Fail test.

[11 Waveform Record](#)

Introduces how to analyze the input signal using waveform recording.

[12 Display Control](#)

Introduces how to control the display of the oscilloscope.

[13 Store and Recall](#)

Introduces how to store and recall the measurement result and the setting of the oscilloscope.

[14 System Function Setting](#)

Introduces how to set the remote interface and system-related functions.

[15 Remote Control](#)

Introduces how to control the oscilloscope remotely.

[16 Troubleshooting](#)

Introduces how to deal with common failures of the oscilloscope.

[17 Specifications](#)

Lists the specifications and general specifications of the oscilloscope.

[18 Appendix](#)

Provides common information such as options and accessories.

Format Conventions in this Manual:

Front panel key: denoted by the format of "Text Box + Button Name (Bold)". For example, **Storage**.

Menu softkey: denoted by the format of "Character Shading + Menu Word (Bold)". For example, **Storage**.

Operation steps: denoted by the arrow "→", for example, **Storage** → **Storage**.

Link: denoted by the format of "Blue Characters (Bold + Underlined)". For example, [Store and Recall](#).

Knob:

Logo	Knob	Logo	Knob
	Multi-function Knob	VERTICAL  <u>SCALE</u>	Vertical Scale Knob
	Navigation Knob	VERTICAL  <u>POSITION</u>	Vertical Position Knob
HORIZONTAL  <u>SCALE</u>	Horizontal Scale Knob	TRIGGER  <u>LEVEL</u>	Trigger Level Knob
HORIZONTAL  <u>POSITION</u>	Horizontal Position Knob	—	—

Content Conventions in this Manual:

This manual takes the MSO4054 for all examples and the descriptions here contains the functions and performance of other models. The MSO4000 and DS4000 series includes the following models:

Model	Analog Bandwidth	Analog Channels	Digital Channel Bandwidth	Digital Channels
MSO4054	500MHz	4	250MHz	16
MSO4052	500MHz	2	250MHz	16
MSO4034	350MHz	4	250MHz	16
MSO4032	350MHz	2	250MHz	16
MSO4024	200MHz	4	250MHz	16
MSO4022	200MHz	2	250MHz	16
MSO4014	100MHz	4	250MHz	16
MSO4012	100MHz	2	250MHz	16
DS4054	500MHz	4	—	—
DS4052	500MHz	2	—	—
DS4034	350MHz	4	—	—
DS4032	350MHz	2	—	—
DS4024	200MHz	4	—	—
DS4022	200MHz	2	—	—
DS4014	100MHz	4	—	—
DS4012	100MHz	2	—	—

Note: The content related to the logic analyzer (digital channels) in this document only applies to MSO4000 mixed signal digital oscilloscope.

Contents

Guaranty and Declaration	I
Safety Requirement	II
General Safety Summary	II
Safety Terms and Symbols	IV
Allgemeine Sicherheits Informationen.....	V
Sicherheits Begriffe und Symbole	VII
Measurement Category	VIII
Ventilation Requirement	IX
Working Environment	X
General Care and Cleaning	XII
Environmental Considerations.....	XIII
MSO4000/DS4000 Series Overview	XIV
Document Overview.....	XVI
1 Quick Start	1-1
General Inspection	1-2
Appearance and Dimensions.....	1-3
To Prepare the Oscilloscope for Use.....	1-4
To Remove the Cover	1-4
To Adjust the Supporting Legs.....	1-5
To Connect to AC Power Supply.....	1-6
Power-on Inspection	1-7
To Connect the Probe	1-8
Function Inspection.....	1-9
Probe Compensation	1-11
To Connect the Logic Probe	1-12
Front Panel Overview	1-13
Rear Panel Overview.....	1-14
Front Panel Function Overview.....	1-16
VERTICAL	1-16
HORIZONTAL	1-17
TRIGGER	1-18
CLEAR	1-18

- AUTO 1-19
- RUN/STOP 1-19
- SINGLE..... 1-19
- Multi-function Knob..... 1-20
- Navigation Knob 1-20
- Default 1-21
- Print..... 1-21
- Function Keys..... 1-21
- Record..... 1-22
- LOGIC ANALYZER 1-23
- User Interface..... 1-24
- To Use the Security Lock..... 1-29
- To Use the Built-in Help System..... 1-30
- 2 To Set the Vertical System..... 2-1**
 - To Enable the Analog Channel 2-2
 - Channel Coupling 2-3
 - Bandwidth Limit 2-4
 - Probe..... 2-5
 - Input Impedance 2-7
 - Waveform Invert 2-7
 - Vertical Scale 2-8
 - Vertical Expansion 2-9
 - Amplitude Unit..... 2-9
 - Channel Label..... 2-10
 - Delay Calibration..... 2-11
- 3 To Set the Horizontal System 3-1**
 - Delayed Sweep 3-2
 - Time Base Mode 3-4
 - Y-T Mode 3-4
 - X-Y Mode 3-5
 - Roll Mode 3-8
 - Horizontal Scale 3-9
 - Horizontal Reference 3-10
- 4 To Set the Sample System 4-1**
 - Acquisition Mode..... 4-2
 - Normal 4-2

Average	4-2
Peak Detect.....	4-4
High Resolution	4-4
Sample Mode	4-5
Sample Rate	4-6
LA Sample Rate.....	4-8
Memory Depth	4-9
Antialiasing	4-10
5 To Trigger the Oscilloscope	5-1
Trigger Source	5-2
Trigger Mode	5-3
Trigger Coupling.....	5-5
Trigger Holdoff.....	5-6
Noise Rejection	5-7
Trigger Type	5-8
Edge Trigger.....	5-9
Pulse Trigger	5-11
Runt Trigger	5-13
Nth Edge Trigger.....	5-15
Slope Trigger	5-17
Video Trigger.....	5-20
Pattern Trigger	5-22
RS232 Trigger.....	5-24
I2C Trigger.....	5-26
SPI Trigger	5-29
CAN Trigger.....	5-31
FlexRay Trigger.....	5-34
USB Trigger.....	5-36
Trigger Output Connector.....	5-38
6 To Make Measurements	6-1
Math Operation	6-2
Addition	6-2
Substraction	6-3
Multiplication.....	6-3
Division.....	6-4
FFT	6-5

Logic Operation	6-8
Advanced Operation.....	6-10
Auto Measurement	6-13
Quick Measurement after AUTO	6-13
One-key Measurement of 24 Parameters.....	6-14
Frequency Counter Measurement	6-19
Measurement Setting	6-20
To Clear the Measurement.....	6-22
All Measurement.....	6-23
Statistic Function	6-24
Measurement History	6-25
Cursor Measurement	6-26
Manual Mode	6-27
Track Mode	6-31
Auto Mode	6-34
X-Y Mode	6-35
7 Logic Analyzer	7-1
To Select the Logic Channel	7-2
To Turn on/off the Logic Channel.....	7-2
Group Setting	7-3
To Set the Waveform Size	7-3
Automatic Arrangement Setting.....	7-4
To Set the Threshold	7-4
To Use the Digital Bus	7-5
To Set the Label.....	7-6
Delay Calibration.....	7-6
8 Protocol Decoding	8-1
Parallel Decoding	8-2
RS232 Decoding (Option)	8-5
I2C Decoding (Option).....	8-10
SPI Decoding (Option).....	8-13
CAN Decoding (Option).....	8-16
FlexRay Decoding (Option).....	8-19
9 Reference Waveform	9-1
To Enable REF Function	9-2
To Set the Color	9-3

To Select REF Source	9-3
To Save to Internal Memory	9-3
To Adjust REF Waveform Display.....	9-4
To Export to Internal or External Memory	9-4
To Import from Internal or External Memory	9-4
10 Pass/Fail Test	10-1
To Enable Pass/Fail Test	10-2
To Select Source.....	10-3
To Create Mask	10-3
Test and Ouput	10-4
To Save the Test Mask	10-5
To Load the Test Mask	10-5
11 Waveform Record	11-1
Waveform Record.....	11-2
Record Constant On.....	11-4
Waveform Playback	11-5
Waveform Analysis	11-7
Analysis Based on Trace	11-10
Analysis Based on Pass/Fail Mask.....	11-11
12 Display Control.....	12-1
To Select the Display Type.....	12-2
To Set the Persistence Time	12-3
To Set the Waveform Brightness	12-5
To Set the Screen Grid	12-5
To Set the Grid Brightness.....	12-5
To Set the Menu Display.....	12-5
13 Store and Recall	13-1
Storage System	13-2
Storage Type	13-3
Internal Storage and Recall	13-5
External Storage and Recall	13-8
Disk Management.....	13-12
To Select File Type	13-13
To Create a New File or Folder	13-14
To Delete a File or Folder	13-17

- To Rename a File or Folder 13-18
- To Clear the Local Memory 13-18
- Factory..... 13-19
- 14 System Function Setting..... 14-1**
 - Remote Interface Configuration..... 14-2
 - LAN Setting..... 14-2
 - USB Device 14-6
 - To Set the GPIB Address 14-6
 - System-related 14-7
 - Sound 14-7
 - Language 14-7
 - System Information 14-8
 - Power-off Recall 14-8
 - System Time 14-9
 - Self-test Information 14-10
 - Screen..... 14-10
 - Error Information..... 14-11
 - Self-calibration 14-12
 - Power Status..... 14-13
 - External Trigger Impedance..... 14-13
 - Aux Output 14-14
 - Reference Clock..... 14-15
 - Option Management 14-16
- 15 Remote Control 15-1**
 - Remote Control via USB..... 15-2
 - Remote Control via LAN 15-6
 - Remote Control via GPIB..... 15-9
- 16 Troubleshooting 16-1**
- 17 Specifications 17-1**
- 18 Appendix 18-1**
 - Appendix A: Accessories and Options 18-1
 - Appendix B: Warranty..... 18-3
- Index..... 1**

1 Quick Start

This chapter introduces the oscilloscope, the front panel, rear panel and user interface as well as 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 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

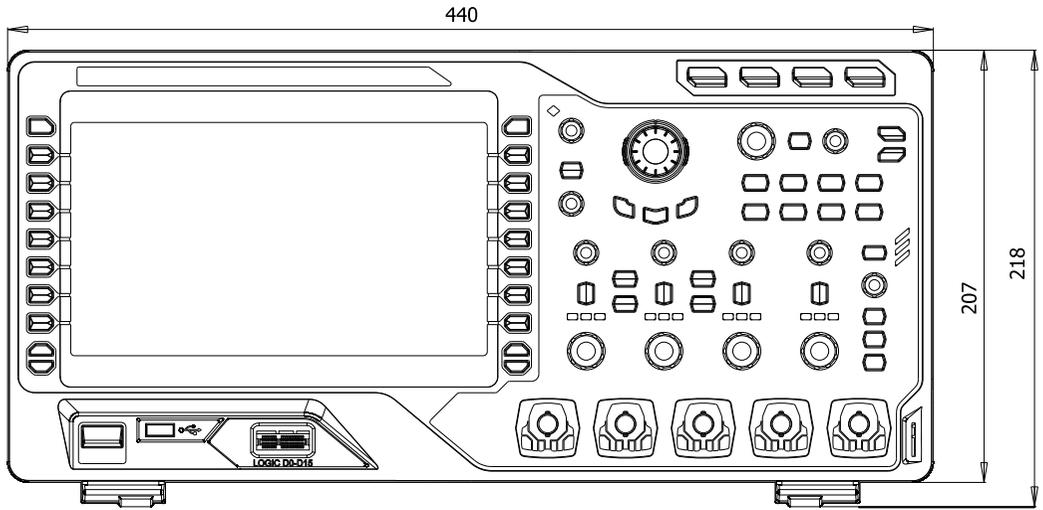


Figure 1-1 Front View

Unit: mm

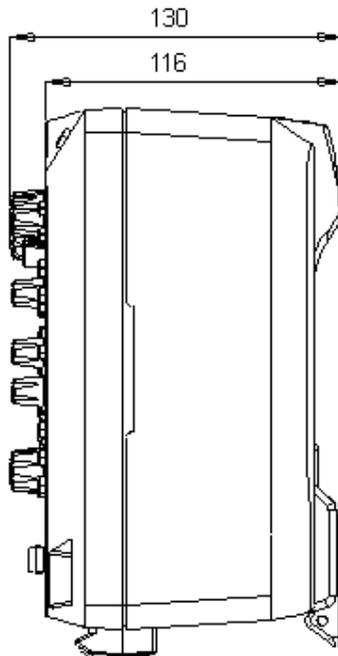


Figure 1-2 Side View

Unit: mm

To Prepare the Oscilloscope for Use

To Remove the Cover

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

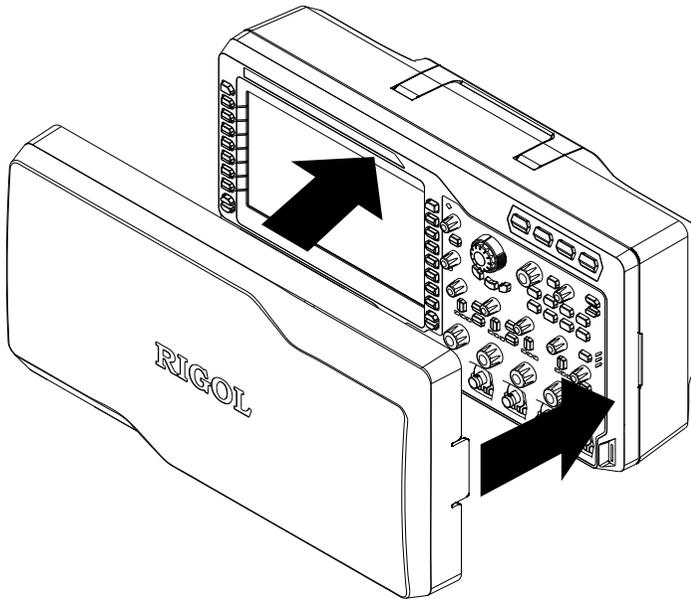


Figure 1-3 To Remove the Cover

To Adjust the Supporting Legs

For better observation, adjust the supporting legs. Unfold or fold the supporting legs in the arrow directions as shown in the figures below.

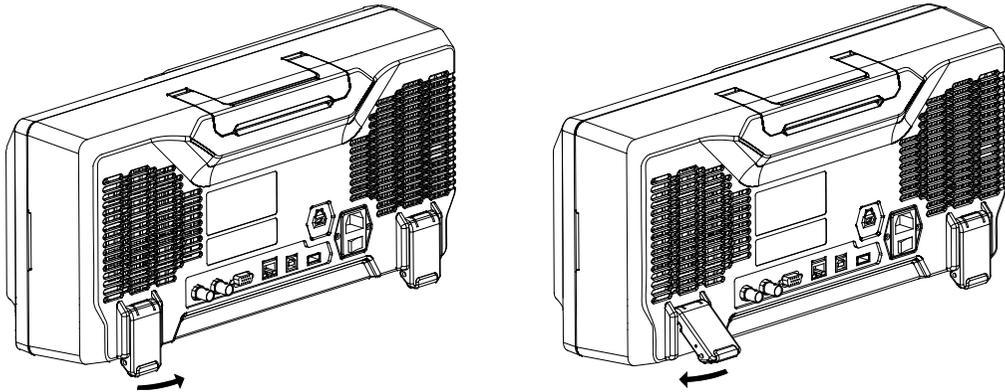


Figure 1-4 To Adjust the Supporting Legs

To Connect to AC Power Supply

This oscilloscope accepts two kinds of AC power supplies: 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. Turn on the power switch under the power plug. At this point, the oscilloscope is energized and the power key  at the lower-left corner of the front panel will pulse.

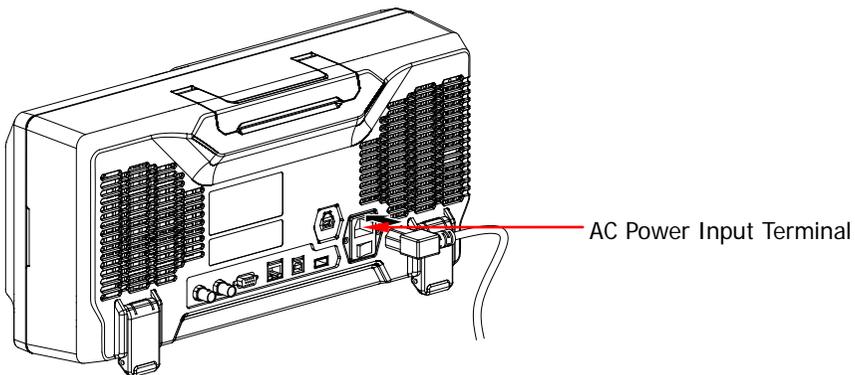


Figure 1-5 To Connect to AC Power Supply

Power-on Inspection

When the oscilloscope is energized, press the power key  at the lower-left corner of the front panel to start the oscilloscope. During the start-up process, the oscilloscope performs a series of self-tests and you can hear the sounds 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 the remaining trial time available in the "Installed Options" pop-up dialog box on the screen if the trial versions of options are currently installed on the instrument. When the instrument is shipped, a trial version of all options are provided with about 2000 minutes of on-time. Press **Utility** → **System** → **SelfTestInfo** to view the self-test results.

To Connect the Probe

RIGOL provides passive and active probes for the MSO4000/DS4000 series oscilloscopes. For detailed technical information, please refer to the corresponding Probe User's Guide. The following are the probes recommended for this oscilloscope.

Model	Description
RP3500	500 MHz, passive probe, 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 to be tested and connect the ground alligator clip of the probe to the circuit ground terminal.

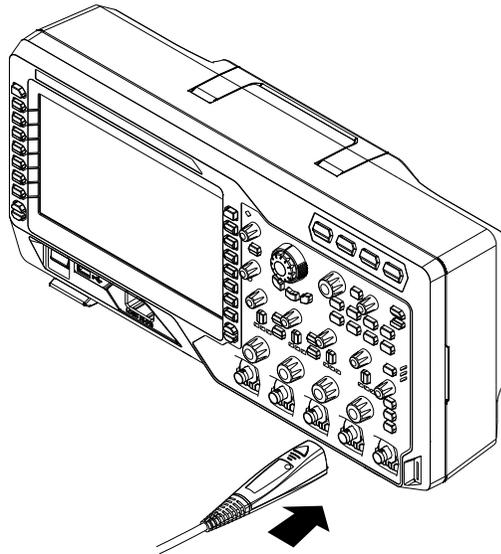


Figure 1-6 To Connect the Probe

Function Inspection

1. Press **Default** 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.

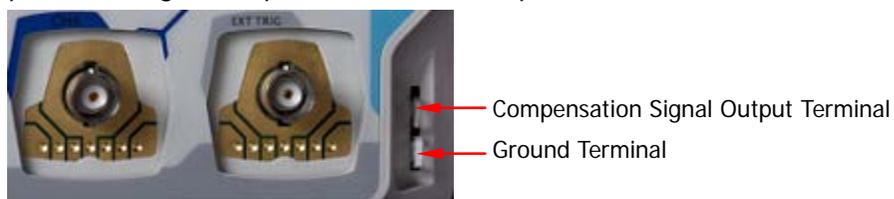


Figure 1-7 To Use the Compensation Signal

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:

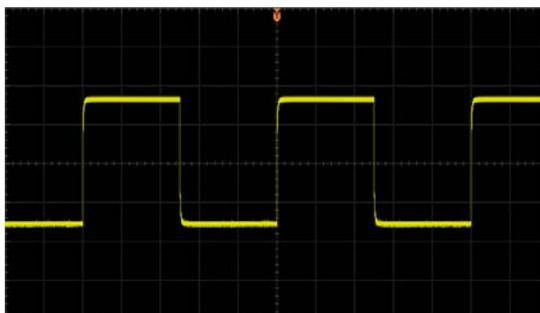


Figure 1-8 Square Waveform

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, 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 a 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 increase measurement 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 waveforms displayed and compare them with the following.

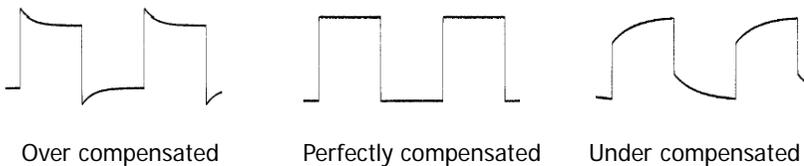


Figure 1-9 Probe Compensation

3. Use a nonmetallic driver to adjust the variable capacitor on the probe until the waveform displayed is as the "Perfectly compensated" in the figure above.

To Connect the Logic Probe

RIGOL provides a logic probe for the MSO4000 series digital oscilloscope. For detailed technical information of the logic probe, please refer to the corresponding logic probe User's Guide.

Connect the logic probe single head to the digital channel input connector **[LOGIC DO-D15]** at the front panel of the MSO4000 series digital oscilloscope. Note: connect the logic probe adaptor to the corresponding branch header (called channel group) before connecting the logic probe to the device under test.

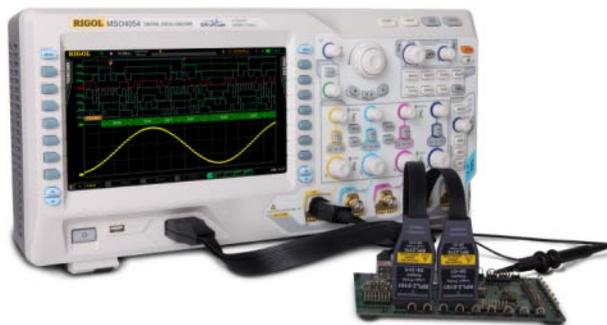


Figure 1-10 To Connect the Logic Probe

Front Panel Overview

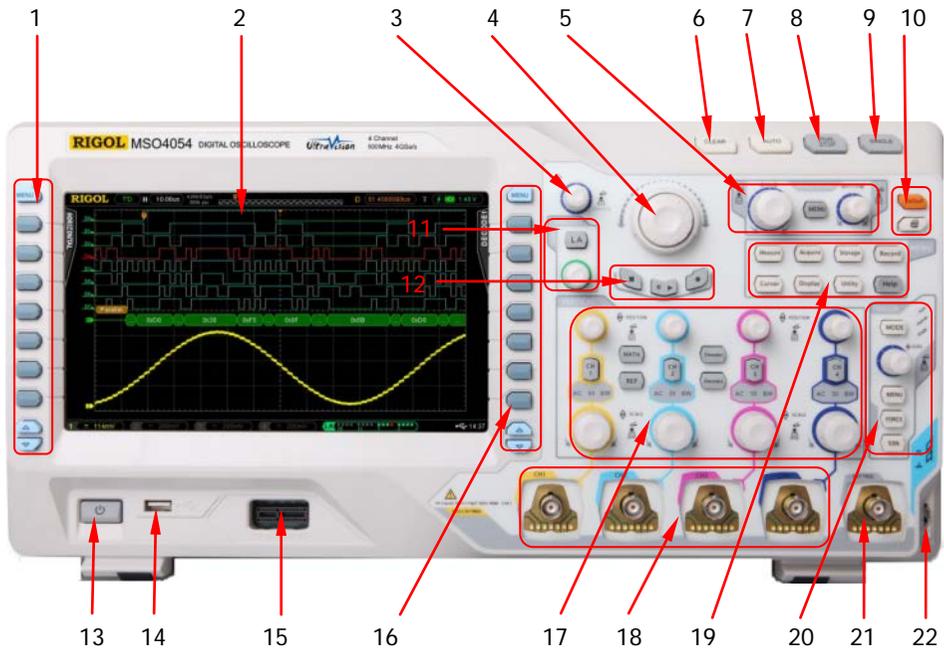


Figure 1-11 Front Panel Overview

Table 1-1 Front Panel Description

No.	Description	No.	Description
1	Menu	12	Record
2	LCD	13	Power Key
3	Multi-function Knob	14	USB HOST
4	Navigation Knob	15	Digital Channel Input Terminal
5	HORIZONTAL	16	Function Setting Menu Softkeys
6	CLEAR	17	VERTICAL
7	AUTO	18	Analog Channel Input Terminals
8	RUN/STOP	19	Function Menu Keys
9	SINGLE	20	TRIGGER
10	Default&Print	21	EXT TRIG Input Terminal
11	LOGIC ANALYZER	22	Probe Compensation Signal Output Terminal/Ground Terminal

Rear Panel Overview

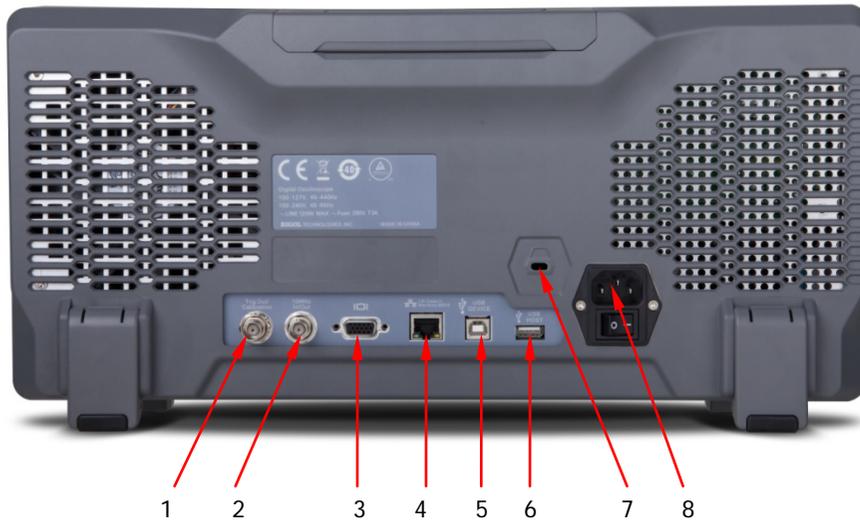


Figure 1-12 Rear Panel Overview

1. Trig Out/Calibration

Various kinds of signals can be output from this connector (press **Utility** → **AuxOutput** to select the desired output type).

- 1) **TrigOut:** the oscilloscope sources a signal which has a frequency proportional to the trigger rate of the instrument. Connect the signal to the waveform display device and measure the frequency of the signal, the measurement result is the same as the current capture rate.
- 2) **Fast:** the oscilloscope outputs a fast edge signal with 500 ps rise time which can be used in the self-calibration of the oscilloscope.
- 3) **GND:** the oscilloscope ground voltage level.
- 4) **PassFail:** the oscilloscope outputs a TTL high level when a failed waveform is detected and low level at the beginning of the next waveform. This signal can be transmitted to other control system for easy view of the test result.

2. Reference Clock

Provides a more precise sample clock signal for the oscilloscope. It can also

synchronize two or more oscilloscope clocks.

3. Video Output

Through this interface, the oscilloscope can be connected to an external monitor for clearer waveform display. Note that the display of the oscilloscope is still operational when connected to an external monitor.

4. LAN

Through this interface, the oscilloscope can be connected to the local area network for remote control.

5. USB DEVICE

Through this interface, the oscilloscope can be connected to printer or PC to print data waveforms or to control the oscilloscope using remote PC software.

6. USB HOST

Through this interface, the oscilloscope can be connected to printer or USB storage device to print waveform data or store waveform files. GPIB interface communication can be realized using the USB-GPIB interface converter provided by **RIGOL**. Note that there is also a USB HOST interface at the front panel.

7. Lock Hole

Use a security lock (available separately) to lock the oscilloscope to a fixed location.

8. AC Input/Switch

AC power input terminal. The power supply requirements of this oscilloscope are 100-127 V, 45-440 Hz and 100-240 V, 45-65 Hz. Please connect the oscilloscope to AC power using the power cord provided with the accessories. Then, turn on the AC power switch to energize the oscilloscope. Press the power key at the front panel to start the oscilloscope.

Front Panel Function Overview

VERTICAL

The MSO4000/DS4000 provides independent vertical control systems for each of the four analog input channels.



- **CH1, CH2, CH3, CH4**: analog input channels. The four channels are marked by different colors which are also used to mark the corresponding waveforms on the screen and the channel input connectors. Press any key to open the corresponding channel menu and press again to turn off the channel.
AC: illuminated when AC channel coupling is enabled.
50: illuminated when the channel input impedance is 50Ω.
BW: illuminated when bandwidth limit is enabled.
- **MATH**: press this key to open the math operation menu under which (add, subtract, multiply, divide) operations, FFT operation, logic operation and advanced operation are provided.
- **REF**: press this key to enable the reference waveform function to compare the waveform actually tested with the reference waveform.
- **VERTICAL**  **POSITION**: vertical position knobs. During the modification, the position label at the lower left corner of the screen would change accordingly. Turn clockwise to increase the position and counterclockwise to decrease. Press down the knob to quickly reset the vertical position.
- **VERTICAL**  **SCALE**: vertical scale knobs. During the modification, the scale label at the bottom of the screen would change accordingly. Turn clockwise to

decrease the scale with a step of 5-2-1 and turn counterclockwise to increase with a step of 1-2-5. Press down the knob to switch the vertical scale adjustment mode between "Coarse" and "Fine".

- **Decode1** and **Decode2**: decoding function keys. Press the corresponding key to open the decoding function menu. MSO4000/DS4000 supports parallel decoding and protocol decoding (for more details, refer to "[Protocol Decoding](#)").

HORIZONTAL



- **MENU**: press this key to open the horizontal control menu with which users can turn on or off the delayed sweep function, switch between different time base modes, switch between "Coarse" and "Fine" adjustments 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. During the modification, waveforms of all the channels will be displayed in expanded or compressed mode and the time base message (such as **H < 200.0us**) at the upper side of the screen would change accordingly. Press down the knob to quickly switch to delayed sweep state.
- **HORIZONTAL POSITION**: modify the trigger position. The trigger point would move 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 left or right and the trigger position message (such as **D < 452.000000us**) at the upper-right corner of the screen would change accordingly. Press down the knob to quickly reset 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 state backlight of the current trigger mode would be illuminated.

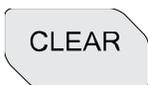
TRIGGER LEVEL: modify the trigger level. Turn clockwise to increase the level and turn counterclockwise to reduce. During the modification, the trigger level line moves up and down and the value in the trigger level message box (such as **Trig Level:340mV**) at the lower-left corner of the screen would change accordingly. Press down the knob to quickly reset the trigger level to zero point.

MENU: press this key to open 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.

CLEAR



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

AUTO



Press this key to enable the waveform auto setting function. The oscilloscope will automatically adjust the vertical scale, horizontal time base and trigger mode according to the input signal to realize optimum waveform display. Besides, the quick parameter measurement function is also provided (for more details, 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, "Can't detect any signal" would be displayed after pressing this key and the quick parameter measurement menu might not be displayed.

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.

Multi-function Knob



This knob can be used to adjust the value of the parameters, adjust the waveform brightness and select the desired menu (the backlight turns on during operation).

- During parameter input, turn the knob clockwise to increase the parameter and counterclockwise to decrease.
- When the menu is hidden, rotate the knob to adjust the waveform brightness. The adjustable range is from 0% to 100%. Turn the knob clockwise to increase the waveform brightness and counterclockwise to decrease. Press down the knob to reset the waveform brightness to 50%. Users can also press **Display** → **WaveIntensity** and then rotate the knob to adjust the waveform brightness.
- During menu operation, after pressing a menu softkey, rotate the knob to switch to the desired sub-menu under that menu and press down the knob to select that sub-menu. In addition, the knob can also be used to modify parameters and input filenames.

Navigation Knob



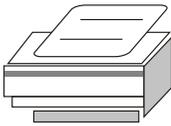
This knob provides quick Adjust/Locate function for numerical parameters with relatively larger settable ranges. Turn the knob clockwise (counterclockwise) to increase (reduce) the value. Note that the inner knob is used for fine adjustment and the outer knob for coarse adjustment. The larger the rotation amplitude of the outer knob is, the faster the change speed of the figure will be. For example, this knob can be used to quickly locate the waveform frame (**Current Frame**) to be played back in waveform playback. Similar menus include trigger holdoff, pulse width setting, slope time etc.

Default



Press this key and then press **OK** to restore the oscilloscope to its default configuration.

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 and the printer is in idle state, press this key to execute print function. If no printer but a USB storage device is currently connected, press this key to save the screen to the USB storage device in ".bmp" format. When printer and USB storage device are connected at the same time, the printer enjoys higher priority.

Function Keys



- **Measure**: press this key to enter the measurement setting menu.
- **Acquire**: press this key to enter the sample setting menu. Here, you can set the acquisition mode, the memory depth, and the antialiasing function of the oscilloscope.
- **Storage**: press this key to enter the file store and recall interface. The file types available include traces, waveforms, setups, picture and CSV. This instrument supports internal/external storage as well as disk management.
- **Cursor**: press this key to enter the cursor measurement menu. The oscilloscope provides three cursor modes (manual, track and auto).

- **Display**: press this key to enter the display setting menu to set the display type, persistence time, wave intensity, grid type, grid brightness and menu display time of the waveform.
- **Utility**: 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. The oscilloscope also supports some advanced functions such as pass/fail test and print setting.
- **Record**: press this key to enter the waveform record menu to set the waveform record and open (record constant on) mode.
- **Help**: press this key to open the help interface. For detailed information, please refer to the introduction in "[To Use the Built-in Help System](#)".

Record



Stop Play/Pause Record

Play/Pause: in stop or pause state, press this key to play the recorded or paused 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. Besides, the backlight will also be illuminated when the open (record constant on) mode is enabled.

LOGIC ANALYZER



- **LA**: press this key to enter the logic analyzer setting interface. You can enable or disable groups of channels or a single channel, change the display size and the logic threshold of the digital channels and group the 16 digital channels and display them as a bus. You can also set a label for each digital channel.
- **LA** : this knob can be used to reposition the digital channels. Press this knob to switch the digital channels currently enabled. The selected number of the digital channel (from small to large) will be highlighted in red. Rotate the knob to move the digital channel to the desired position and press this knob again to fix this channel.

User Interface

The MSO4000/DS4000 oscilloscope features 9 inch, WVGA (800*480) 160,000 color TFT LCD. The display also features a 14-grid ultra-wide screen view that enables you to observe "longer" waveforms.

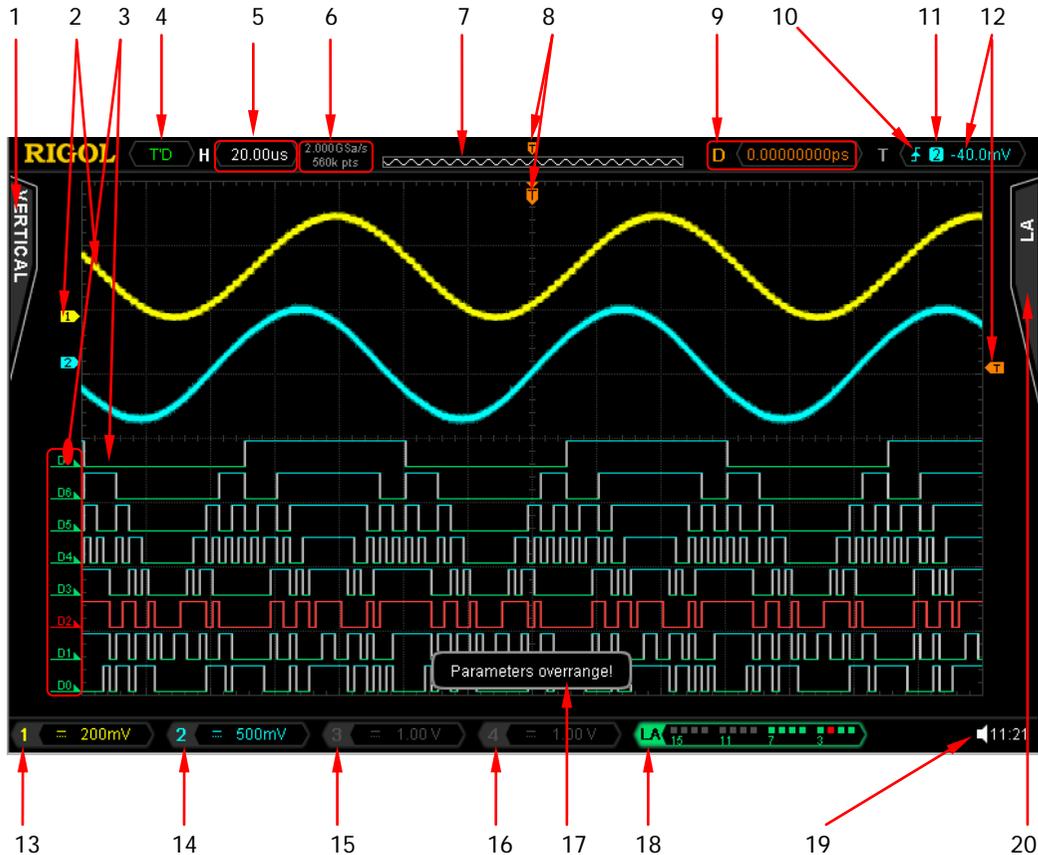


Figure 1-13 User Interface

1. 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. Analog Channel Label/Waveform

Each analog channel is color coded to match the input. For example, CH1 is yellow and CH2 is light blue.

3. Digital Channel Label/Waveform

The logic high levels of the digital waveforms are displayed in blue, the logic low levels are displayed in green which complies with the color of the digital channel label and the edges are displayed in white. Both the label and the waveform of the digital channel currently selected are displayed in red.

4. Status

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

5. Horizontal Time Base

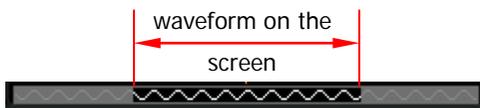
- Represent the time per grid on the horizontal axis on the screen.
- Use **HORIZONTAL**  **SCALE** to modify this parameter. The range available is from 1.000 ns to 1000 s.

6. Sample Rate/Memory Depth

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

7. Waveform Memory

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



8. Trigger Position

Displays the trigger position of the waveform.

9. Horizontal Position

Use **HORIZONTAL**  **POSITION** to modify this parameter. Press down the knob to automatically set the parameter to 0.

10. Trigger Type

Displays the currently selected trigger type and trigger condition setting. A different label is displayed when a different trigger type is selected.

For example,  represents triggering on the rising edge in “Edge” trigger.

11. Trigger Source

Displays the trigger source (CH1 to CH4, EXT, EXT/5, AC Line or D0-D15) currently selected. A different label is displayed when a different trigger source is selected and the color of the trigger parameter area will change accordingly.

For example,  denotes that CH1 is selected as the trigger source.

12. Trigger Level/Threshold

- The trigger level label  will be displayed at the right of the screen when the trigger source is set to CH1 to CH4 and the trigger level value is displayed at the upper-right corner of the screen.

Note: In runt trigger and slope trigger, there are two trigger level labels ( and ). For detailed information, refer to “Vertical Window” on page [5-14](#) and [5-18](#).

- When using **TRIGGER**  **LEVEL** to modify the trigger level, the trigger level value will change with the up and down of .
- There is no trigger level label displayed when the trigger source is set to D0-D15 and the trigger threshold is displayed at the upper-right corner of the screen.

13. CH1 Vertical Scale

- Displays the voltage value per grid of the CH1 waveform.
- Use **VERTICAL**  **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**).

14. CH2 Vertical Scale

- Displays the voltage value per grid of the CH2 waveform.
- Use **VERTICAL**  **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**).

15. CH3 Vertical Scale

- Displays the voltage value per grid of the CH3 waveform.
- Use **VERTICAL**  **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. CH4 Vertical Scale

- Displays the voltage value per grid of the CH4 waveform.
- Use **VERTICAL**  **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**).

17. Message Box

Displays prompt messages.

18. Digital Channel Status Area

Displays the current state of the 16 digital channels (D0 to D15 from right to left). The digital channels currently enabled are displayed in green and the digital channel currently selected is highlighted in red. Any digital channel that is turned off will be grayed out in the Digital Channel Status Area.

19. Notification Area

Displays system time, sound 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 **Utility** → **System** → **System Time** to set the time in the following format:

yyyy-mm-dd hh:mm:ss
(year-month-date hour:minute:second)

- Sound Icon: when sound is enabled,  is displayed. Press **Utility** → **Sound** to turn the sound on or off.
- USB Storage Device Icon: when the oscilloscope detects a USB storage device,  is displayed.

20. Operation Menu

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



Denotes that  at the front panel can be used to select parameter items. The backlight of  turns on when the parameter selection is valid.



Denotes that  can be used to modify a parameter value. The backlight of  turns on when the parameter input is valid.



Denotes that you can use the navigation knob to quickly adjust/locate parameters.



Denotes that users can rotate and then press down  to select the desired parameter.



Denotes that the current menu has several options.



Denotes 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:



Denotes that you can open the next page menu.



Denotes that you can return to the previous page menu.

To Use the Security Lock

If needed, you can use a security lock (available separately) to secure 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.

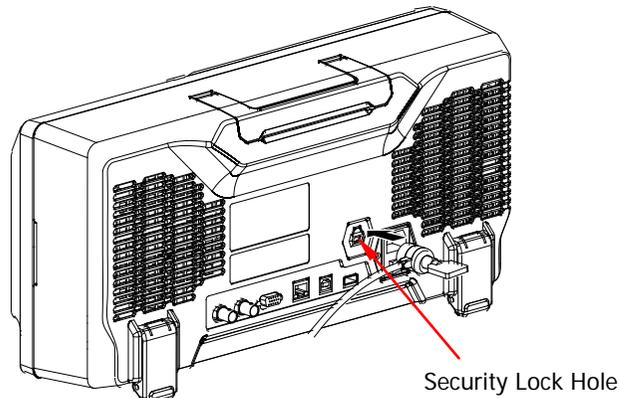


Figure 1-14 To Use the Security Lock

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

To Use the 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" mode for selection. The right is "Help Display Area".



Figure 1-15 Help Information

Button:

Default mode. In this mode, you can press the keys (except the power key , the knob  and menu page up/down key ) at the front panel directly to get the corresponding help information of that key in the "Help Display Area".

Use  to select "To Index" and then press down the knob to switch to **Index** mode.

Index:

In this mode, use  to select the desired item (for example, "Band Width"). The item currently selected is displayed with brown shading. Press down the knob to get the corresponding help information in the "Help Display Area".

Use  to select "To Button" and then press down the knob to switch to **Button** mode.

2 To Set the Vertical System

The contents of this chapter:

- [To Enable the Analog Channel](#)
- [Channel Coupling](#)
- [Bandwidth Limit](#)
- [Probe](#)
- [Input Impedance](#)
- [Waveform Invert](#)
- [Vertical Scale](#)
- [Vertical Expansion](#)
- [Amplitude Unit](#)
- [Channel Label](#)
- [Delay Calibration](#)

To Enable the Analog Channel

The MSO4000/DS4000 provides four analog input channels (CH1 to CH4) and provides independent vertical control system for each channel. As the vertical system setting methods of the four channels are completely the same, this chapter uses CH1 as the example.

Connect a signal to the channel connector of 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 has been enabled, the character "AC", "50" or "BW" will also be illuminated. Note that the on/off state of the key light of "AC", "50" or "BW" is not controlled 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 undesired signals. 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" below **CH1** at the front panel will be illuminated. You can also press **Coupling** continuously to switch the coupling mode.

Bandwidth Limit

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

- When bandwidth limit is disabled, the high frequency components of the signal under test can pass the channel.
- Enable bandwidth limit and limit the bandwidth to 20 MHz, 100 MHz or 200 MHz, the high frequency components that exceed 20 MHz, 100 MHz or 200 MHz are attenuated.

Press **CH1** → **BW Limit** and use  to enable or disable the bandwidth limit (the default is OFF). When the bandwidth limit (20 MHz, 100 MHz or 200 MHz) is enabled, 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 probes and active differential probes. It can automatically identify the type of the probe currently connected and the probe ratio. Press **CH1** → **Probe** to open the probe operation menu.

1. ProbeType

Read the type of the probe currently connected as “Nor-Probe” or “DiffProbe”.

Note that when a 50 Ω “DiffProbe” is used, the [Input Impedance](#) of the channel is set to “50 Ω ” automatically.

- Normal Probe: such as **RIGOL** RP3500A.
- Differential Probe: such as **RIGOL** RP7150.

2. Ratio

For some “Nor-Probe”s, the oscilloscope can identify the probe ratio automatically. If not, press this softkey to select the corresponding probe ratio. The values available are listed in the table below.

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 connected, 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 information about the probe, such as the manufacturer, model, serial number and the date of last calibration.

6. Bias Voltage

The oscilloscope provides a bias voltage for differential probes. This bias voltage is used to adjust the input dynamic range of the probe amplifier to an appropriate range to ensure accurate measurement 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 between the oscilloscope and the circuit under test, the oscilloscope provides two input impedance modes: 1 M Ω (default) and 50 Ω .

- 1 M Ω : 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 with 50 Ω output impedance.

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

Waveform Invert

When waveform invert is enabled, the waveform display rotates 180 degree relative to the ground potential. When waveform invert is disabled, the waveform display is normal. Press **CH1** → **Invert** to enable or disable waveform invert.

Vertical Scale

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

Press **CH1** → **Volts/Div** to select the desired mode. Rotate **VERTICAL**  **SCALE** to adjust the vertical scale (clockwise to reduce the scale and counterclockwise to increase).

The scale information (such as ) 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 ratio is 1X and the vertical scale range is from 1 mV/div to 5 V/div.

- Coarse adjustment (take counterclockwise as an example): set the vertical scale in 1-2-5 step namely 1 mV/div, 2 mV/div, 5 mV/div, 10 mV/div...5 V/div.
- Fine adjustment: enables fine adjustment to the vertical scale. 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 signal details.

Note: You can also press **VERTICAL**  **SCALE** to quickly switch between “Coarse” 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 signal vertically around the center of the screen or the ground point of the signal.

Press **Utility** → **System** → **VerticalExp** to select “Center” or “Ground” and the default is “Ground”.

- Center: when the vertical scale is modified, the waveform will expand or compress around the center of the screen.
- Ground: when the vertical scale is modified, 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 displayed 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 (such as 1) by default and the length of the label can not exceed 4 characters.

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

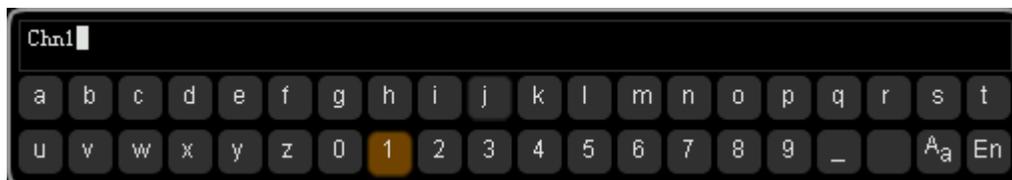


For example, change "1" to "Chn1".

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

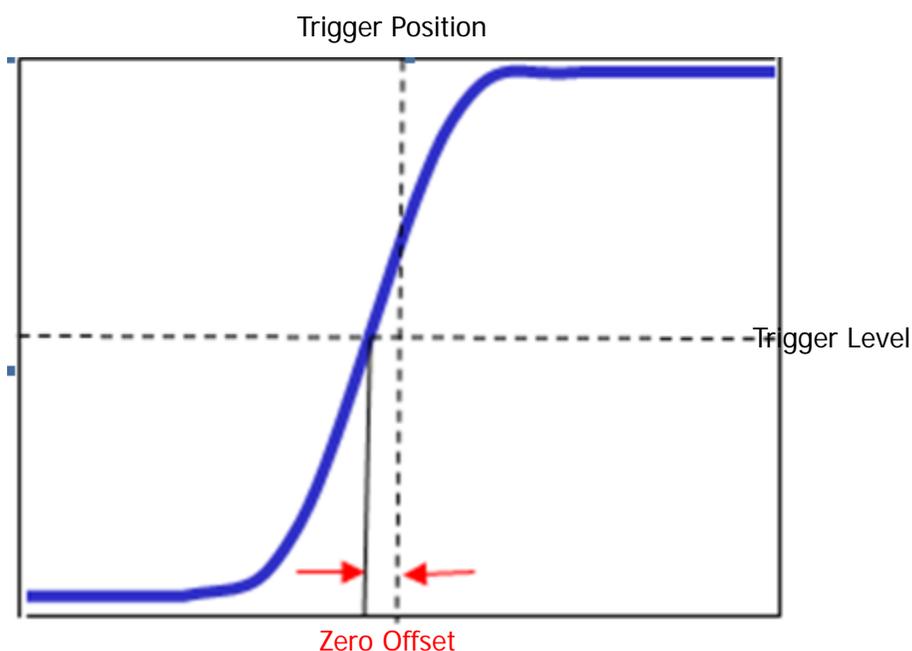
To modify or delete the input character, press **Name** to select the "Name Input Area" and use ↻ to select the character to be modified or deleted. Enter the desired character or press **Delete** to delete the character selected.

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



Delay Calibration

When using an oscilloscope for actual measurements, the transmission delay of the probe cable may bring greater error (zero offset). The MSO4000/DS4000 allows users to set a delay time to calibrate the zero offset of the corresponding channel. Zero offset is defined as the offset of the crossing point of the waveform and trigger level line relative to the trigger position, as shown below.



Press **CH1** → **Delay-Cal** and use  to select the desired delay time. The range available is from -100 ns to 100 ns.

Note: This parameter is related to the current horizontal time base. For example, when the horizontal time base is 5 μ s, the step of the delay time setting is 100 ns. when the horizontal time base is 1 μ s, the step of the delay time setting is 20 ns. when the horizontal time base is 500 ns, the step of the delay time setting is 10 ns.

3 To Set the Horizontal System

The contents of this chapter:

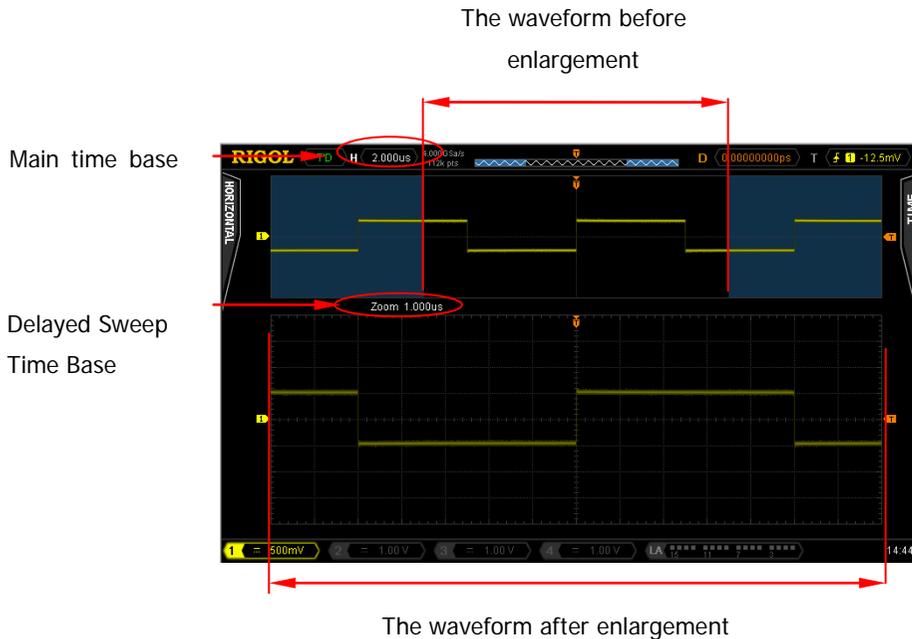
- [Delayed Sweep](#)
- [Time Base Mode](#)
- [Horizontal Scale](#)
- [Horizontal Reference](#)

Delayed Sweep

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

Press **MENU** in the horizontal control area (HORIZONTAL) and press **Delayed** to enable or disable delayed sweep. Note that 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:

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 that compared to the main time base, the delayed time base has increased waveform resolution (as shown in the figure above).

Note: The delayed time base should be less than or equal to the main time base.

Tip

When the current channel is CH1 to CH4, you can also press down **HORIZONTAL** **SCALE** (delayed sweep shortcut key) to directly switch to delayed sweep mode.

Time Base Mode

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

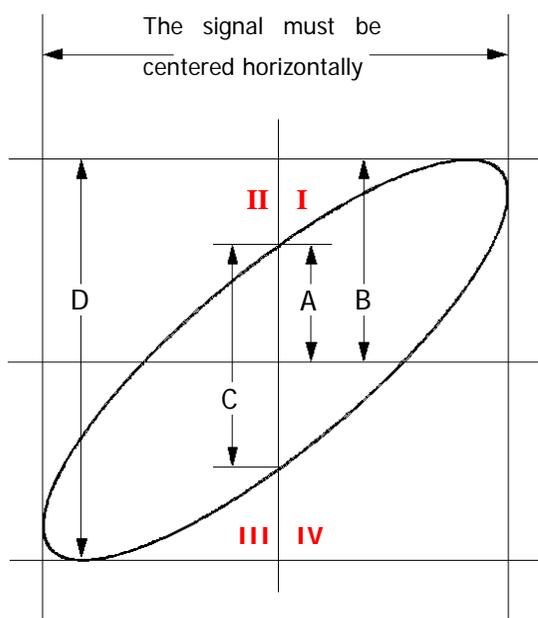
Y-T Mode

This mode is the main time base mode and is applicable to CH1 to CH4. In this mode, the Y axis represents voltage and the X axis represents time. Note that only when this mode is enabled can [Delayed Sweep](#) be turned on.

X-Y Mode

In this mode, the oscilloscope automatically turns on all four channels (CH1 to CH4) and the screen is divided into two coordinate areas. X1 and Y1 track the voltages of CH2 and CH1 and X2 and Y2 track the voltages of CH4 and CH3.

The phase deviation between two signals with the same frequency can be easily measured via the Lissajous method. The figure below shows the measurement schematic diagram of the phase deviation.



According to $\sin\theta = A/B$ or C/D (wherein, 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:

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

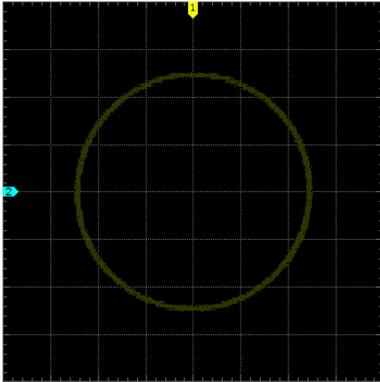
If the principal axis of the ellipse is within quadrant I and III, the phase deviation angle obtained should be within quadrant I and IV, namely within $(0 \text{ to } \pi/2)$ or $(3\pi/2 \text{ to } 2\pi)$. If the principal axis of the ellipse is within quadrant II and IV, the phase deviation angle obtained should be within quadrant II and III, namely within $(\pi/2 \text{ to } \pi)$ or $(\pi \text{ to } 3\pi/2)$.

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

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** and enable X-Y mode. Then, press **XY1** to select "On".
3. Rotate **VERTICAL**  **POSITION** of CH1 and CH2 to display the signals at the center of the screen and rotate **VERTICAL**  **SCALE** to make the signals easy to observe. At this point, the circle as shown in the figure below should be displayed.



4. As shown in the figure above, the distances from the crossing points of axis and the circle to the origin of the coordinates are approximately equal. Thus, the phase deviation angle $\theta = \pm \arcsin 1 = 90^\circ$.

Note:

- In Y-T mode, the oscilloscope could use any sample rate (within the guaranteed range) to capture the waveform. The maximum sample rate of X-Y mode is 2.0 GSa/s. Generally, reducing the sample rate can improve the displayed Lissajous figure.

- When X-Y mode is enabled, [Delayed Sweep](#) will be disabled automatically.
- The following functions are not available in X-Y mode:
Auto measure, cursor measure, math operation, reference waveform, delayed sweep, vector display, **HORIZONTAL  POSITION**, trigger control, memory depth, acquisition mode, Pass/Fail test and waveform record.

Method 2: Use the shortcut measurement function

Please refer to “Phase A→B $\frac{f}{t}$ ” and “Phase A→B $\frac{t}{f}$ ” measurement functions of “Delay and Phase” on page [6-16](#).

Roll Mode

In this mode, the waveform scrolls from the right to the left, much like a strip chart. This is useful for viewing slower signals as the signal is “drawn out” on the display as time elapses. The waveform horizontal position and trigger controls are not available. The range of horizontal scale adjustment is from 200.0 ms to 50.00 s.

Note: When the Roll mode is enabled, “[Delayed Sweep](#)”, “[Protocol Decoding](#)”, “[Pass/Fail Test](#)”, “[Measurement Range](#)”, “[Waveform Record](#)”, “[To Set the Persistence Time](#)”, “[To Trigger the Oscilloscope](#)” and “[Logic Analyzer](#)” 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 “slow sweep” mode in which the instrument first acquires the data to the left of the trigger point and then waits for trigger event. After the trigger occurs, the instrument continues to finish the waveform at the right of the trigger point. When the slow sweep mode is used to observe low frequency signal, it is recommended that the [Channel Coupling](#) is set to “DC”.

Horizontal Scale

Similar to "[Vertical Scale](#)", the horizontal scale can be adjusted in "Coarse" or "Fine" mode.

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 information (such as **H 200.0us**) at the upper left corner of the screen will change accordingly during the adjustment. The range of horizontal scale adjustment is from 1.000 ns to 1000 s.

- Coarse (take counterclockwise as an example): sets the horizontal scale in 1-2-5 step namely 1 ns, 2 ns, 5 ns...1000 ns.
- Fine: higher resolution steps allow fine adjustment of horizontal scale.

Horizontal Reference

The horizontal reference is the reference position according to which the screen waveform expands and compresses horizontally when adjusting **HORIZONTAL SCALE**. In Y-T mode (this function is not available in X-Y mode and Roll 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. In the horizontal direction, the screen can display a maximum of 700 points with the leftmost as 350 and the rightmost as -350.

4 To Set the Sample System

The contents of this chapter:

- [Acquisition Mode](#)
- [Sample Mode](#)
- [Sample Rate](#)
- [LA Sample Rate](#)
- [Memory Depth](#)
- [Antialiasing](#)

Acquisition Mode

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

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

Normal

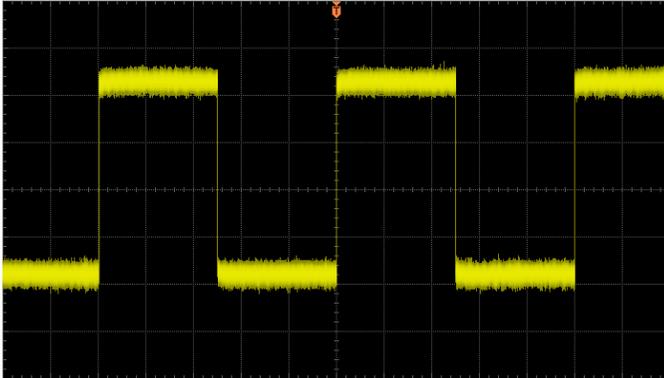
In this mode, the oscilloscope samples the signal at equal time interval to rebuild the waveform. For most of the waveforms, the best display effect can be obtained using this mode.

Average

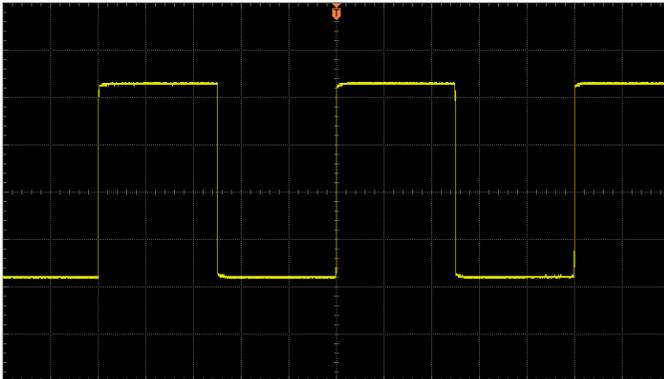
In this mode, the oscilloscope averages the waveforms from multiple samples to reduce the random noise of the input signal and improve the vertical resolution. More averages can lower the noise and may increase the vertical resolution but it will slow the overall response of the displayed waveform to input 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 signal within the sample interval to get the envelope or the narrow pulse of the signal that might be lost. In this mode, signal confusion can be prevented but the noise displayed would be larger.

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

High Resolution

This mode uses an over-sample technique to average the neighboring points of the sample waveform. This reduces the random noise on the input signal and generates a much smoother waveform on the screen. This is generally used when the sample rate of the digital converter is higher than the storage rate of the acquisition memory.

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

Sample Mode

This oscilloscope only supports real-time sampling. In this mode, the oscilloscope samples and displays waveforms within a trigger event. The maximum real-time sample rate of MSO4000/DS4000 is 4 GSa/s and the current sample rate is displayed under the **Sa Rate** menu.

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

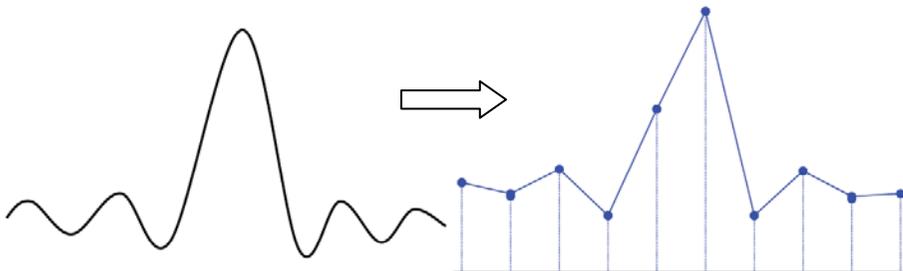
Sample refers to the process that the oscilloscope converts analog signals to digital signals at a certain time interval and stores the data in order. Sample rate is the reciprocal of the time interval.

$$\text{Sample Rate} = 1/\Delta t$$

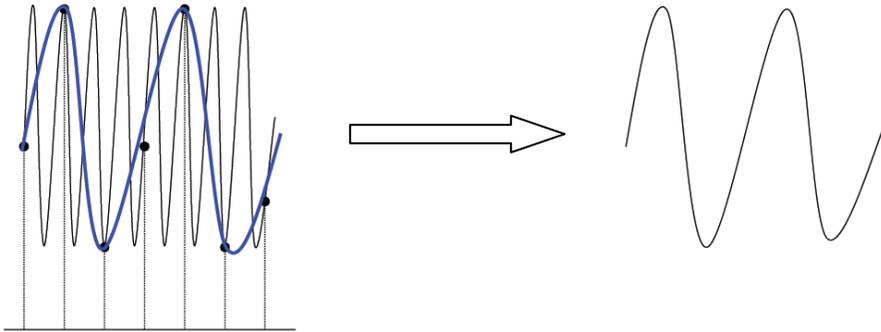
The analog channels sample rate of this oscilloscope is up to 4 GSa/s. Note that the sample rate is displayed in the status bar at the upper side of the screen and in the **Sa Rate** menu and can be changed by adjusting the horizontal time base (s/div) through **HORIZONTAL**  **SCALE** or modifying the "[Memory Depth](#)".

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

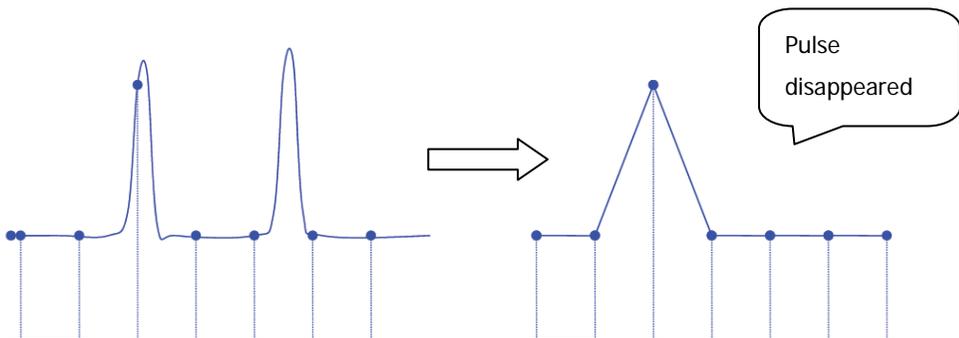
- 1. Waveform Distortion:** when the sample rate is too low, some waveform details are lost and the waveform displayed is rather different from the actual signal.



2. **Waveform Confusion:** when the sample rate is lower than twice the actual signal frequency (Nyquist Frequency), the frequency of the waveform rebuilt from 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 from the sample data does not reflect all the actual signal information.



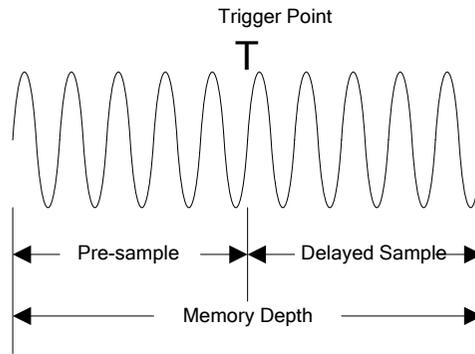
LA Sample Rate

LA sampling refers to the process that the oscilloscope samples the compared digital signal at a certain time interval. LA sample rate is the reciprocal of the interval. For example, the oscilloscope with 1 GHz LA sample rate samples digital signals every 1 ns. The digital signal mentioned here refers to the signal generated by comparing the input signal with the threshold specified by users. The input signal is judged as logic high level when its amplitude greater than the threshold and logic low level when its amplitude is lower than the threshold.

The digital channels sample rate of this oscilloscope is up to 1 GSa/s (per channel). Note that the LA sample rate is displayed in the **La SaRate** menu and can be changed by adjusting the horizontal time base (s/div) through **HORIZONTAL**  **SCALE** or modifying the "[Memory Depth](#)".

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 sample memory. MSO4000/DS4000 provides up to 140 M points memory depth.



The relation of memory depth, sample rate and waveform length fulfills the equation below:

$$\text{Memory depth} = \text{sample rate (Sa/s)} \times \text{waveform length (s/div} \times \text{div)}$$

Thus, higher memory depth can ensure higher sample rate.

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

When a single channel is enabled, 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 dual channels are enabled (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 dedicated display algorithm is used to minimize the possibility of aliasing.

Press **Acquire** → **Anti_aliasing** to enable or disable 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

The trigger system of an oscilloscope allows users to tailor data collection and isolate particular events by providing a flexible control system to “select” certain events (rising edge, falling edge, etc.). You can set a 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 then displays the captured waveforms on the screen. The oscilloscope samples waveform data continuously, but only a stable trigger can guarantee stable waveform displays. The trigger circuit guarantees that each sweep or sample starts from the trigger condition defined by the user. Each sweep and sample are synchronous and the waveforms acquired overlap to display the stable waveform.

The 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 can help you to focus on the waveform details of interest.

The contents of this chapter:

- [Trigger Source](#)
- [Trigger Mode](#)
- [Trigger Coupling](#)
- [Trigger Holdoff](#)
- [Noise Rejection](#)
- [Trigger Type](#)
- [Trigger Output 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 CH1 to CH4, the **[EXT TRIG]** and **[LOGIC D0-D15]** connectors as well as the AC Line can all be used as trigger source.

Analog channel input:

Signals input from analog channels CH1 to CH4 can all be used as the trigger source. No matter whether the input of the channel selected is enabled, the channel can work normally.

External trigger input:

External trigger source can be used to trigger on the fifth channel when all the four channels are sampling data. The trigger signal (such as external clock and signal of the circuit to be tested) should be connected to **EXT** or **EXT/5** trigger source via the **[EXT TRIG]** connector. You can set the trigger condition within the range of trigger level (-0.8 V to +0.8 V).

Digital channel input:

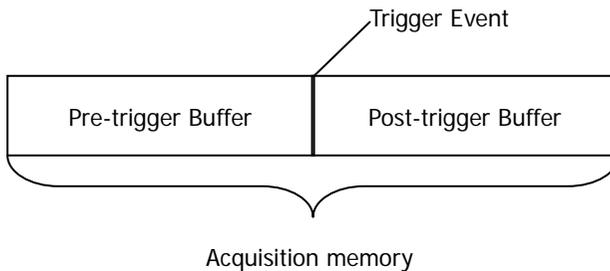
Signals input from digital channels D0 to D15 can all be used as the trigger source. No matter whether the input of the channel selected is enabled, the channel can work normally.

AC line:

The trigger signal is obtained from the AC power input of the oscilloscope. This kind of signals can be used to display the relationship between the 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

The trigger mode affects the way in which the oscilloscope searches for the trigger. The following is the schematic diagram of the acquisition memory. As shown in the figure below, the position of the trigger event in the acquisition memory is determined by the reference time point and the delay setting.



Pre-trigger/Delayed trigger:

This area is where you can view data acquired before/after the trigger event. The trigger position is usually at the horizontal center of the screen. In full-screen display, seven-grid pre-trigger and delayed trigger information are displayed respectively. You can adjust the horizontal position through **HORIZONTAL POSITION** to view more pre-trigger information, through which the signal information before/after the trigger (such as capture the glitch generated by the circuit and analyze the pre-trigger data to find out the reasons for glitch) can be obtained.

Press **MODE** in the trigger control area (TRIGGER) at the front panel or press **MENU** → **Sweep** to select the desired trigger mode. The corresponding status light of the mode currently selected turns on.

Auto:

This trigger mode automatically triggers regardless of whether the trigger condition is met. A horizontal line is displayed when there is no input signal.

In this mode, the oscilloscope operates by first filling the pre-trigger buffer. It starts searching for a trigger after the pre-trigger buffer is filled and continues to flow data through this buffer while it searches for the trigger. While searching for the trigger, the oscilloscope overflows the pre-trigger buffer and the first data put into the buffer is first pushed out (FIFO). When a trigger is found, the pre-trigger buffer contains

the data acquired just before the trigger. If no trigger is found, the oscilloscope will force a trigger. If the forced trigger is invalid, the oscilloscope still displays waveform but the waveform will not be stable. If the forced trigger is valid, the oscilloscope displays a stable waveform.

This trigger mode is applicable to low-repetitive-rate signals and unknown signal levels. To display DC signals, you must use auto trigger mode.

Note: When the horizontal time base is set to 50 ms/div or greater, this trigger mode allows the absence of trigger signal.

Normal:

Displays a waveform when the trigger condition is met. Otherwise, the oscilloscope holds the original waveform and waits for the next trigger.

In this mode, the oscilloscope fills the pre-trigger buffer first and then search for a trigger while at the same time continues filling data. While searching for the trigger, the oscilloscope overflows the pre-trigger buffer and the first data put into the buffer is first pushed out (FIFO). When a trigger is found, the oscilloscope will fill the post-trigger buffer and display the acquisition memory.

Use normal trigger mode for low-repetitive-rate signals or when auto trigger is not required.

Note: In this mode, pressing **FORCE** can generate a forced trigger signal.

Single:

When 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, pressing **FORCE** can generate a forced trigger signal.

Trigger Coupling

The trigger coupling setting determines the type of signal components that will be transmitted to the trigger circuit. This is similar to the settings of "[Channel Coupling](#)" but is specific to the triggered signal only.

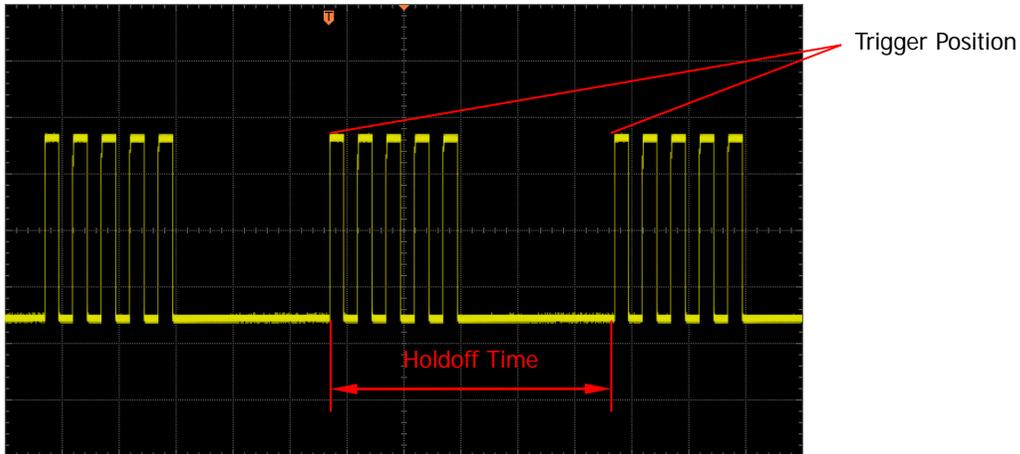
- DC: allow DC and AC components into the trigger path.
- AC: block all the DC components and attenuate signals lower than 8 Hz.
- LF reject: block the DC components and reject the low frequency components (lower than 5 kHz).
- HF reject: reject the high frequency components (higher than 50 kHz).

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 invalid in the video trigger mode.

Trigger Holdoff

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



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

Noise Rejection

Noise rejection minimizes the possibility of triggering on a noise signal reducing the trigger sensitivity. This decrease in sensitivity does require a greater amplitude waveform to trigger the oscilloscope.

Press **MENU** → **Setting** → **Noise Reject** in the trigger control area (TRIGGER) at the front panel to enable or disable noise rejection.

Trigger Type

The MSO4000/DS4000 provides various trigger functions, including various serial bus triggers.

- [Edge Trigger](#)
- [Pulse Trigger](#)
- [Runt Trigger](#)
- [Nth Edge Trigger](#)
- [Slope Trigger](#)
- [Video Trigger](#)
- [Pattern Trigger](#)
- [RS232 Trigger](#)
- [I2C Trigger](#)
- [SPI Trigger](#)
- [CAN Trigger](#)
- [FlexRay Trigger](#)
- [USB Trigger](#)

Edge Trigger

The edge trigger configures the instrument to capture data when the rising or falling edge of the input signal passes the threshold (trigger) voltage setting.

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 to 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 the channel with the signal input as the trigger source to obtain stable trigger.

Edge Type:

Press **Slope** to select the kind of edge of the input signal on which the oscilloscope triggers. 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, trigger holdoff and noise rejection) under this trigger type.

Trigger Level:

Use **TRIGGER**  **LEVEL** to modify the level. An orange trigger level line and the trigger mark " appear on the screen and move up and down with the rotation of the knob, while at the same time, the trigger level value (such as ) at

the lower left corner of the screen also changes accordingly. The trigger level line and the trigger mark will disappear in about 2 seconds if no input (rotation) is detected.

Pulse Trigger

Trigger on the positive or negative pulse with a 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 to CH4, EXT or D0-D15 as the [Trigger Source](#). The current trigger source is displayed at the upper right corner of the screen.

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

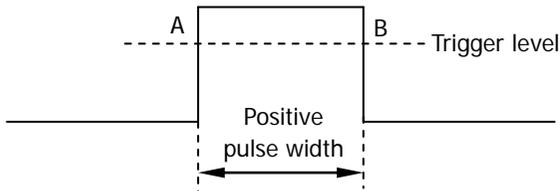
Pulse Condition:

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 Setting.
- : trigger when the positive pulse width of the input signal is lower than the specified Pulse Width Setting.
- : 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 Setting.
- : trigger when the negative pulse width of the input signal is lower than the specified Pulse Width Setting.
- : 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:

Positive pulse width is defined as the time difference between the two crossing points of the trigger level and positive pulse as shown in the figure below.



- When the **Pulse Condition** is set to , ,  or , press **Setting** and use  to input the desired value. The range available is from 4 ns to 4 s.
- When the Pulse Condition is set to  or , press **Upper Limit** and **Lower Limit** and use  to input the desired values respectively. The range of the upper limit is from 12 ns to 4 s. The range of the lower limit is from 4 ns to 3.99 s. Note that 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:

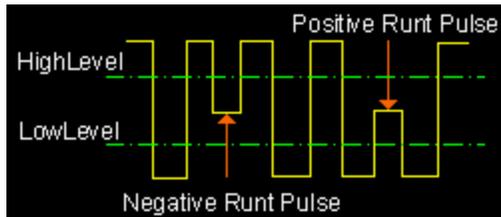
Press **Setting** to set the trigger parameters (trigger coupling, trigger holdoff and noise rejection) under this trigger type.

Trigger Level:

Use **TRIGGER**  **LEVEL** to modify the level. For details, please refer to the description of "[Trigger Level](#)" on page [5-9](#).

Runt Trigger

Runt trigger is used to trigger pulses that pass one trigger level but fail to pass the other one, as shown in the figure below.



Trigger Type:

Press **Type** to select "Runt". 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 to CH4 as the [Trigger Source](#). The current trigger source is displayed at the upper right corner of the screen.

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

Pulse Polarity:

Press **Polarity** to select the pulse polarity of runt trigger.

- : positive polarity. The instrument triggers on the positive runt pulse.
- : negative polarity. The instrument triggers on the negative runt pulse.

Qualifier:

Press **Qualifier** to set the trigger conditions of runt trigger.

- **None**: do not set the trigger condition of runt trigger.
- **>**: press **Lower Limit** to set the minimum pulse width of runt trigger. The range available is from 4 ns to 4 s.
- **<**: press **Upper Limit** to set the maximum pulse width of runt trigger. The range available is from 4 ns to 4 s.
- **<>**: press **Upper Limit** to set the maximum pulse width of runt trigger and the range is from 12 ns to 4 s. Press **Lower Limit** to set the minimum pulse width

of runt trigger and the range is from 4 ns to 3.99 s.

Vertical Window:

Press **Vertical** to select the desired vertical window type. Note that under the “Runt trigger” menu, you can press the trigger level knob continuously to switch among different vertical window types.

You can select the boundary of the trigger level to be set and then use **Trigger**  **LEVEL** to adjust the trigger level. During the adjustment, two orange trigger level lines and trigger labels (**T1** and **T2**) appear on the screen and move up and down with the rotation of the knob. At the same time, the current trigger level values are displayed at the lower left corner of the screen. The trigger level lines and trigger labels disappear after you stop rotating the knob for about 2 seconds.



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

- : only adjust the upper limit of the trigger level. During the adjustment, the “Up Level” changes accordingly and “Low Level” remains unchanged.
- : only adjust the lower limit of the trigger level. During the adjustment, the “Low Level” changes accordingly and 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 “Low Level” change accordingly.

Trigger Mode:

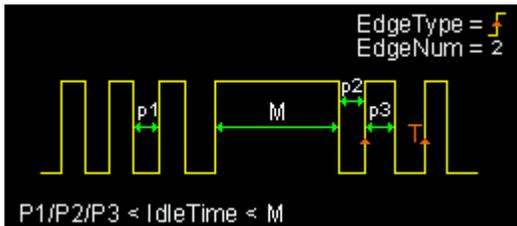
Press **Sweep** to select the **Trigger Mode** (page [5-3](#)) under this trigger type as auto, normal or single. The corresponding status light of the current trigger mode turns on.

Trigger Setting:

Press **Setting** to set the trigger parameters (trigger coupling, trigger holdoff and noise rejection) under this trigger type.

Nth Edge Trigger

Trigger on the nth edge that appears after the specified idle time, as shown in the figure below.



Trigger Type:

Press **Type** to select “Nth 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 to CH4 or D0-D15 as the [Trigger Source](#). The current trigger source is displayed at the upper right corner of the screen.

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

Edge Type:

Press **Slope** to select the edge on which the input signal triggers.

- : trigger on the rising edge of the input signal when the voltage level meets the specified trigger level.
- : trigger on the falling edge of the input signal when the voltage level meets the specified trigger level.

Idle Time:

Press **Idle** to set the idle time before the edge counting in Nth edge trigger. The range available is from 40 ns to 1 s.

Edge Number:

Press **Edge** to set the value of “N” in Nth edge trigger and the range available is from 1 to 65535.

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, trigger holdoff and noise rejection) 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 [5-9](#).

Slope Trigger

Trigger on the positive or negative slope of the 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 to CH4 as the [Trigger Source](#). The current trigger source is displayed at the upper right corner of the screen.

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

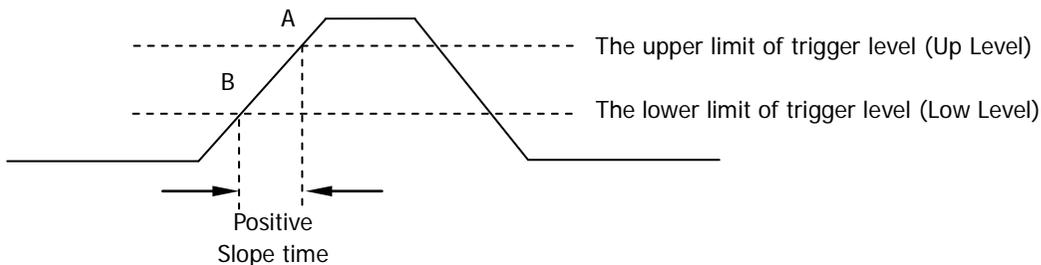
Slope Condition:

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

- : trigger when the positive slope time of the input signal is greater than the specified Time Setting.
- : trigger when the positive slope time of the input signal is lower than the specified Time Setting.
- : 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.
- : trigger when the negative slope time of the input signal is greater than the specified Time Setting.
- : trigger when the negative slope time of the input signal is lower than the specified Time Setting.
- : 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:

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** and use to input the desired values respectively. The range of time upper limit is from 20 ns to 1 s. The range of the time lower limit is from 10 ns to 999 ms. Note that the time lower limit must be lower than the upper limit.

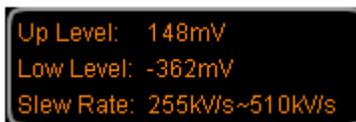
Vertical Window:

Press **Vertical** to select the desired vertical window. Note that under the “Slope” trigger menu, you can also press down the trigger level knob continuously to switch the 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 rotation of the knob. The trigger level value and the slope value are displayed at the lower left corner of the screen. The trigger level lines and trigger marks disappear in about 2 seconds after the knob has stopped rotating. When the **Slope Condition** is set to or , the current trigger level value and the slope range will be displayed at the lower left corner of the screen.



$$\text{SlewRate} = \frac{\text{UpLevel} - \text{LowLevel}}{\text{TimeSettingValue}}$$



$$\text{SlewRate} = \frac{\text{UpLevel} - \text{LowLevel}}{\text{TimeUpperLimit}} \sim \frac{\text{UpLevel} - \text{LowLevel}}{\text{TimeLowerLimit}}$$

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, "UP Level" and "Slew Rate" change accordingly but "Low Level" remains unchanged.
- : only adjust the lower limit of the trigger level. During the adjustment, "Low Level" and "Slew Rate" change accordingly but "UP Level" remains unchanged.
- : adjust the upper and lower limits of the trigger level at the same time. During the adjustment, "UP Level" and "Low Level" change accordingly but "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 coupling, trigger holdoff and noise rejection) under this trigger type.

Video Trigger

Trigger on the standard video signal field or line 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 to CH4 as the [Trigger Source](#). The current trigger source is displayed at the upper right corner of the screen.

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

Video Polarity:

Press **Polarity** to select the desired video polarity. You can select  (positive polarity) or  (negative polarity).

Sync:

Press **Sync** to select the desired sync type.

- All Lines: trigger on all the horizontal sync pulses.
- Line Num: 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 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 and noise rejection) 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 [5-9](#).

Pattern Trigger

This trigger mode will identify a trigger condition by looking for a specified pattern. This pattern is a logical “AND” combination of the channels. Each channel can have a value of high (H), low (L) or don’t care (X). A rising or falling edge can be specified for one channel included in the pattern. When an edge is specified, the oscilloscope will trigger at the edge specified if the pattern set for the other channels are true (H or L). If no edge is specified, the oscilloscope will trigger on the last edge that makes the pattern true. If all the channels in the pattern are set to “Don’t Care”, the oscilloscope will 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.



Channel Selection:

Press **Source** to select CH1 to CH4, EXT or D0 to D15 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 (as shown in the figure below) is displayed at the bottom of the screen.



- **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 channel selected to “Don’t Care”, namely this channel is not used as 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: Only one rising or falling edge can be specified in the pattern. If one edge item is currently defined and then another edge item is defined in another channel in the

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

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:

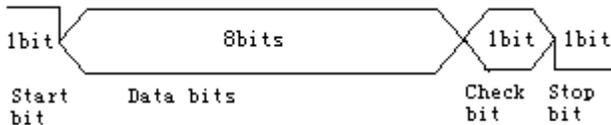
Press **Setting** to set the trigger parameters (trigger coupling, trigger holdoff and noise rejection) under this trigger type.

Trigger Level:

Use **TRIGGER**  **LEVEL** to modify the level. For details, please refer to the description of "[Trigger Level](#)" on page [5-9](#). 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 to CH4 or D0-D15 as the [Trigger 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 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", "Odd Checkout" or "Even 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 **Even-OddCheck** to select "Odd Checkout" or "Even 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 limits are 31, 63, 127 and 255 respectively.

Baud Rate:

Set the baud rate of data transmission (equal to specifying a clock frequency). Press **Baud** to set the desired baud rate to 2400 bps, 4800 bps, 9600 bps (default), 19200 bps, 38400 bps, 57600 bps, 115200 bps and user-defined. When "User" is selected, press **Setup** and use  to set a more specific value from 1 to 900000 with an adjustment step of 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:

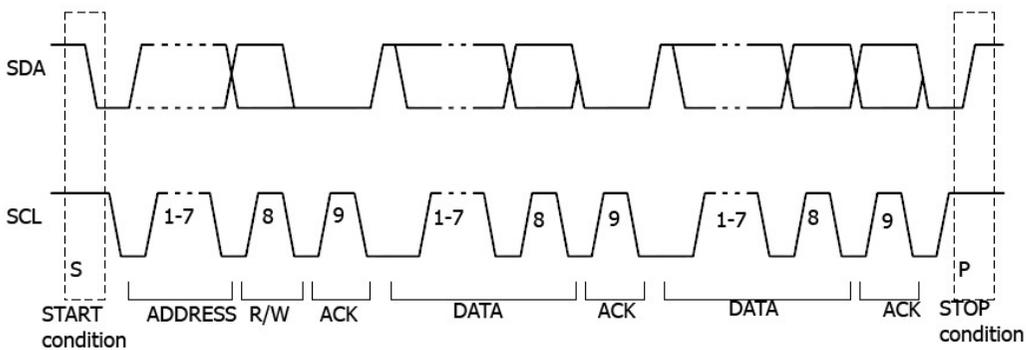
Press **Setting** to set the trigger parameters (trigger coupling, trigger holdoff and noise rejection) under this trigger type.

Trigger Level:

Use **TRIGGER**  **LEVEL** to modify the level. For details, please refer to the description of "[Trigger Level](#)" on page [5-9](#).

I2C Trigger

Trigger on the start condition, restart, stop, missing acknowledgement or on the read/write frame with specific device address and data value. In 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 to CH4 or D0-D15 can be selected as the [Trigger 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 position.

- Address: trigger on the clock (SCL) edge corresponding to the byte of 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, from 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", it 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. When this event occurs, the oscilloscope will trigger on the clock edge of the acknowledgement bit behind the data byte. After this trigger condition is selected:
 - press **Byte Length** to set the length of the data and the range is from 1 to 5.
 - press **CurrentBit** to select the desired data bit and the range is from 0 to (Byte Length×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 **Byte Length** to set the length of the data and the range is from 1 to 5.
 - press **CurrentBit** to select the desired data bit and the range is from 0 to (Byte Length×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", it 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:

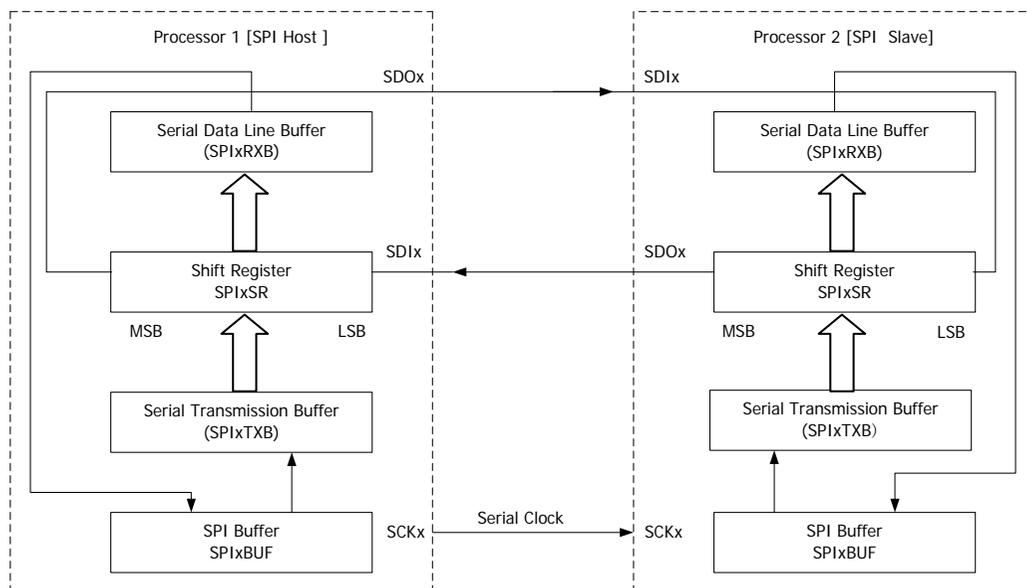
Press **Setting** to set the trigger parameters (trigger coupling, trigger holdoff and noise rejection) under this trigger type.

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 [5-9](#).

SPI Trigger

Trigger on the data pattern on the specified edge. When using the 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**, **SDA** and **CS** to specify the data sources of SCL, SDA and CS respectively. CH1 to CH4 or D0-D15 can be selected as the **Trigger 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: trigger when the CS channel is 1 at high level or low level. After this trigger

condition is selected, press **Mode** to select  (high level is 1) or  (low level is 1). Note that **Mode** is valid only when this condition is selected.

- **TimeOut**: set the minimum time that the clock (SCL) signal must be idle before the oscilloscope starts to search for the data (SDA) on which to trigger. After this trigger condition is selected, press **TimeOut** to set the timeout value and the range is from 100 ns to 1 s. Note that, at this point, **Mode** is invalid (not displayed).

Data Line Setting:

The oscilloscope triggers after the data channel (SDA) has transmitted data with specified bits and length.

- Press **Data Bits** to select 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 **Clock Edge** 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:

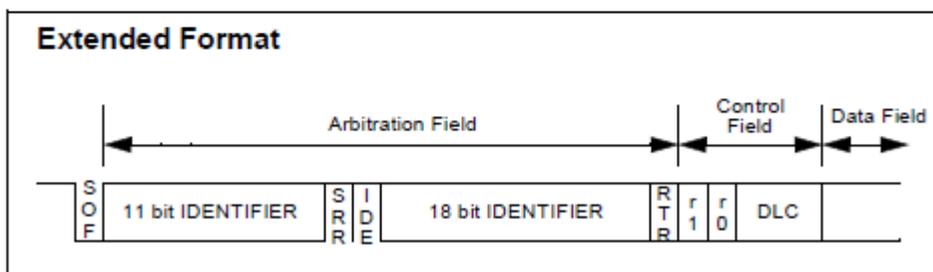
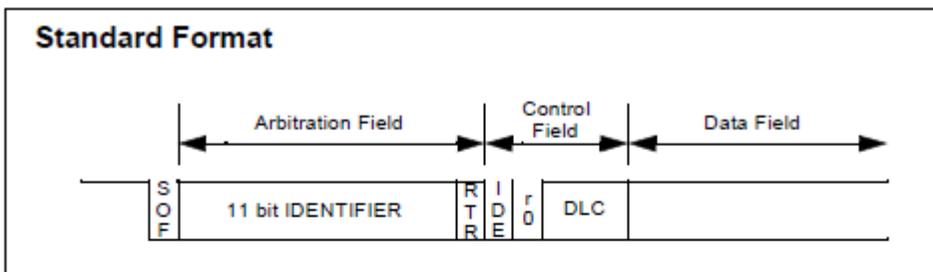
Press **Setting** to set the trigger parameters (trigger coupling, trigger holdoff and noise rejection) under this trigger type.

Trigger Level:

Use **TRIGGER**  **LEVEL** to modify the level. For details, please refer to the description of "[Trigger Level](#)" on page [5-9](#).

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 to CH4 or D0-D15 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 on the CAN signal line.

- **Tx**: transmission signal on 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 via the differential probe.

Trigger Condition:

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

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

"Data" trigger:

--press **ID Setup** and select "Specific ID" or "Random ID". When "Specific ID" is selected, press **ID Format** to select "Standard" or "Expand"; press **ID Data** and use  to input 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 input 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 ("Byte Length"×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 "Specific ID" is selected, press **ID Format** and select "Standard" or "Expand". Press **ID Data** and use  to input 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:

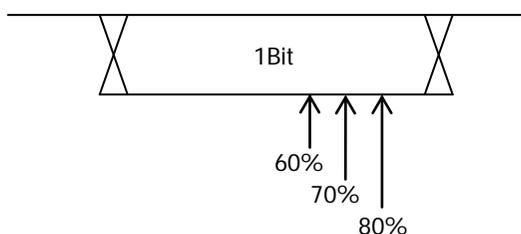
Press **Baud** to set the CAN baud to match the CAN bus signal and the values available are 10 kb/s (default), 20 kb/s, 33.3 kb/s, 50 kb/s, 62.5 kb/s, 83.3 kb/s, 100

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.

Sample Point:

Sample point is a point within a bit's time. The oscilloscope samples the bit level at this point. "Sample point" is represented by the percentage of "the time from the start of the bit's time to the sample point time" in the "bit's time".

Press **SamplePoint** and use  to modify the parameter with a step of 1% and the range is from 5% to 95%.



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:

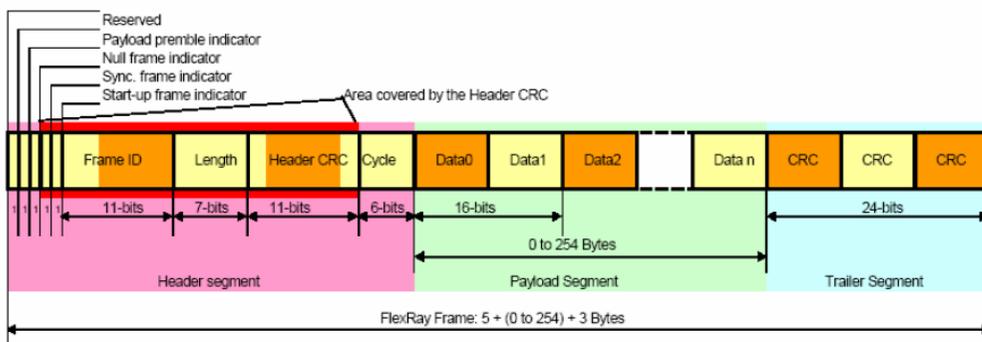
Press **Setting** to set the trigger parameters (trigger coupling, trigger holdoff and noise rejection) under this trigger type.

Trigger Level:

Use **TRIGGER**  **LEVEL** to modify the level. For details, please refer to the description of "[Trigger Level](#)" on page [5-9](#).

FlexRay Trigger

Trigger on the specified frame, symbol, error or TSS (Transmission Start Sequence) of the FlexRay bus. FlexRay is a type of differential serial bus configured with three continuous segments (namely a packet header, payload and a 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 pair, but a shielded twisted pair can be used to improve EMC performance. The figure below shows the frame format of the FlexRay 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 set CH1 to CH4 or D0-D15 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 "2.5Mb/s", "5Mb/s" or "10Mb/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: FlexRay frame rates can be 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 conditions. In addition, 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 coupling, trigger holdoff and noise rejection) 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 [5-9](#).

USB Trigger

Trigger on the SOP, EOP, RC, Suspend and Exit Suspend 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.

USB  62.5mV

Source Selection:

Press **D+** and **D-** to specify data sources for D+ and D- data cables respectively. CH1 to CH4 or D0-D15 can be selected as the [Trigger 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 start of the data packet (SOP).
- **EOP:** trigger at the end of the SEO portion of the EOP of the data packet.
- **RC:** trigger when SEO is greater than 10 ms.
- **Suspend:** trigger when the idle time of the bus is greater than 3 ms.
- **ExitSuspend:** 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 parameters (trigger coupling, trigger holdoff and noise rejection) 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 [5-9](#).

Trigger Output Connector

The trigger output connector at the rear panel can output trigger signals determined by the current setting.



Trigger Output Connector

Press **Utility** → **Aux Output** to select "TrigOut". When the oscilloscope is triggered, it will output a trigger signal determined by the current trigger setting through the **[Trig Out/Calibration]** connector.

6 To Make Measurements

The MSO4000/DS4000 can perform math operations, cursor measurements and auto measurements on sampled and displayed data.

The contents of this chapter:

- [Math Operation](#)
- [Auto Measurement](#)
- [Cursor Measurement](#)

Math Operation

The MSO4000/DS4000 include various math operations that include addition (A+B), subtraction (A-B), multiplication (AxB), division (A÷B), FFT, logic operation and advanced math operations. The results of math operations also allow for further measurements (for details, please refer to "[Cursor Measurement](#)").

Press **MATH** → **Operate** in the vertical control area (VERTICAL) at the front panel to select the desired operation function. The result of 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  and use  to adjust the vertical position of the operation results.
- Press  and use  to adjust the vertical scale of the operation results.
- Press **Invert** to turn the inverted display of the operation results on or off.
- **HORIZONTAL**  **POSITION** and **HORIZONTAL**  **SCALE** can also be used to adjust the horizontal position and scale of the operation results.

Subtraction

Subtract the waveform voltage values of signal source B from that of source 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  and use  to adjust the vertical position of the operation results.
- Press  and use  to adjust the vertical scale of the operation results.
- Press **Invert** to turn the inverted display of the operation results on or off.
- **HORIZONTAL**  **POSITION** and **HORIZONTAL**  **SCALE** can also be used to adjust the horizontal position and scale 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  and use  to adjust the vertical position of the operation results.
- Press  and use  to adjust the vertical scale of the operation results.
- Press **Invert** to turn the inverted display of the operation results on or off.
- **HORIZONTAL**  **POSITION** and **HORIZONTAL**  **SCALE** can also be used to adjust the horizontal position and scale of the operation results.

Division

Divide the waveform voltage values of signal source A by that of source 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":

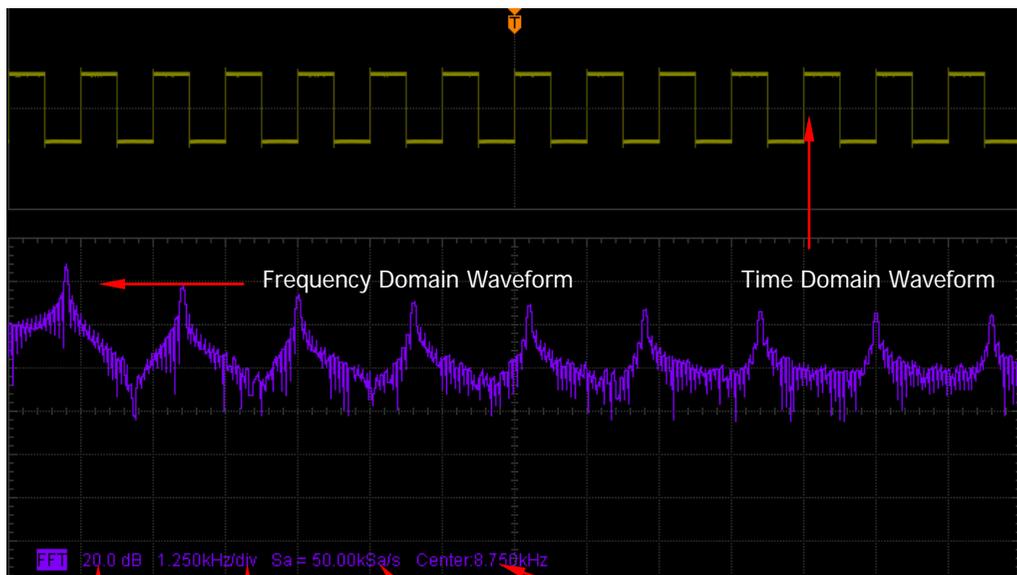
- Press **Source A** and **Source B** to select the desired channels. The channels available are CH1, CH2, CH3 and CH4.
- Press  and use  to adjust the vertical position of the operation results.
- Press  and use  to adjust the vertical scale of the operation results.
- Press **Invert** to turn the inverted display of the operation results on or off.
- **HORIZONTAL**  **POSITION** and **HORIZONTAL**  **SCALE** can also be used to adjust the horizontal position and scale of the operation results.

FFT

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

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

Press **MATH** → **Operate** to select “FFT” and set the parameters of FFT operation. Note: LA bus display functions will be automatically turned off and disabled after selecting the FFT operation.



Vertical Amplitude Horizontal Scale FFT Sample Rate Center Frequency

1. Select Source

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

2. Select Window Function

Spectral leakage can be considerably decreased when a window function is used.

MSO4000/DS4000 provides four kinds of FFT window functions which have different characteristics and are applicable to measure different waveforms. You need to select the window function according to different waveforms and their characteristics. Press **Window** to select the desired window function and the default is "Rectangle".

Table 6-1 Window Functions

Window	Characteristics	Waveforms Suitable for Measurement
Rectangle	The best frequency resolution. The poorest amplitude resolution. Similar to the situation when no window is applied.	Transient or short pulse, the signal levels before and after the multiplication are basically the same. Sine waveforms with the same amplitude and rather similar frequencies. Wide band random noise with relatively slowly changing waveform spectrum.
Hanning	Better frequency resolution. Poor 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 displayed clearly.

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 between "Split" and "Full Screen".

4. Set the Vertical Scale

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

The unit of the vertical axis can be dB or Vrms. dB and Vrms use logarithmic mode and linear mode to display vertical amplitude 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.

Press  and  and use  to set the vertical position and amplitude of the FFT frequency spectrum respectively.

5. Antialiasing

Press **Anti-Aliasing** to enable or disable the antialiasing function. When enabled, horizontal scale, FFT sample rate and center frequency are doubled.

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 operations on the waveform voltage values of the specified sources point by point and display the results. 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

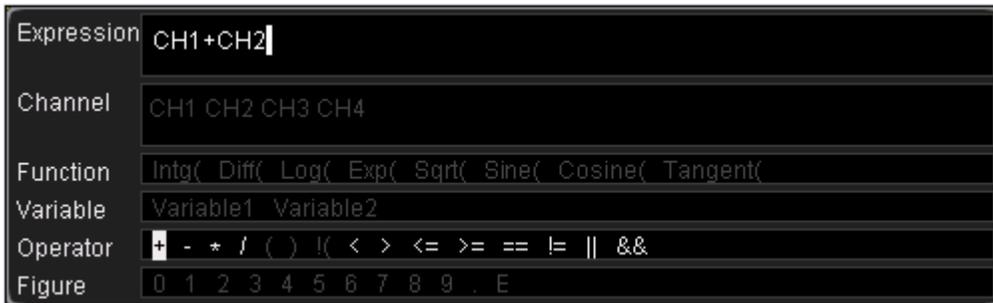
Operation	Description															
AND	<p>The results of logic AND operation of two binary bits are as follows:</p> <table> <thead> <tr> <th>A</th> <th>B</th> <th>A AND B</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>0</td> <td>0</td> </tr> <tr> <td>0</td> <td>1</td> <td>0</td> </tr> <tr> <td>1</td> <td>0</td> <td>0</td> </tr> <tr> <td>1</td> <td>1</td> <td>1</td> </tr> </tbody> </table>	A	B	A AND B	0	0	0	0	1	0	1	0	0	1	1	1
A	B	A AND B														
0	0	0														
0	1	0														
1	0	0														
1	1	1														
OR	<p>The results of logic OR operation of two binary bits are as follows:</p> <table> <thead> <tr> <th>A</th> <th>B</th> <th>A OR B</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>0</td> <td>0</td> </tr> <tr> <td>0</td> <td>1</td> <td>1</td> </tr> <tr> <td>1</td> <td>0</td> <td>1</td> </tr> <tr> <td>1</td> <td>1</td> <td>1</td> </tr> </tbody> </table>	A	B	A OR B	0	0	0	0	1	1	1	0	1	1	1	1
A	B	A OR B														
0	0	0														
0	1	1														
1	0	1														
1	1	1														
NOT	<p>The results of logic NOT operation of a binary bit are as follows:</p> <table> <thead> <tr> <th>A</th> <th>NOT A</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>1</td> </tr> <tr> <td>1</td> <td>0</td> </tr> </tbody> </table>	A	NOT A	0	1	1	0									
A	NOT A															
0	1															
1	0															
XOR	<p>The results of logic XOR operation of two binary bits are as follows:</p> <table> <thead> <tr> <th>A</th> <th>B</th> <th>A XOR B</th> </tr> </thead> <tbody> <tr> <td>0</td> <td>0</td> <td>0</td> </tr> <tr> <td>0</td> <td>1</td> <td>1</td> </tr> <tr> <td>1</td> <td>0</td> <td>1</td> </tr> <tr> <td>1</td> <td>1</td> <td>0</td> </tr> </tbody> </table>	A	B	A XOR B	0	0	0	0	1	1	1	0	1	1	1	0
A	B	A XOR B														
0	0	0														
0	1	1														
1	0	1														
1	1	0														

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

- Press **Log.Formula** 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 or D0-D15.
- Press  and use  to adjust the vertical position of the operation results.
- Press  and use  to adjust the vertical scale of the operation results.
- Press **Invert** to turn the inverted display of the operation results on or off.
- 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.
- **HORIZONTAL**  **POSITION** and **HORIZONTAL**  **SCALE** can also be used to adjust the horizontal position and scale of the operation results.

Advanced Operation

MSO4000/DS4000 provides advanced operation function that allows users to define operation functions. Press **MATH** → **Operate** → Advance → **Expression** → ON and the editing window as shown in the figure below is displayed.



Turn  to select any item in “Channel”, “Function”, “Variable”, “Operator” and “Figure” (if they are currently available for selection), then press down the knob and the item selected will be displayed in the entry box on the right of the “Expression”.

During the expression editing, 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 editing, press **Apply** and the oscilloscope will operate according to the expression you set and display the result. Note that, after **Apply** is pressed, 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 inverted 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 one or more channels from CH1, CH2, CH3 and CH4.

3. Function

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

Table 6-3 Functions

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 differentiation of the selected source. You can use differentiate to measure the instantaneous slope of a waveform.
Ln(Calculate the natural logarithm of the selected source (use constant e (approximately 2.718282) as the base
Lg(Calculate the natural logarithm of the selected source (use constant e (approximately 10) as the base).
Exp(Calculate the exponent of the selected source. For example, Exp(A) means calculate the Ath power of e .
Sqrt(Calculate the square root of the selected source.
Abs(Calculate the absolute value of the source selected.
Square(Calculate the square value of the source selected.
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 the desired values of Variable1 and Variable2. Press **Variable** and turn on the setting menu as follows.

- **Variable:** press this softkey to select the variable to be set as "Variable1" and "Variable2".
- **Step:** press this softkey to set the step used when using the knob to modify the "Mantissa". The steps available are $\times 1$, $\times 0.1$, $\times 0.01$, $\times 0.001$ and $\times 0.0001$.
- **Mantissa:** press this softkey to set the number of significant digits of the variable. After setting the "Step" and 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 bottom number in the variables. The range is from -9 to 9.

For example, Variable1 is set to 6.1074×10^8 via the following settings.

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 Operators

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

In division operation, when the divisor is 0, the result of the division is treated as 0.

6. Figure

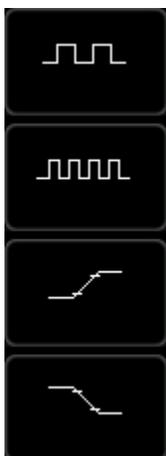
Select from figure 0 to 9, decimal point and character E. Wherein, character E represents the nth power of 10. For example, 1.5E3 means 1.5×10^3 .

Auto Measurement

MSO4000/DS4000 provides auto measurements of 24 waveform parameters and the statistics and analysis of the measurement results. What's more, you can also use the frequency counter to realize more precise frequency measurement.

Quick Measurement after **AUTO**

When the oscilloscope is correctly connected and has detected input signal, press **AUTO** to 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.

Note: The auto setting function 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 of the signal under test exceed these limits, the quick parameter measurement items might not be displayed in the pop-up menu.

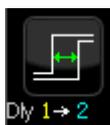
One-key Measurement of 24 Parameters

Press **MENU** at the left of the screen to turn on the measurement menu of the 24 waveform 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.

Note: When the measurement source is set to D0-D15, the "one-key" measurement only supports period, frequency, positive width, negative width, positive duty cycle, negative duty cycle, delay and phase.

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

Parameter Icons:



Measurement Results:

Freq = 1.000kHz

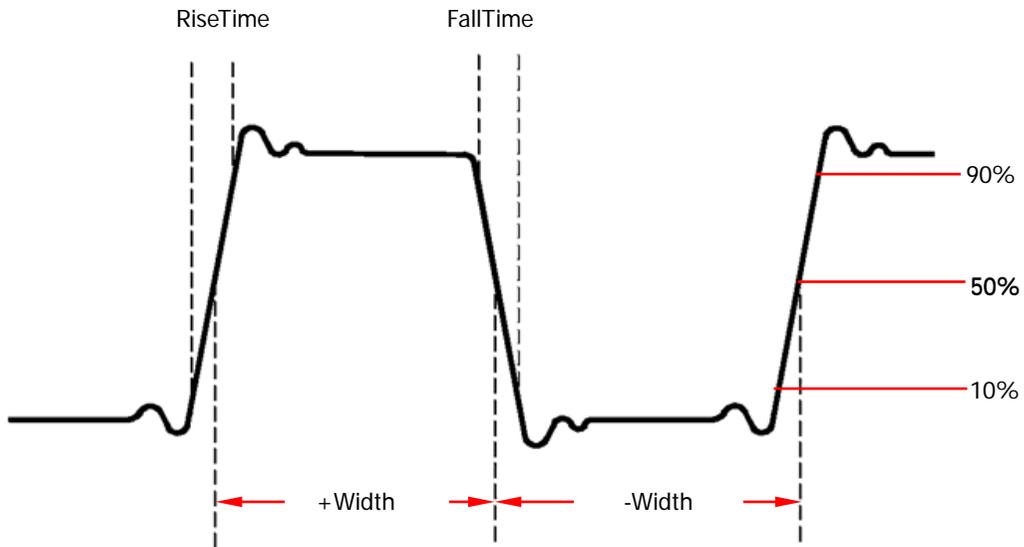
Max = 1.61 V

Dly 1→2 = 60.00µs

Phs 1→2 = 14.40 °

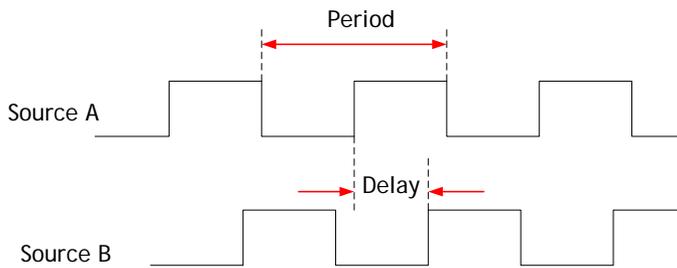
Note: If the measurement result is displayed as "*****", it means that there is no signal input in the current source or the measurement result is not within the valid range (too large or too small).

Time Parameters



1. **Period**: defined as the time between the middle threshold points of two consecutive, like-polarity edges.
2. **Frequency**: defined as the reciprocal of period.
3. **Rise Time**: the time for the signal amplitude to rise from 10% to 90%.
4. **Fall Time**: the time for the signal amplitude to fall from 90% to 10%.
5. **+ Width**: the time difference between the 50% threshold of a rising edge to the 50% threshold of the next falling edge.
6. **- Width**: the time difference between the 50% threshold of a falling edge to the 50% threshold of the next rising edge.
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

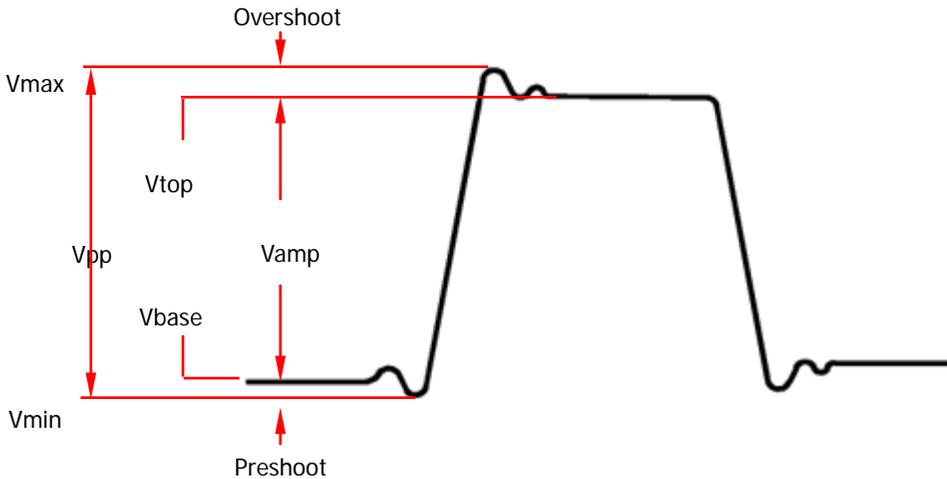


Source A and source B can be any channel from CH1 to CH4, MATH or D0-D15.

1. **Delay A→B_f**: the time difference between the rising edges of source A and source B. Negative delay indicates that the selected rising edge of source A occurred after the selected edge of source B.
2. **Delay A→B_t**: the time difference between the falling edges of source A and source B. Negative delay indicates that the selected edge of source A occurred after the selected edge of source B.
3. **Phase A→B_f**: phase difference calculated according to "**Delay A→B_f**" and the period of source A, expressed in degree.
4. **Phase A→B_t**: phase difference calculated according to "**Delay A→B_t**" and the period of source A, expressed in degree.

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

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.

$$\text{Average} = \frac{\sum x_i}{n}, \text{ wherein, } x_i \text{ is the } i\text{th point being measured, } n \text{ is the number}$$

of points being measured.

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

$$\text{the gating area. } \text{RMS} = \sqrt{\frac{\sum_{i=1}^n x_i^2}{n}}, \text{ where, } x_i \text{ is the } i\text{th point being measured, } n$$

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

-  **Area:** the area of the whole waveform within the screen. The units are the volt-second. The area measured 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.
-  **Period Area:** the area of the first period of waveform on the screen. The units are the volt-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 algebraic 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 input signal frequency.

Press **Measure** → **Counter** to select anyone of CH1 to CH4, EXT or D0-D15 as the measurement 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 frequency measurement of the input signal of CH1.



Select "OFF" to disable the frequency counter measurement function.

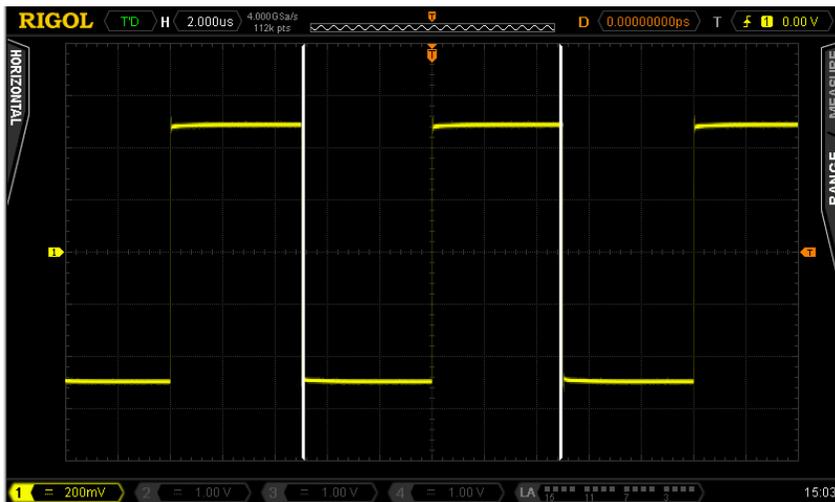
Measurement Setting

1. Source Selection

Press **Measure** → **Source** to select the desired channel for measurement (CH1 to CH4, MATH or D0-D15). The color of the parameter icons under **MENU** at the left of the screen will change with the source selected.

2. Measurement Range

Press **Measure** → **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 position of the two cursor lines respectively so as to determine the measurement range. Or press **CursorAB** and use  to adjust the positions of the cursor A and B at the same time. Note that you can press  continuously to switch between the cursors.



3. Delayed Measurement Setting

Specify the source A and source B in the measurement items “**Delay A→B \bar{f}** ” and “**Delay A→B \bar{t}** ”.

Press **Measure** → **Meas.Setting** → **Type** → “Delay” and then press **Source A** and **Source B** to set the two channel sources (CH1 to CH4, MATH or D0-D15) of delayed measurement respectively.

4. Phase Measurement Setting

Specify the source A and source B in the measurement items “Phase A→Bf” and “Phase A→Bt”.

Press **Measure** → **Meas.Setting** → **Type** → “Phase” and then press **Source A** and **Source B** to set the two channel sources (CH1 to CH4, MATH or D0-D15) of phase measurement respectively.

5. Threshold Measurement Setting

Specify the vertical level (in percentage) being measured in the analog channel. Measurements of all the time, delay and phase parameters will be influenced by this setting.

Press **Measure** → **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%.

To Clear the Measurement

If you have currently enabled one or more items in the 24 measurement parameters, you can “Delete” or “Recover” the first five parameters or “Delete” or “Recover” all the measurement items enabled. Note that the first five parameters are 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 **Measure** → **Clear** → **Item n** to “Delete” or “Recover” the specified measurement item. When one measurement item is deleted or recovered, the measurement results at the bottom of the screen will move one-item left or right.

Press **Measure** → **Clear** → **All Item** to “Delete” or “Recover” all the measurement items enabled.

Note: Press and hold **Measure** to quickly clear or restore all the measurement items enabled.

All Measurement

All measurements can be enabled. This setting displays all of the time and voltage parameters (each measurement source has 20 items, measurements can be performed on the four measurement sources and MATH at the same time) of the current measurement source and displays the results. Press **Measure** → **All Measure** to enable or disable the all measurement function. Press **All Measure Source** and use  to select the channel(s) to be measured (CH1 to CH4 and MATH).

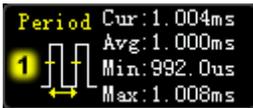
- When all measurement is enabled, “One-key” measurement is also valid.
- [“To Clear the Measurement”](#) will not clear the results of all measurement.



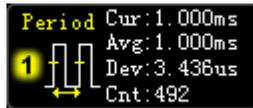
Statistic Function

The MSO4000/DS4000 also features statistics which can display the current, average, minimum (or standard deviation) and maximum (or count) values of up to 5 measurement items.

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 **Measure** → **Reset Statistic** to clear the history data and make statistic again.

Measurement History

To view the history measurement data, press **Measure** → **MeasHistory** → **MeasHistory** → “ON”. The history data can be displayed in two modes:

- Graph: display the results of the 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 Measurement](#)" parameters can be measured through cursor measurement.

Press **Cursor** → **Mode** at the front panel and use  to select the desired cursor mode (the default is "OFF") and then press down the knob. The modes available are "Manual", "Track", "Auto" and "X-Y".

Manual Mode

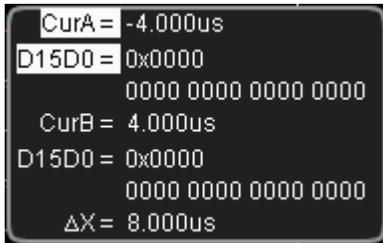
In this mode, a pair of cursors will appear. You can adjust the cursors manually to measure the X (or Y), X increment (or Y increment) between cursors and the reciprocal of X increment on the waveform of the selected source (CH1 to CH4, MATH or LA). When the measurement source is set to LA, the logic level value of the digital channel currently opened will be displayed in binary form and high level is 1, low level is 0.

Press **Cursor** → **Mode** → "Manual" to turn the manual cursor function on. When the display mode is set to X-Y and the measurement source is set to CH1 to CH4 or MATH, the measurement results will be displayed at the upper left corner of the screen in the following mode.



- A->X : the X value at cursor A (CurA). X value takes the trigger position as reference.
- B->X : the X value at cursor B (CurB). X value takes the trigger position as reference.
- A->Y: the Y value at cursor A (CurA). Y value takes the channel GND as reference.
- B->Y: the Y value at cursor B (CurB). Y value takes the channel GND as reference.
- ΔX : the horizontal difference between cursor A and B.
- $1/\Delta X$: the reciprocal of the horizontal difference between cursor A and B.
- ΔY : the vertical difference between cursor A and B.

When the measurement source is set to LA and the display mode is automatically set to X, the measurement result will be displayed at the upper left corner of the screen in the following mode.



- CurA: the X value at cursor A. X value takes the trigger position as reference.
- D15D0: display the logic level value of the current cursor A with hex and binary data which correspond to D15-D0 from left to right. If the current digital channel is turned off, the channel is represented by X.
- CurB: the X value at cursor B. X value takes the trigger position as reference.
- D15D0: display the logic level value of the current cursor B with hex and binary data which correspond to D15-D0 from left to right. If the current digital channel is turned off, the channel is represented by X.

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

1. Select Display Mode

Press **DisplayMode** to select "X", "Y" or "X-Y". The X cursors are a pair of vertical solid/dotted lines and are usually used to measure the time parameters. The Y cursors are a pair of horizontal solid/dotted lines and are usually used to measure the voltage parameters. The X-Y cursors are a pair of vertical and horizontal solid/dotted lines and can be used to measure the time and voltage parameters at the same time.

2. Select Measurement Source

Press **Source** to select the waveform of the analog channels (CH1 to CH4) or math operation results (MATH) or the digital channel (LA) for measurement. If "None" is selected, no cursor will be displayed.

3. Select Screen Region

- When Zoom is turned on (pressing Horizontal  **SCALE** can turn on Zoom), the screen is divided into the Main and Zoom areas. Press **Screen Region** to select Main or Zoom. When Main is selected, the cursors are displayed in the Main area and are used to measure the parameters in the Main are; the

measurement results are displayed in the Zoom area. When Zoom is selected, the cursors are displayed in the Zoom area and are used to measure the parameters in the Zoom area; the measurement results are displayed in the Main area.

- When Zoom is turned off, **Screen Region** is automatically set to Main and users can not modify it.

4. Select X (Y) Axis Unit

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

- s: when this unit is selected, in the measurement results, A->X, B->X and ΔX are in "s" and $1/\Delta X$ is in "Hz".
- Hz: when this unit is selected, in the measurement results, A->X, B->X and ΔX are in "Hz" and $1/\Delta X$ is in "s".
- °: when this unit is selected, in the measurement results, A->X, B->X and ΔX are in "°". At this point, A->X, B->X and ΔX will change to "0°", "360°" and "360°" respectively when you press **Set Cursor**, no matter where cursor A and B are currently located.
- %: when this unit is selected, in the measurement results, A->X, B->X and ΔX are in "%". At this point, A->X, B->X and ΔX will change to "0%", "100%" and "100%" respectively when you press **Set Cursor**, no matter where cursor A and B are currently located.

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

- Source Unit: when this unit is selected, in the measurement results, the units of A->Y, B->Y and ΔY will be automatically set to the unit of the current source.
- %: when this unit is selected, in the measurement results, CurA, CurB and ΔX are in "%". At this point, A->Y, B->Y and ΔX will change to "0%", "100%" and "100%" when you press **Set Cursor**, no matter where cursor A and B are currently located.

5. Adjust the Cursor Position

- Adjust the horizontal position of the cursor:

When the current display mode is "X" or "X-Y" (press **SelectCursor** to select "X"), press **CursorA** and use  to adjust the horizontal position of

cursor A. During the adjustment, the measurement result will change accordingly. The adjustable range is limited within the screen. Press **CursorB** and use  to adjust the horizontal position of cursor B. During the adjustment, the measurement result will change accordingly. The adjustable range is limited within the screen. Press **CursorAB** and use  to adjust the horizontal 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.

- Adjust the vertical position of cursor:
When the current display mode is "Y" or "X-Y" (press **SelectCursor** to select "Y"), press **CursorA** and use  to adjust the vertical position of cursor A. During the adjustment, the measurement result will change accordingly. The adjustable range is limited within the screen. Press **CursorB** and use  to adjust the vertical position of cursor B. During the adjustment, the measurement result will change accordingly. The adjustable range is limited within the screen. Press **CursorAB** and use  to adjust the vertical 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.

6. Measurement Example

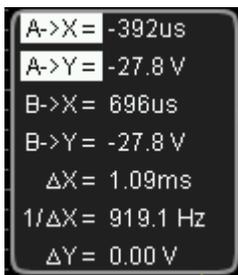
Use manual cursor measurement to measure the period (ΔX) of a square waveform and the result is 1 ms equaling the result from auto measurement.



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 at the last adjustment of the cursors.

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



- A->X : the X value at cursor A. X value takes the trigger position as reference and "s" or "Hz" (when measuring FFT waveform) as its unit.
- A->Y: the Y value at cursor A. Y value takes the channel GND as reference and uses the same unit as the current source.
- B->X: the X value at cursor B . X value takes the trigger position as reference and "s" or "Hz" (when measuring FFT waveform) as its unit.
- B->Y : the Y value at cursor B. Y value takes the channel GND as reference and uses the same unit as the current source.
- ΔX: the horizontal difference between cursor A and B.
- 1/ΔX: the reciprocal of the horizontal difference between cursor A and B.
- ΔY : the vertical difference between cursor A and B.

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

1. Select Measurement Source

Press **Cursor A** to select the waveform of analog channels (CH1 to 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 **Cursor B** to select the waveform of analog channels (CH1 to 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 you can also press continuously to switch the current cursor)

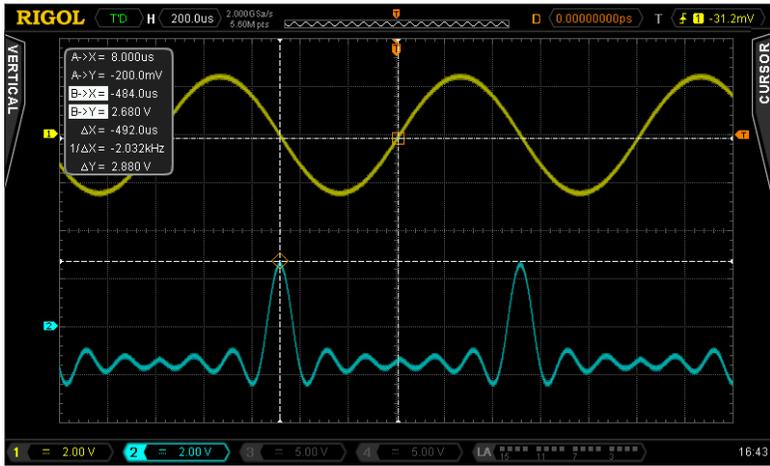
- Adjust cursor A: press **CursorA** and use  to adjust the position of cursor A. During the adjustment, the measurement result will change accordingly. The adjustable range is limited within the screen.
- Adjust cursor B: press **CursorB** and use  to adjust the position of cursor B. During the adjustment,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



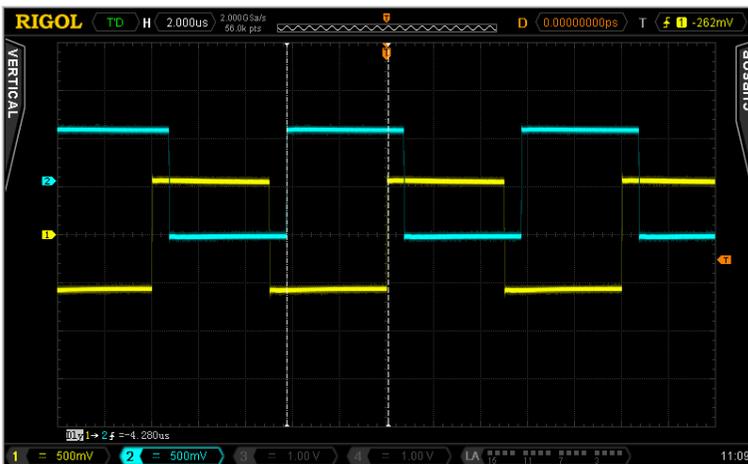
Auto Mode

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

Press **Cursor** → **Mode** → “Auto” and the number of cursors displayed on the screen is determined by the measurement parameter enabled (different measurement parameter needs different number of cursors). Note that no cursor will be displayed if no auto cursor measurement parameter is enabled or the measurement source has no input. 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 to the desired parameter.

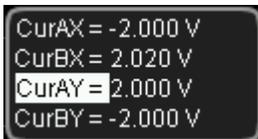
The following figure shows the auto measurement of the delay between two channels.



X-Y Mode

The X-Y mode is only available when the main time base mode is "X-Y". In this mode, you can adjust the cursor positions to measure the X and Y values at the crossing points of the two pair of cursors.

Press **MENU** in the horizontal control area (HORIZONTAL) at the front panel and press **Time Base** to select "X-Y". Then, press **Cursor** → **Mode** → "X-Y" to enable the "X-Y" cursor measurement function. The measurement results are displayed at the upper-left corner of the screen, as shown in the figure below.



- CurAX: the X value at cursor A.
- CurBX: the X value at cursor B.
- CurAY: the Y value at cursor A.
- CurBY: the Y value at cursor B.

You can adjust the cursor positions in the following way:

1. Select cursor region

Press **Region** to select "XY1" or "XY2".

2. Select cursor type and adjust cursor positions

- Press **Type** to select "X". Press **CursorA** and use  to adjust the horizontal position of cursor A. Press **CursorB** and use  to adjust the horizontal position of cursor B. Press **CursorAB** and use  to adjust the horizontal positions of cursor A and cursor B at the same time. During the adjustment, the measurement results will change accordingly and the adjustable range is limited within the screen.
- Press **Type** to select "Y". Press **CursorA** and use  to adjust the vertical position of cursor A. Press **CursorB** and use  to adjust the vertical position of cursor B. Press **CursorAB** and use  to adjust the vertical positions of cursor A and cursor B at the same time. During the adjustment,

the measurement results will change accordingly and the adjustable range is limited within the screen.

7 Logic Analyzer

The logic analyzer is a waveform test device which is similar in function to the oscilloscope. It can monitor and save the logic levels (high or low) of input signals and display them in a graphical way. It is very convenient for users to detect and analyze the errors in the digital circuit design (hardware or software design). This chapter will introduce how to use the logic channels of the MSO4000 mixed signal digital oscilloscope and about the using methods of the logic probe, please refer to the corresponding probe User's Guide.

The contents of this chapter:

- [To Select the Logic Channel](#)
- [To Turn on/off the Logic Channel](#)
- [Group Setting](#)
- [To Set the Waveform Size](#)
- [Automatic Arrangement Setting](#)
- [To Set the Threshold](#)
- [To Use the Digital Bus](#)
- [To Set the Label](#)
- [Delay Calibration](#)

To Select the Logic Channel

Press **CH/Group** and use  to select any channel of D0-D15 or any group of Group1-Group4 as the current active channel (only the groups that contain digital channels can be selected, please refer to the instruction in "[Group Setting](#)") as the current active channel. If "None" is selected, there is no channel selected. The labels and waveform of the channels currently selected are highlighted in red. If any group of Group1-Group4 is selected, the labels of all the channels and the waveform of the first channel in the selected group are displayed in red.

Tip

You can move the channels currently selected to the desired position using the **LA_** , press this knob again to fix the channels and select the next channel or group at the same time.

To Turn on/off the Logic Channel

Press **On/Off** to enter the "On/Off" setting menu of the digital channel.

- Press **D7-D0** to turn the channels (D7-D0) on or off at the same time.
- Press **D15-D8** to turn the channels (D15-D8) on or off at the same time.
- Press **Sel CH**, use  to select the desired channel and press  to turn it on or off.
- Press **Group**, use  to select the desired group and press  to turn the channels in the group on or off.

Note: Only the groups that contain digital channels can be selected. For how to group the channels, please refer to the instruction in "[Group Setting](#)".

Group Setting

Press **Group Set** to enter the group setting menu and you can group 16 digital channels.

- **Group:**

Press **Group1** to open the channel list (there is a status icon at the left of each channel), use  to select the desired channel and press  to group the channel into Group1. At this point, the corresponding status icon is .

Group the other channels using the same method. Any channel can only be grouped into one group. The channels grouped into other groups cannot be selected and the status icon is . The channels with  can be selected.

- **UnGroup**

Press **UnGroup** to open the channel and group list, use  to select the desired channel or group and press  to cancel the grouping status of the channel selected or all the channels contained in the group selected.

To Set the Waveform Size

Press **WaveSize** to adjust the waveform size of the channels that have been turned on, you can select S (small), M (medium) or L (large).

Note: L is only available when the number of active channels is not more than 8.

Automatic Arrangement Setting

Press **ReOrder** to select the waveform ordering mode of the channels currently turned on. You can select "D0-D15" or "D15-D0" and the default is "D0-D15".

- D0-D15: from top to bottom, the waveforms on the screen are D0-D15.
- D15-D0: from top to bottom, the waveforms on the screen are D15-D0.

To Set the Threshold

Press **Threshold** to enter the threshold setting menu. You can set the threshold of the two channel groups independently according to your need for the threshold level of the two channel groups (D7-D0 and D15-D8) can be adjusted independently. When the voltage of the input signal is greater than the threshold currently set, it is regarded as logic 1, otherwise logic 0.

Press **LevelType** to select preset values or user-defined.

The preset values include TTL, 5.0V CMOS, 3.3V CMOS, 2.5V CMOS, 1.8V CMOS, ECL, PECL, LVDS and 0V. After any one of the preset values is selected, press **App-toD7-D0** to apply this threshold to the channels of D7-D0; press **App-toD15-D8** to apply this threshold to the channels of D15-D8.

After "User Def" is selected, press **D7-D0** and **D15-D8** respectively and use  to set the desired threshold.

To Use the Digital Bus

Users can display the three groups of channels (D7-D0, D15-D8 and D15-D0) as buses (BUS1 and BUS2). The value of each bus will be displayed at the bottom of the screen in data or graph mode.

Press **Bus** to enter the digital bus setting menu.

- Press **Bus** to select BUS1 or BUS2.
- Press **BusStatus** to turn the digital bus on or off.
- Press **Sel CH** to select the corresponding channel group of the digital bus (BUS1 or BUS2) and D7-D0, D15-D8 and D15-D0 can be selected.
- Press **Endian** to set the endain of the bus. You can select LSB (D0 is located at the high bit) or MSB (D15 is located at the high bit), and the default is LSB.
- Press **RefClock** to select any channel of D0-D15 or CH1-Ch4 as the clock channel of the bus. When "None" is selected, the clock channel is not set.
- Press **Slope** to set the sample slope type of the reference clock and you can select  (the rising edge) or  (the falling edge).
- Press **Type** to set the display type of the digital bus and you can select "Graph" or "Data". In the data type, the bus value is displayed directly; in the graph type, you can view the variation trend of the bus.
- Press **Format** to select the display format of the bus and you can select Hex, Decimal, Binary or ASCII.
- Press **JitterReject** to turn the jitter reject function on or off. When the reference clock is not selected for the bus, the transition state of each channel will be lead to changes of the bus data and instable data may occur on the bus when the channel changes. When jitter reject is enabled, the bus will not display the data of which the hold time is less than the jitter time and retain the previous valid data set.
- Press **JitterTime** and use LA_  or the navigation knob to set the jitter time and the range available is from 1 ns to 1 ms.

To Set the Label

To identify different data buses conveniently, you can set another label for each digital channel. You can use the preset label or input the label manually.

Press **Sel CH** to select the digital channel which needs label and you can select any channel from D0-D15.

Press **Sel Preset** to select the preset labels (ACK, ADO, ADDR, BIT, CAS, CLK, DATA, HALT, INT, UB, LOAD, NMI, OUT, PIN, RAS, READY, RESET, RX, TX or WR).

You also can press **Input Lab** to open the label input interface and input the label manually. For specific input method, please refer to the instruction in "[Channel Label](#)".

Delay Calibration

When using an oscilloscope for actual measurement, the transmission delay of the probe cable may bring great error (zero offset). MSO4000/DS4000 allows users to set a delay time to calibrate the zero offset of the corresponding channel. Zero offset is defined as the offset of the crossing point of the waveform and threshold level line relative to the trigger position.

Press **Delay-Cal** and use  to set the delay calibration time and the range available is from 100 ns to -100 ns.

8 Protocol Decoding

Protocol analysis can be used to discover errors, debug hardware and accelerate development of digital I/O circuits more conveniently than analog troubleshooting alone. Protocol decoding is the basis of protocol analysis and can provide more error information as well. The MSO4000/DS4000 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 to CH4). As the decoding functions and setting methods of the two buses are the same, this chapter only takes Decode1 for illustration purposes.

To get the decoding option information, please refer to "[Appendix A: Accessories and Options](#)" on page [18-1](#).

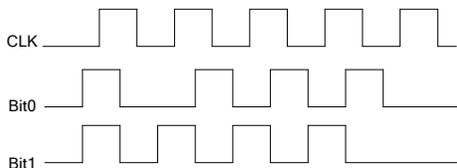
When you have ordered the decoding option, please refer to "[Option Management](#)" on page [14-16](#) to activate the corresponding option.

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 a clock line and data lines. As shown in the figure below, CLK is the clock line, while Bit0 and Bit1 are the 0 bit and 1st 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 each data point (logic "1" or logic "0") according to the preset threshold level.

Press **Decode1** → **Decode** to select "Parallel" and open the parallel decoding function menu.

1. Clock Line Setting (CLK)

Press **CLKChannel** to select any channel (CH1 to CH4 or D0-D15) as the clock channel. If "None" is selected, no clock channel is set.

Press **Slope** to set the oscilloscope to sample the channel data on the rising edge () , falling edge () or rising&falling edges (). If no clock channel is 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.**

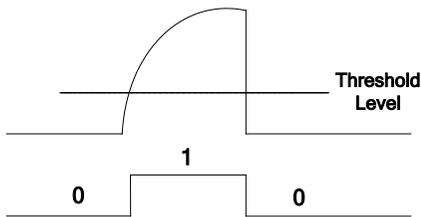
First, press **CurrentBit** to select the bit that needs to specify channel. The default is 0 and the range available is always 1 smaller than the bus bits. For

example, when the bus bits 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.

- 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.
- ThresholdL: 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 set the display format of the bus to Hex, Decimal, Binary or ASCII.

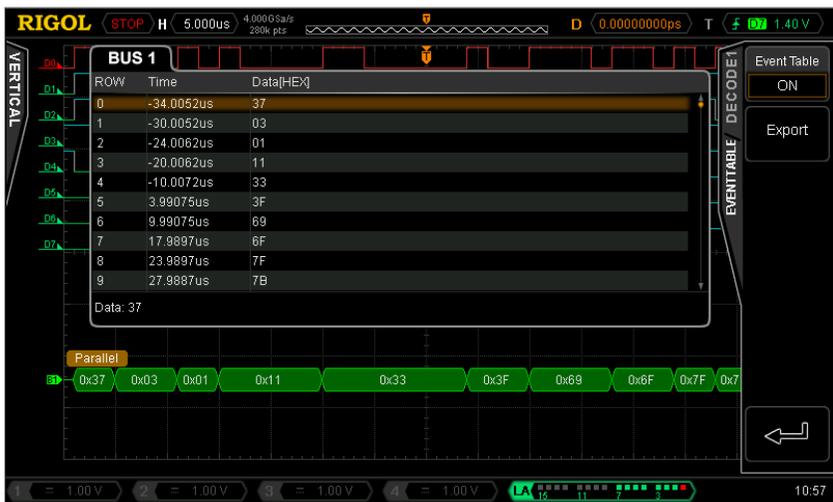
Press **Offset** and use  to adjust the vertical display position of the bus.

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

5. Decoding Table

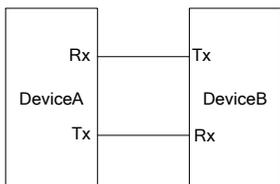
The decoding table displays the decoded data and the corresponding line number and time in table format. This can be useful when observing longer data transmissions by presenting the data in a tabular format.

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. The decoding table lists the decoded data in time order. 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.

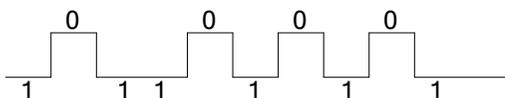


RS232 Decoding (Option)

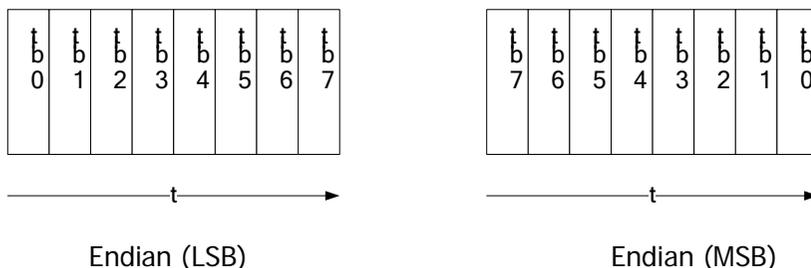
RS232 serial bus consists of the transmitting data line (TX) and the receiving data line (RX).



Industry standard RS232 uses "Negative Logic", namely high level is logic "0" and low level is logic "1".

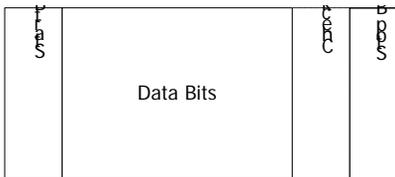


By default RS232 also uses LSB (Least Significant Bit) transmission sequence, namely the lowest bit of the data is transmitted first. While for MSB (Most Significant Bit), the highest bit of the data is transmitted first.



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

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



Start Bit: represents when the data begins. Setting the **Polarity** is equivalent to specifying the "Start Bit".

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

Even-OddCheck: check the correctness of the data transmission.

- Odd-Check: 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 Check: the number of "1" in data bit and check bit is an even. For example, when 0x55 (01010101) is sent, a 0 should be filled in the check bit.
- None: no check bit during the transmission.

Press **Decode1** → **Decode** to select "RS232" to open the RS232 decoding function menu.

1. TX and RX Channel Setting

Press **TX** to select any channel (CH1 to CH4 or D0-D15) 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 normal. The oscilloscope will select the rising or falling edge as the start position during decoding.

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 9600bps.

5. Data Packet Setting

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

- Press **Data Bits** to set the data width of each frame. 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 Check or Even Check.
- Press **Packet** to enable or disable the packet end. When packet end is enabled, several data blocks are combined according to the packet end.
- Press **PacketEnd** to set the packet end during 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 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 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. This can be useful when observing longer data transmissions by presenting the data in a tabular 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. The decoding table lists the decoded data in time order. If an 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.

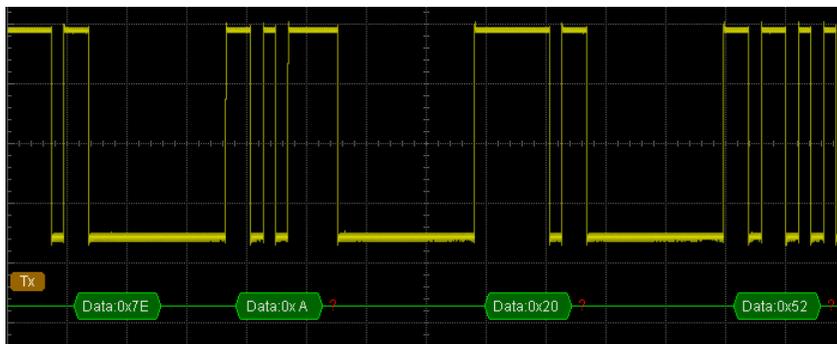


8. The Error Expression during Decoding

The MSO4000/DS4000 makes full use of the resources such as color and view to express the results of the protocol decoding effectively so as to let users find the desired information quickly.

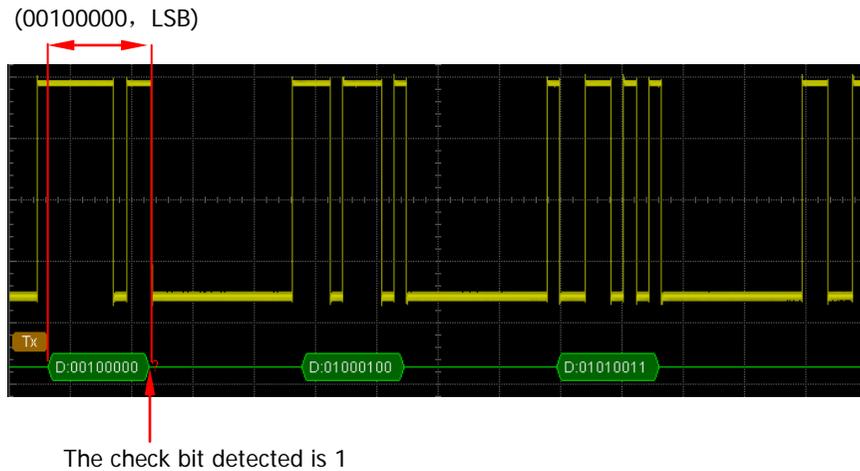
End Frame Error:

This error is generated when the end frame condition is not met. When the stop bit is set to 1.5, a red error mark is displayed (note that the red mark is displayed in different modes according to the horizontal time base setting. When the horizontal time base is small,  is displayed. Otherwise,  is displayed) is displayed.



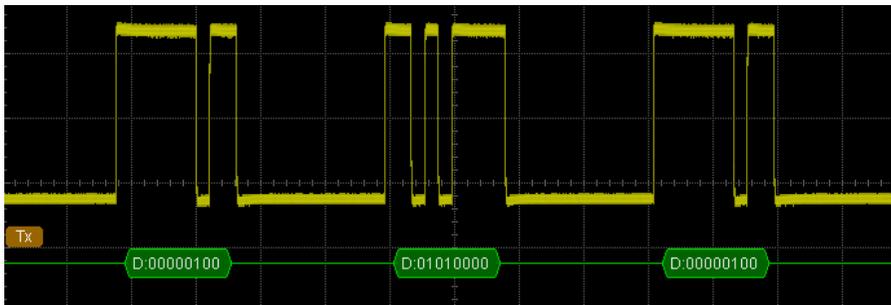
Check Error:

When a check bit error is detected during the decoding, a red error mark will be displayed. For example, when the transmitting terminal is set to none check and the decoder is set to odd check, the following check error occurs:



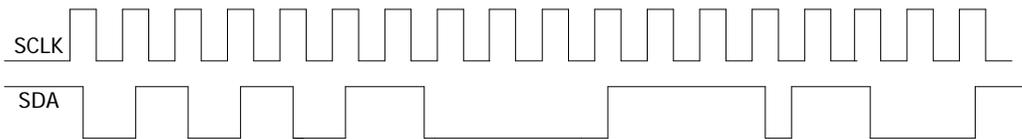
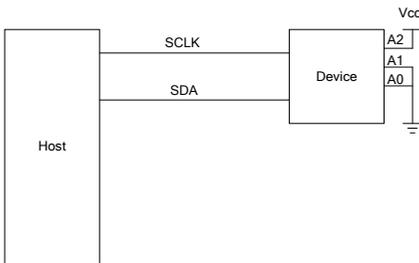
Wherein, there is an odd number (1) of 1 in the 8 bits data 00100000 and the check bit should be 0. The check bit detected on the TX is 1, thus a check error occurs.

After the decoder is set to none check, the decoding shows no error.



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: denotes the data channel.

Press **Decode1** → **Decode** to select "I2C" and open the I2C decoding function menu.

1. SCLK Setting

Press **SCLK** to select any channel (CH1-CH4 or D0-D15) 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 or D0-D15) as the data channel.

Press **SDAThreshold** to set the threshold of the data channel.

3. 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 turn the bus display on or off.

4. Decoding Table

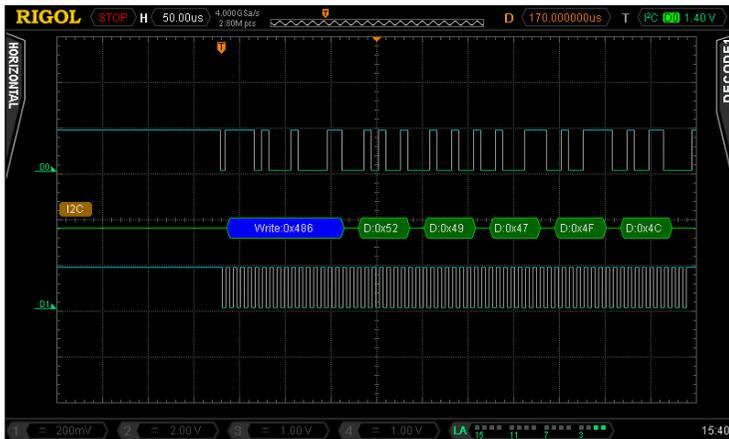
The decoding table displays the decoded data, the corresponding line number, time, data direction, ID and ACK information in table format.

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, press **Export** to export the data table to the external USB storage device in CSV format.

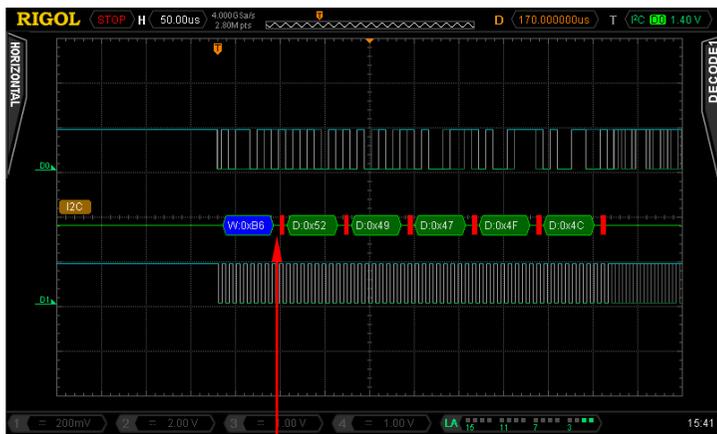


5. Error Expressions during Decoding

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. Press **Include R/W** to select open and "R/W" will be the part of the address value in the **AddrBits**.



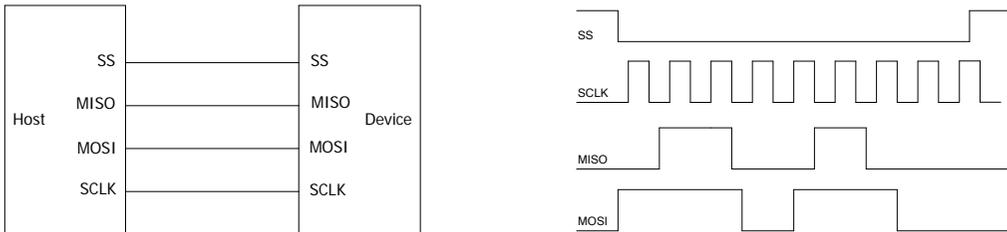
When the ACK (ACKnowledge Character) is not met, the red error marks as shown in the figure below will be displayed.



ACK=1

SPI Decoding (Option)

SPI serial bus consists of chip select line (SS), clock line (SCLK), MISO and MOSI.



Press **Decode1** → **Decode** to select “SPI” and open the SPI decoding function menu.

1. Decoding mode

Press **Mode** to set the desired decoding mode.

- CS: press **SS** to enter the chip select line setting interface. Press **Channel** to select D0-D15 or CH1-CH4 as the chip select channel. When CH1-CH4 is selected, press **Threshold** to set the threshold of the selected channel. When D0-D15 is selected, the threshold does not need to be set. Press **Polarity** to set the polarity of the chip select channel to **High** (positive polarity) or **Low** (negative polarity). Note that the **SS** softkey is only valid when this mode is selected.
- TimeOut: press **TimeOut** to set the timeout time and the range available is from 1 ns to 1 s. Note that at this point, the **SS** softkey is invalid (not displayed).

2. SCLK Setting

Press **SCLK** to turn on the clock line setting interface.

- Press **Channel** to select any channel (CH1-CH4 or D0-D15) as the clock channel.
- Press **Slope** to set the instrument 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 enter the MISO data line setting interface.

- Press **Channel** to select any channel (CH1-CH4 or D0-D15) as the MISO data channel. When "None" is selected, this data line is not set.
- Press **Polarity** to set the polarity of the MISO data line to  (the high level is 1) or  (the low level is 1).
- Press **Threshold** to set the threshold of the MISO data channel.

4. MOSI Setting

Press **MOSI** to enter the MOSI data line setting interface.

- Press **Channel** to select any channel (CH1-CH4 or D0-D15) as the MOSI data channel. When "OFF" is selected, this data line is not set.
- Press **Polarity** to set the polarity of the MOSI data line  (the high level is 1) or  (the low level is 1).
- Press **Threshold** to set the threshold of the MOSI data channel.

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 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 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 in table format. This can be useful when observing longer data transmissions by presenting the data in a tabular format.

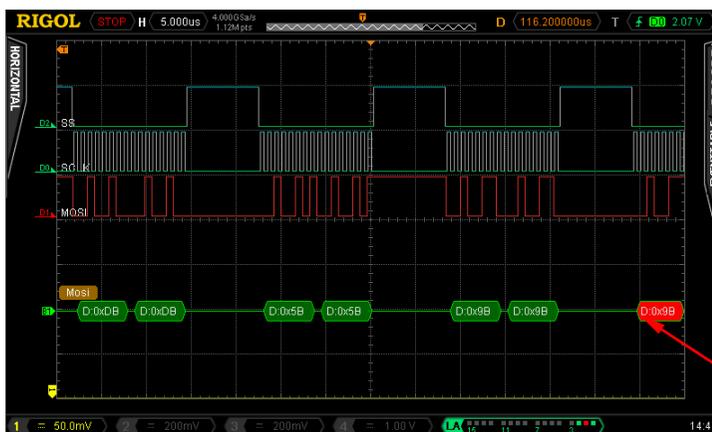
Press **Event Table** → **Event Table** to select "ON" (note that this operation is only available when **BusStatus** is set to "ON") to enter the decoding table

interface as shown in the figure below. If a USB storage device is currently connected to the instrument, press **Export** to 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 does not contain enough visible waveform to cover the data frame, the data will be filled with red patches. For example, when Data Bits is set to 7 and SCLK slope is set to rising edge, a decoding error will be generated.



Not enough
for 7 bits

CAN Decoding (Option)

Press **Decode1** → **Decode** and select “CAN” to open the CAN decoding function menu.

1. Source

Press **Source** and select any channel (CH1-CH4 or D0-D15) 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. The positive lead of the probe connects CAN_H and the negative lead connects CAN_L.

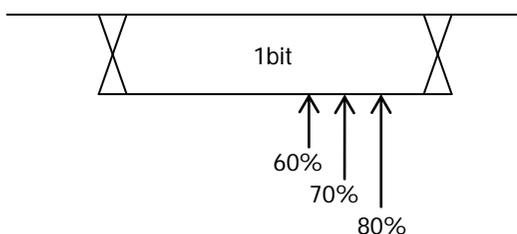
3. Baud

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 the 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 a point within a bit’s time. The oscilloscope samples the bit level at this point. “Sample point” is represented by the percentage of “the time from the start of the bit’s time to the sample point time” in the “bit’s 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** to set the bus display format to Hex, Decimal, Binary or ASCII.

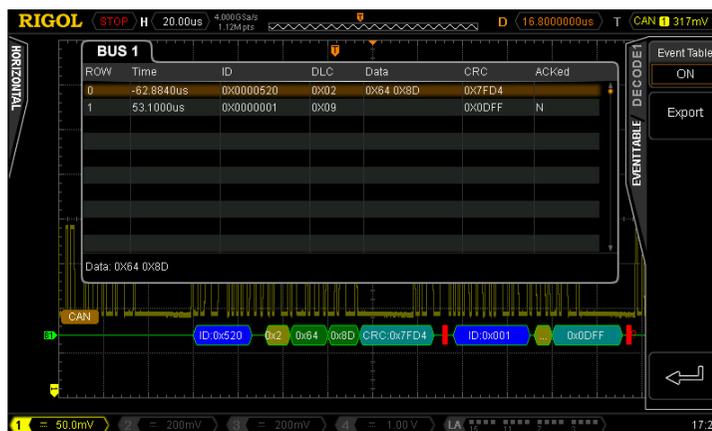
Press **Offset** and use  to adjust the vertical display position of the bus.

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

7. Decoding Table

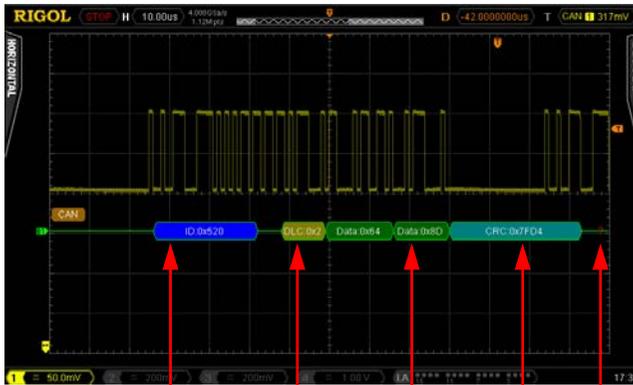
The decoding table displays the decoded data, the corresponding line number, time, frame ID, DLC, CRC and ACK information in table format.

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, press **Export** to export the data table to the external USB storage device in CSV format.



8. Decoded CAN Data Interpretation

- Frame ID: display as hex digits in blue.
- Data Length code (DLC): displayed as a chartreuse patch.
- Data Frame: displayed as green patches if data is successfully decoded. The frames appear as red patches if the data frame is lost.
- Cyclic Redundancy Check (CRC): displayed in a light blue patch when valid and red error mark is displayed when error occurs.



Address ID Data Length Data CRC Check Error

FlexRay Decoding (Option)

Press **Decode1** → **Decode** and select “FlexRay” to open the FlexRay decoding function menu.

1. Source

Press **Source** to select any channel (CH1-CH4 or D0-D15) 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.

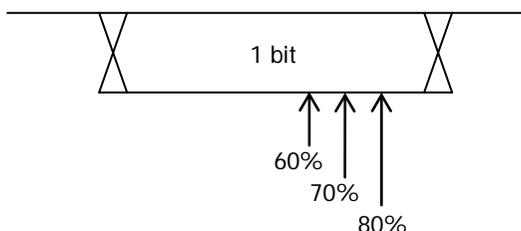
4. Baud

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

5. Sample Point Position

The sample point is a point within a bit's time. The oscilloscope samples the bit level at this point. “Sample Point” is expressed by the percentage of “the time from the start of bit to the sample bit time” in “bit's 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

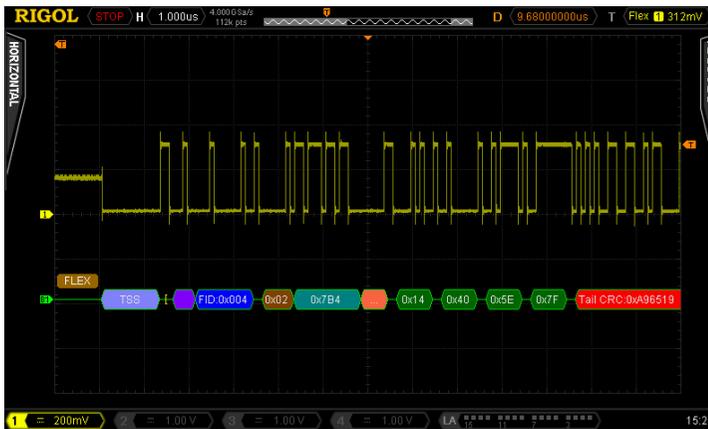
The decoding table lists the decoded data, the corresponding line number, time and error information in table format. This can be useful when observing longer data transmissions by presenting the data in a tabular format.

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, press **Export** to export the data table to the external USB storage device in CSV format.

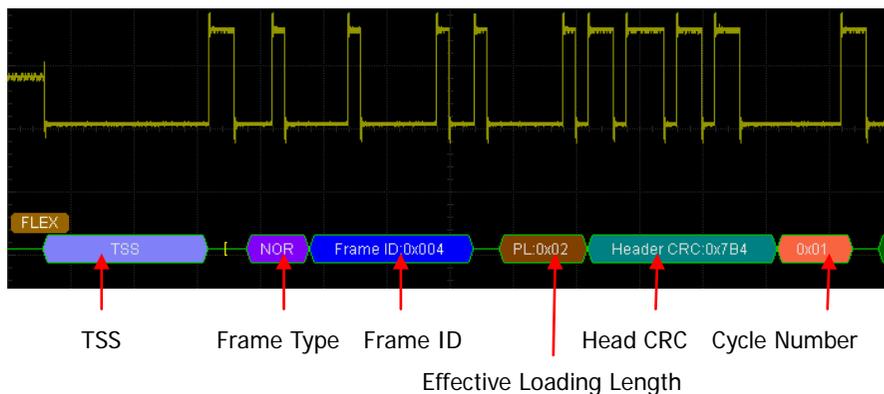


9. Explanation of the Decoded FlexRay Frame Data

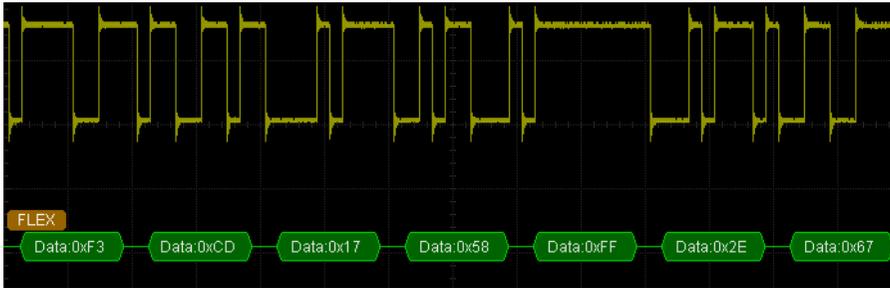
The decoded FlexRay frame data is as shown in the figure below.



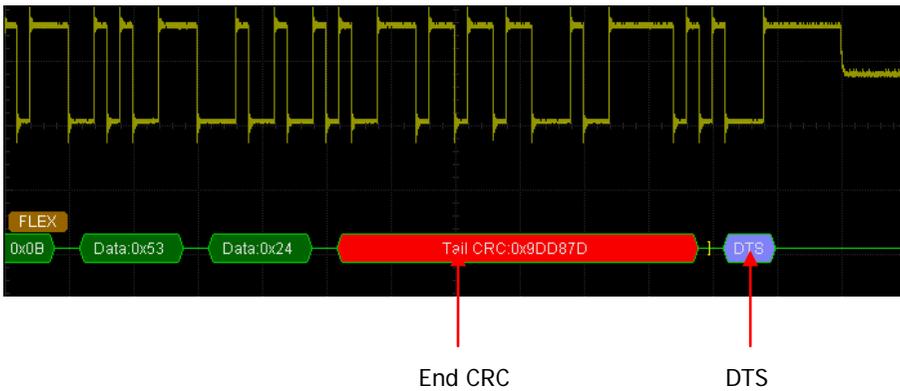
- TSS: transmission start sequence and is expressed by a light purple patch.
- Frame Type: FlexRay frame can be NORMAL, SYNC, SUP or NULL. The frame type in the figure above is "NOR" (namely NORMAL) and is expressed by a purple patch.
- Frame ID: decimal number and is expressed by a blue patch.
- Effective Loading Length: a decimal number and is expressed by a brown patch.
- Head CRC: a hexadecimal number and is expressed by a blue-green patch. When CRC is invalid, it is expressed by red patch.
- Cycle Number: a decimal number and is expressed by a pink patch.



- Data: displayed in the format (Hex, Decimal, Binary or ASCII) specified in **Format** and expressed by a green patch.



- End CRC: a hexadecimal number and is expressed by a red patch. When CRC is invalid, it is expressed by a red patch.
- DTS: dynamic end sequence and is expressed by a light purple patch.



9 Reference Waveform

In actual testing processes, the waveforms being tested can be compared with a reference waveform.

The contents of this 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 the REF function. Note that when the time base is in X-Y mode, REF function can not be enabled.

The MSO4000/DS4000 provides 10 reference waveform channels. Press **Channel** and use  to set the desired reference channel to on or off and a channel icon (for example, ) of the channel enabled will be display at the left side of the screen grid.



When the 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 introductions below.

To Set the Color

MSO4000/DS4000 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.

To Select REF Source

Press **Current** and use  to select any reference channel (Ref1 to Ref10) that has been turned on and then press **Source** to specify a reference source (CH1 to CH4, MATH or D0-D15) for this channel.

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 in the volatile memory and the waveform will be cleared at power-off.

To Adjust REF Waveform Display

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

Press **REF** to enable the REF function. 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 the **Save** operation was executed.

To Export to Internal or External Memory

Users can also save the reference waveform to the internal Flash memory or 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 store 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 reference waveform stored in the internal Flash memory or external USB storage device to the internal memory.

Press **Import** to enter file recall interface. Please refer to the relative descriptions in "[Store and Recall](#)" to import the reference waveform to the internal memory of the instrument.

10 Pass/Fail Test

The MSO4000/DSO4000 features a Pass/Fail test that enables the ability to monitor the change of an input signal and analyze whether the input signal is within a user-defined limit mask. The test results can be displayed on the screen as well as be declared by an audible beep or a pulse signal output from the **[Trig Out/Calibration]** connector at the rear panel.

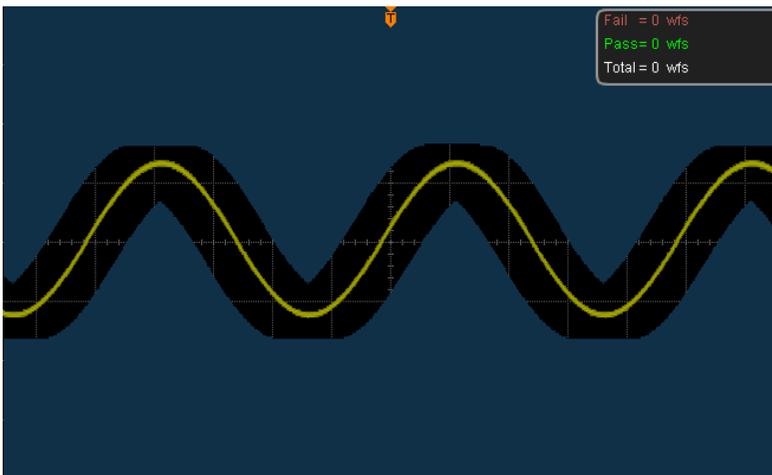
The contents of this chapter:

- [To Enable Pass/Fail Test](#)
- [To Select Source](#)
- [To Create Mask](#)
- [Test and Output](#)
- [To Save the Test Mask](#)
- [To Load the Test Mask](#)

To Enable Pass/Fail Test

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

To start testing, press **Enable Test** and select "ON". Then, press **Operate** to select "▶" to start testing and select "■" to stop testing. The figure below is the test interface:



You can select the source, create mask as well as save and load the test mask. For details, please refer to the following introductions.

To Select Source

Press **Source** to select the channel (CH1 to CH4) to be tested. Only the channels that have been opened can be selected. 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.

To Create Mask

Users can define their own test masks.

Press **Enable Test** to turn the mask display area on. Press **MaskRange** to open the following setting menu.

1. MaskRange

Press **MaskRange** to select "Screen Rigion" or "Cursor Rigion" and the default is "Screen Rigion". After "Cursor Rigion" is selected, two gray cursor lines will be displayed on the screen. At this point, press **Cursor A** and **Cursor B** and use  to adjust the positions of the two gray cursor lines to determine the analysis range. Moreover, press **CursorAB** to adjust the positions of the two gray cursor lines at the same time. Note that you can press  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 4.00 div. During the adjustment, two curves would be displayed to show the outline of the current mask as shown in the figure below.

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.

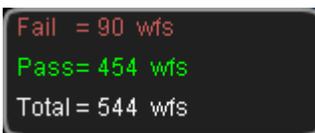
4. Create Mask

The mask is a template used to perform waveform analysis. Press **Create Mask** to immediately apply the mask (X Mask and Y Mask) currently created.

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.



```
Fail = 90 wfs
Pass = 454 wfs
Total = 544 wfs
```

Press **Stop On Output** 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 on the screen (if display is turned on) and only one pulse is output from the **[Trig Out/Calibration]** (if enabled) connector at the rear panel.
- OFF: the oscilloscope will continue with the test even though failed waveforms are detected. The test results on the screen will update continuously and a pulse will be output from the **[Trig Out/Calibration]** connector at the rear panel 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 output of test results from the **[Trig Out/Calibration]** connector at the rear panel. You can also press **Utility** → **AuxOutput** and select "PassFail" to set this output.

To Save the Test Mask

Users can save the current test mask to the internal Flash memory 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 files (LocalPF.pf).

Press **Save** to enter file store interface. Please refer to the relative descriptions 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 internal Flash memory or external USB storage device to the internal memory.

Press **Load** to enter the file recall interface. Please refer to the relative descriptions in "[Store and Recall](#)" to load the test masks to the internal memory of the instrument.

11 Waveform Record

The waveform record feature enables the oscilloscope to record the waveforms of the analog input channels (CH1 to CH4) and the digital channels (D0-D15) much like frames in a movie or video. After they are recorded, they can be replayed in slow motion or real time and also compared and analyzed. In “record constant on” mode, the oscilloscope can record the input waveform continuously until users press **RUN/STOP**. Note: The horizontal time base must be set to Y-T mode during waveform record.

The contents of this chapter:

- [Waveform Record](#)
- [Record Constant On](#)
- [Waveform Playback](#)
- [Waveform Analysis](#)

Waveform Record

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

Press **Record** → **Mode** and use  to select "Record" to open the waveform record operation menu.

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. Record Operation

Waveform record can be realized via the menu or the shortcut buttons at the front panel.

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

3. Interval

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

4. Maximum Number of Frames

The menu shows the maximum number of frames that can be recorded based on the current oscilloscope settings.

Waveform memory capacity is fixed. More points per frame results in a smaller number of waveform frames. Thus, the maximum end frame of waveform record is decided by the "Memory Depth" setting currently selected. Please refer to the instruction in "[LA Sample Rate](#)" to select the desired memory depth.

Table 11-1 Memory Depth and Maximum Number of Frames

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

Record Constant On

MSO4000/DS4000 provides record “Open” (record constant on) mode for waveform record. When this mode is enabled, the oscilloscope records the waveform continuously according to the current memory depth. The length of waveform recorded is limited by the memory depth and the waveform data recorded before will be overwritten by the waveform currently recorded. In this mode, the other oscilloscope operations (except [Pass/Fail Test](#) as well as [Acquisition Mode](#), [Sample Mode](#), [Sample Rate](#) and [Memory Depth](#) in the sample system setting) are not affected.

Press **Record** → **Mode** and use  to select “Open”. At this point, the waveform record key  at the front panel turns red and the oscilloscope records the waveform continuously according to the current memory depth until users press **RUN/STOP**. At this point, users can play back or analyze the waveform.

Waveform Playback

Waveform playback will display the currently recorded frames. Press **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 will change continuously.



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 playback. The range available 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 playback. 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 **Start Frame** value automatically and will change continuously during the playback process.

5. End Frame

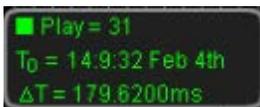
Press this softkey and use  to set the end frame of the playback. The default is the total number of frames of the waveform to be played back.

6. Playback Operation

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

7. Time Tag

The time tag is used to display the absolute recording time of each frame of the waveform currently recorded. Press **Time Tag** to enable or disable the time tag function. When it is enabled, the time tag information is displayed at the upper right corner of the screen as shown in the figure below.



T₀: display the absolute starting time of the waveform record in “Hour:Minute:Second Month Date” format.

ΔT: display the recording time deviation of the current waveform relative to the first frame of waveform. This value changes accordingly during the waveform playback process.

Menu	Front Panel
Press Operate and select “▶” to start playing back.	Press  ; the backlight goes on (in yellow), indicating 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

During waveform playback, **RUN/STOP** can be used to switch between playback 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 **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 refer to the following explanations to set the waveform analysis parameters.

1. Analyze

Press **Analyze** to select the desired analysis mode.

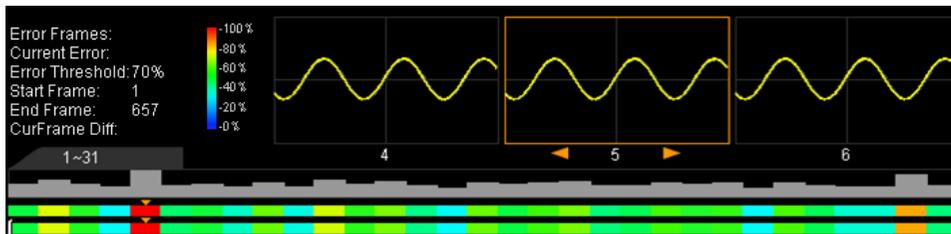
- Trace: perform analysis on the basis of the template selected by users. For detailed information, refer to "[Analysis Based on Trace](#)".
- Pass/Fail: perform analysis on the basis of the Pass/Fail mask created by users. For detailed information, refer to "[Analysis Based on Pass/Fail Mask](#)".

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. The first error frame is located as shown in the figure below. At this point, the next error frame, the previous error frame as well as every frame in this waveform analysis can be located using the corresponding softkeys.



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 trace-based analysis, the oscilloscope compares each frame with the template and computes the difference between each sampled point and the template values. “CurFrame Diff” (relative difference) is the normalized value of the difference between the current frame and template.

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 that 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, pressing **Previous** can locate the error frame previous to the current error frame. Pressing **RUN/STOP** can also perform the operation.

6. Next

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

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

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. Using the navigation knob can also perform the operation.

8. Setup

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

- Screen Start: press this key to set the start point of waveform analysis. The range is from 5 to 685. The start point must be less than the "Screen End" currently set.
- Screen End: press this key to set the end point of waveform analysis. The range is from 15 to 695. The end point must be greater than the "Screen Start" currently set.
- Start Frame: press this key to set the start frame of waveform analysis. The default is the first frame.
- End Frame: press this key to set the end frame of waveform analysis. 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 is recognize as an error one if the (relative) difference value between the frame and the template is equal to or larger than the threshold currently set.

9. TemplateDisp

Press **TemplateDisp** to enable or disable template display. For the template

setting, refer to the introductions in "[Analysis Based on Trace](#)" and "[Analysis Based on Pass/Fail Mask](#)".

Analysis Based on Trace

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

1. Trace

Press **Trace** to select the creation method of analysis template.

- Current Frame: select the current frame as the analysis template.
- Average: select the average of the current data frame as the analysis template.

2. Setup Template

Press **SetupTemplate** to configure the template immediately. After the analysis starts, the oscilloscope compares each frame with the template to compute the difference value and determines whether error frame exists by comparing the difference value 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 template used in analysis based on Pass/Fail mask through the method below.

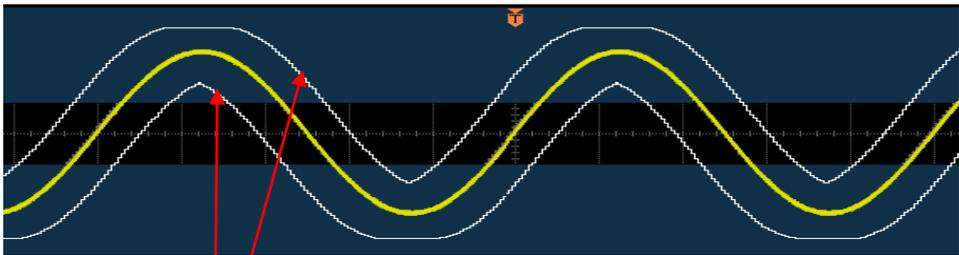
Press **MaskRange**, open the following settings menu.

1. MaskRange

Press **MaskRange** to select "Screen Rigion" or "Cursor Rigion" and the default is "Screen Rigion". After "Cursor Rigion" is selected, two gray cursor lines will be displayed on the screen. At this point, press **Cursor A** and **Cursor B** and use  to adjust the positions of the two gray cursor lines to determine the analysis range. Moreover, press **CursorAB** to adjust the positions of the two gray cursor lines at the same time. Note that you can press  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 4.00 div. During the adjustment, two curves would be displayed to show the outline of the current mask as shown in the figure below.



Outline of the Current Mask

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

Press **Save** to enter the file store 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 recall interface. Load the test mask into the internal memory of the instrument by referring to the introduction in "[Store and Recall](#)".

12 Display Control

You can set the type, persistence time and brightness of the waveform display. Other settings include 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

Press **Display** → **Type** to set the waveform display mode to “Vectors” or “Dots”.

- **Vectors:** the sample points are connected by lines and displayed. Normally, this mode can provide the most vivid waveform to view the steep edge of the waveform (such as square waveform).
- **Dots:** displays 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.

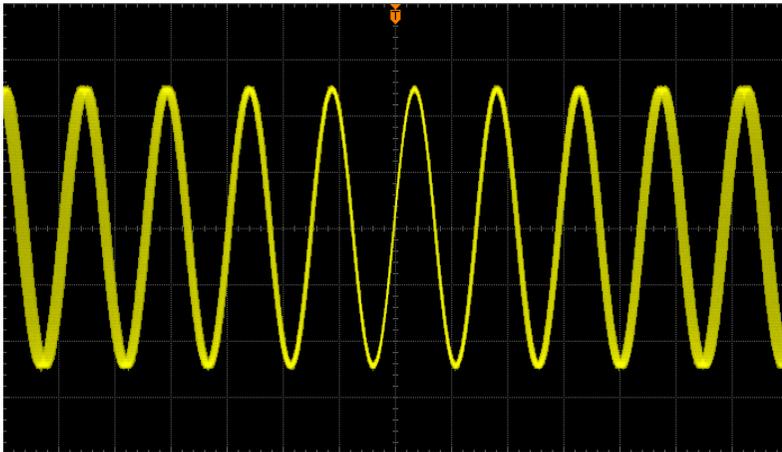
To Set the Persistence Time

Press **Display** → **Persis.Time** to set the persistence time of the oscilloscope to Min, 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.

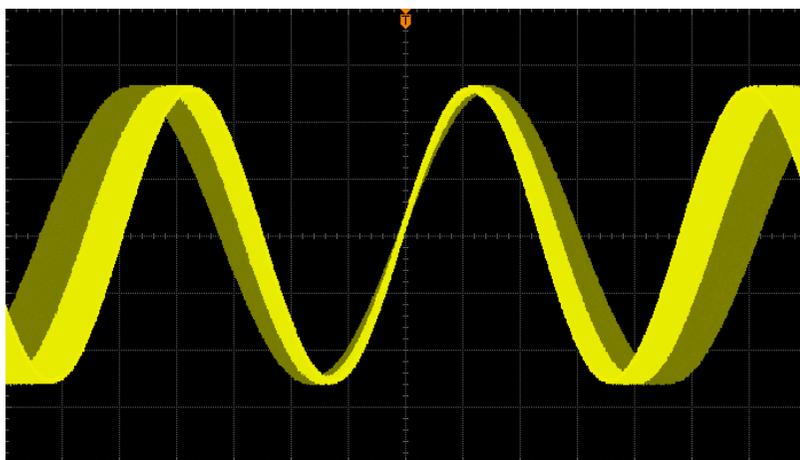
1. Min

Enable to view waveform changing in high refresh rate.



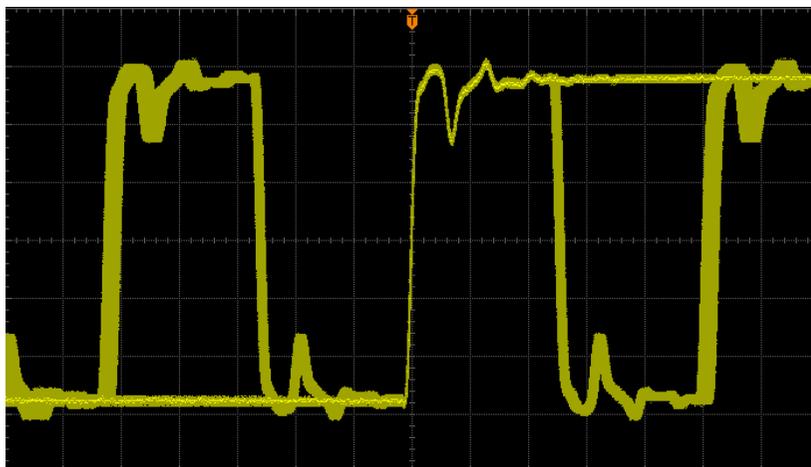
2. Specific Values

Enable to observe glitches that change relatively slowly or glitch with a low repetition rate. 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 newly acquired waveforms 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 **Display** → **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 **Display** → **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 **Display** → **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%.

Note: Reducing the grid brightness can improve waveform display brightness.

To Set the Menu Display

Press **Display** → **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 is hidden. The display time can be set to 1 s, 2 s, 5 s, 10 s, 20 s or Infinite (the menu is not hidden).

13 Store and Recall

Users can save the current settings, waveforms, and screen image of the oscilloscope in internal memory or external USB mass storage device (such as a USB memory disk) in various formats and recall the stored settings or waveforms when needed.

The contents of this chapter:

- [Storage System](#)
- [Storage Type](#)
- [Internal Storage and Recall](#)
- [External Storage and Recall](#)
- [Disk Management](#)
- [Factory](#)

Storage System

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

This oscilloscope provides two USB Host interfaces; one located on the front panel and another on the rear panel. These can be used to connect USB storage devices for external storage. The USB storage device connected is marked as "Disk D" (front panel) and "Disk E" (rear panel). The internal memory (Local Disk) of the instrument can store 10 setting files, 10 reference waveform files and 10 mask files of the Pass/Fail test. Below is the disk selecting interface:

Name	Size/(B)	Time
Up...		
Local Disk	64.0k	
Disk D	3.73G	
Disk E	3.91G	

Table 13-1 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 Record File
	Unknown File		Reference Waveform File
	Bitmap File		Setting File
	CSV File		Trace File

Storage Type

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

1. Traces

Save the waveform data in external memory in "*.trc" format. The data from all active channels can be saved in the same file. At recall, the data will be displayed on the screen directly. Trace files are binary files similar to reference waveforms. They are intended to be transferred and viewed on similar model scopes.

2. Waveforms

Save the waveform data in external memory in "*.wfm" format. 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. Waveform files are binary files that store the instrument settings and waveform data. They are intended to be transferred and viewed on similar model scopes.

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 selecting 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  at the front panel to quickly save the current screen image under the root directory of the USB storage device in bitmap format (*.bmp).

5. CSV

Save the waveform data displayed on the screen or of the specified channels 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 Storage and Recall

Internal storage and recall support “Setups” in **Storage**. In the following section, the storage and recall method and procedures are introduced.

1. Save the specified oscilloscope setting in internal memory.

- 1) Connect the signal to the oscilloscope and obtain a stable waveform.
- 2) Press **Storage** → **Storage** to select “Setups” and press **Save** to turn on the interface as shown in Figure a. use  to select “Local Disk” (the characters turn green) and press down  to open the local disk (Figure b).

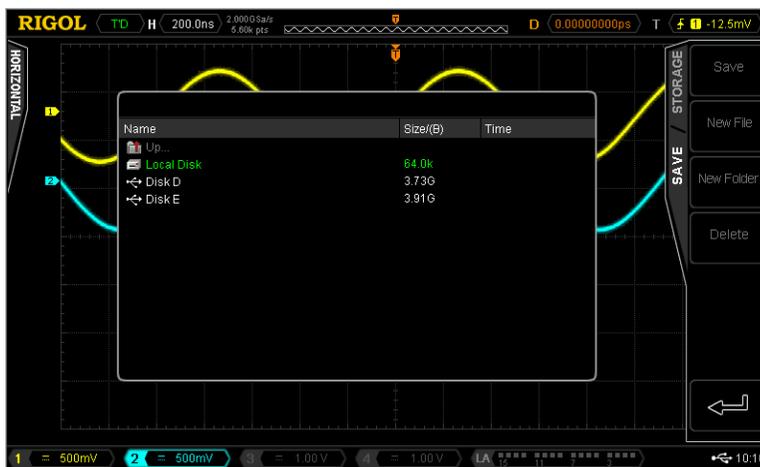
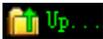


Figure a

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

Use  to select  and then press down  to return to the previous directory.

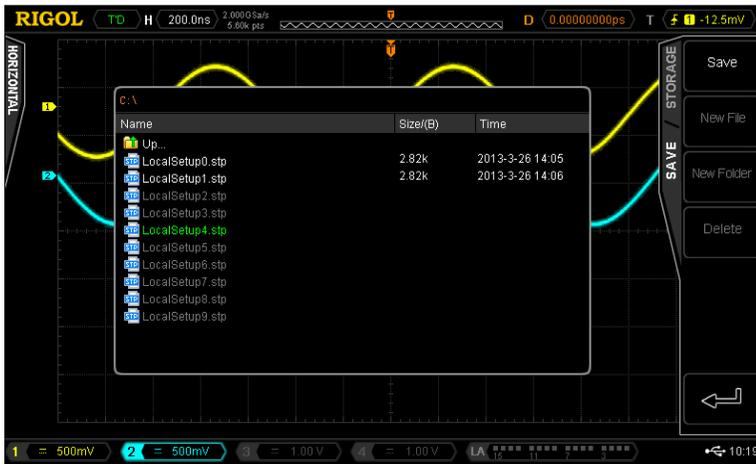


Figure b

Note: In internal storage, **New File** and **New Folder** are not available.

2. Load the specified type of file in internal memory.

- 1) Press **Storage** → **Storage** to select “Setups” 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).

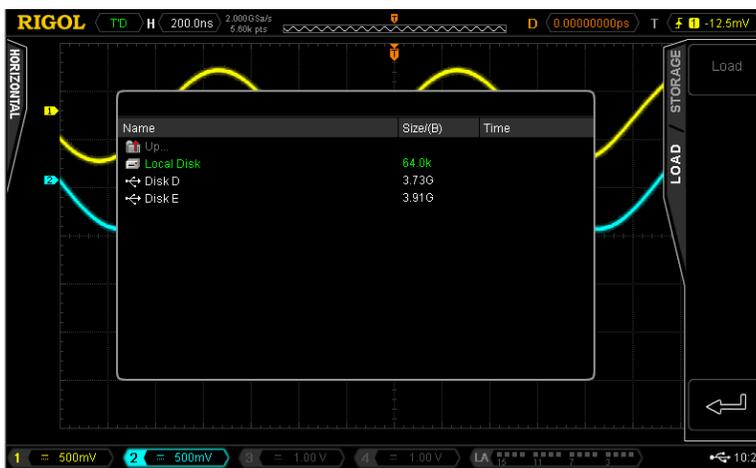


Figure c

- 2) As shown in Figure d, use  to select the desired file to load and press **Load** to load the file selected.

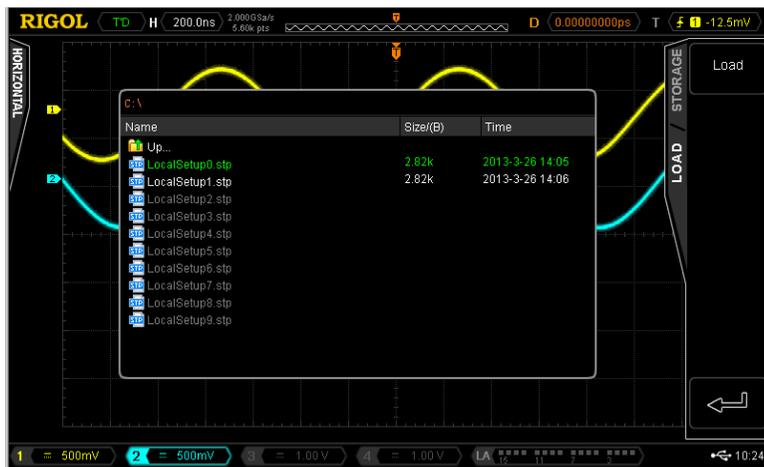


Figure d

External Storage and Recall

Before using external storage and recall, make sure that the 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 **Storage** → **Storage** to select "CSV" and press **Save** to turn on the interface as shown in Figure e. Use ↻ to select "Disk D" 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. The file can be stored 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 turn on 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.

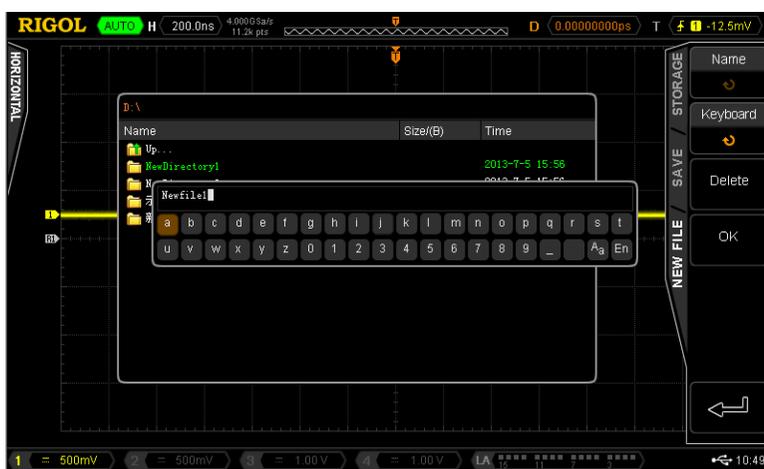


Figure g

- 5) Press **OK** to execute the saving operation.
- 6) If the file is saved in the root directory or in a folder below of the current USB disk, use **↺** to select the file and press **Save**, a message box will be popped up on the screen that prompts you to overwrite the original file, as shown in Figure h, press **OK** to overwrite the original file and then press **↻** to return to the previous menu.



Figure h

2. Load the specified type of file in the external USB storage device.

- 1) Press **Storage** → **Storage** to select “Traces” and then press **Load** to turn on the interface as shown in Figure i. Use ↺ to select “Disk D” and press down ↵ to open the USB storage device (Figure j).



Figure i

- 2) As shown in Figure i, use  to select the desired file to load and then press **Load** to load the selected file.

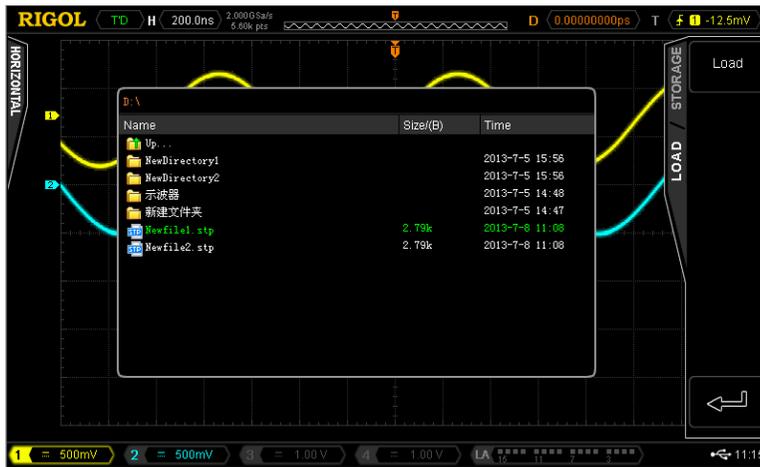


Figure j

Disk Management

Press **Storage** → **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 disk selected.



Name	Size/(B)	Time
Up...		
Local Disk	64.0k	
Disk D	3.73G	
Disk E	3.91G	

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

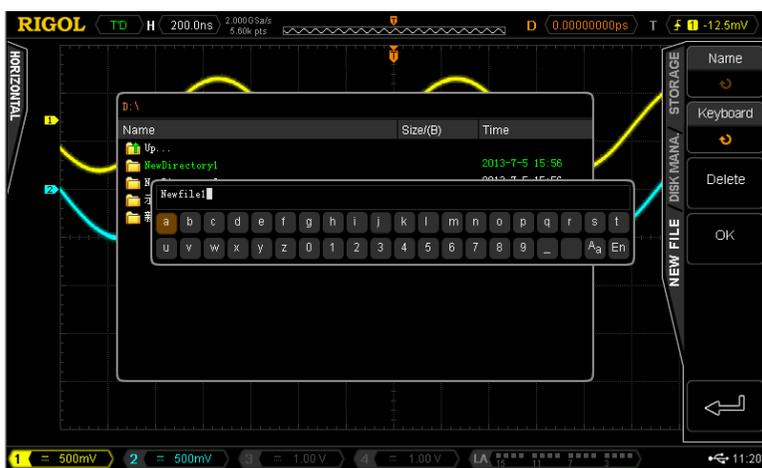
The oscilloscope can also display, save or read some files for advanced applications such as mask files from the Pass/Fail tests (*.pf), waveform record files (*.rec), upgrade files (.rgl), parameter files (*.txt) and reference waveform files (*.ref).

Press **Storage** → **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 an external disk, make sure that the USB storage device is connected correctly.

First, press **Storage** → **Disk.Manage** and use  to select and open the external disk ("Disk D" or "Disk E"). Then, select the desired file type and 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 can contain 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

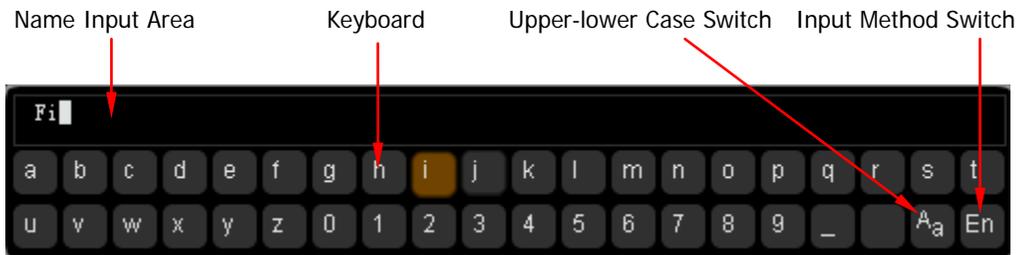
During the name input, use the menu softkeys to select different operation areas, then turn  to select the desired content and press down  to input the content selected.

English Input Method

For example, create a file or folder with the name "Filename".

1. Press **Keyboard**.

- 1) Use  to select English input method "En" and uppercase input state "A".
- 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".



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 name under the current directory.

Chinese Input Method

For example, create a file or folder with the name “文件名”.

1. Press **Keyboard**.

- 1) Use  to select Chinese input method “中”. Note that **Chinese** 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 Character Selecting Area”.
- 3) Press **Chinese** and use  to select and input “文”.
- 4) Use the same method to input “件” and “名”.



2. During the name input, you can press **Name** to select the “Name Input Area” and then press **Delete** to delete the Chinese 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 name under the current directory.

To Delete a File or Folder

Folder operation is valid only in external storage mode. Before using the external disk, make sure that the USB storage device is connected correctly.

1. Delete a file in internal memory.

- 1) Press **Storage** → **Disk.Manage** and use  to select and open the local disk ("local Disk").
- 2) Press **File Type** to select the desired type of file to delete (note that the file types of internal storage include "*.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 **Storage** → **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

The rename operation is valid only in external storage mode. Before using the external disk, make sure that the USB storage device is connected correctly.

Press **Storage** → **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 turn on the rename interface. For specific operations, please refer to the descriptions in "[To Create a New File or Folder](#)".

To Clear the Local Memory

Press **Storage** → **Disk.Manage** and select "Local Disk", and open the next menu page, then press **Delete** → **OK** to delete all the files and settings stored in the local memory and restore the instrument to the default settings.

Factory

Press **Default** to return the oscilloscope to its factory state (refer to the table below).

Table 13-2 Factory

Parameter	Factory
Horizontal Setting (HORIZONTAL)	
Horizontal Time Base	1 μ s
Horizontal Offset	0 s
Delayed Sweep	OFF
Time Base Type	Y-T
Time Base Scale	Coarse
Horizontal Reference	The Center of the Screen
Vertical 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 M Ω
Channel Invert	OFF
Vertical Scale	Coarse
Channel Unit	V
Acquisition Setting (Acquire)	
Acquisition Mode	Normal
Sampling Mode	Real Time
Sa Rate	4.0 GSa/s
La SaRate	1.0 GSa/s

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
Noise Reject	OFF
Display Setting (Display)	
Display Type	Vectors
Persistence Time	Min
Waveform Intensity	50%
Screen Grid	
Brightness	50%
Menu Display	Infinite
Cursor Setting (Cursor)	
Mode	None
Manual	
Display Type	X-Y
Source Select	CH1
Select Cursor	X
Time Unit	s
Vertical Unit	Source Unit
CurA	-4*1 μ s
CurB	4*1 μ s
Track	
Cursor A	CH1
Cursor B	CH1
CurA	-4*1 μ s
CurB	4*1 μ s

Storage Setting (Storage)	
Storage Type	Picture
Picture Type	png
Parameter Save	OFF
Inverted	OFF
Color Setting	Color
Header Setting	OFF
Footer Setting	OFF
Utility Function Setting (Utility)	
I/O Setting	
Network Configuration Mode	DHCP, Auto IP
USB Device	Computer
GPIB	1
Sound	
Sound	OFF
Pass/Fail Test	
Enable Test	OFF
Source	CH1
Operate	OFF
MaskRange	Screen Region
X Mask	0.24 div
Y Mask	0.38 div
Message Display	OFF
Stop On Output	OFF
Output	Fail
Axu Output	OFF
Waveform Record (Record)	
Mode	OFF
Record	
End Frame	7996
Operate	OFF
Interval	100 ns
Total Frames	7996
Play back	
Operate	OFF

Play Mode	Single
Interval	100 ns
Start Frame	1
Analyze Mode	
Analyze Mode	Trace
Source	CH1
Trace	Current Frame
Current Frame	1
Template Display	ON
System Setting	
Power-off Recall	Default
Vertical Expansion	Ground
Screen Saver	Default
Screen Saver Time	OFF
External Trigger Impedance	1 M Ω
Aux	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
Vertical Invert	OFF
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
Horizontal Scale	1.250 MHzdiv
Sample Rate	50 MSa/s
Anti-Aliasing	OFF
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
Invert	OFF
Vertical Scale	2 V
Logic Analyzer (LA)	
CH/Group	D0
On/Off	D7-D0: Turn Off; D15-D8: Turn Off
WaveSize	M
ReOrder	D0-D15
Level Type	TTL
Bus	BUS1
Bus Status	Off
Sel CH	D7-D0
Endian	LSB

Reference Clock	None
Slope Type	Rising Edge
Display Type	Data
Format	Hex
JitterReject	Turn On
JitterTime	5.00 ns
Protocol Decoding (Decode 1, Decode 2)	
Decoding Type	Parallel
BUS Status	ON
Format	Hex
Offset	0
Threshold	0
Parallel	
Clock Channel	None
Slope	Rising Edge
Bus Bits	8
Current Bit	0
Channel	D0
RS232	
TX	CH1
RX	None
Polarity	Normal
Endian	LSB
Baud	9600 bps
Data Bits	8
Stop Bit	1
Even-Odd Check	None
Packet	OFF
Packet End	NULL
I2C	
SCLK	CH1
SDA	CH2
Include R/W	Close
SPI	
Mode	TimeOut

TimeOut	1.00 μ s
SCLK Channel	CH1
SCLK Slope	Rising Edge
MISO Channel	CH2
MISO Polarity	High level is 1.
MOSI Channel	None
MOSI Polarity	High level is 1
Data Bits	8
Endian	MSB
CAN	
Source	CH1
Signal Type	CAN_H
Baud	500 kb/s
Sample Point	50%
FlexRay	
Source	CH1
Signal Path	A
Signal Type	BP
Baud	10Mb/s
Sample Point	50%
Reference Waveform Setting (REF)	
Channel Setting	REF1
Current Channel	REF1
Source	CH1
Color	Gray

14 System Function Setting

The contents of this chapter:

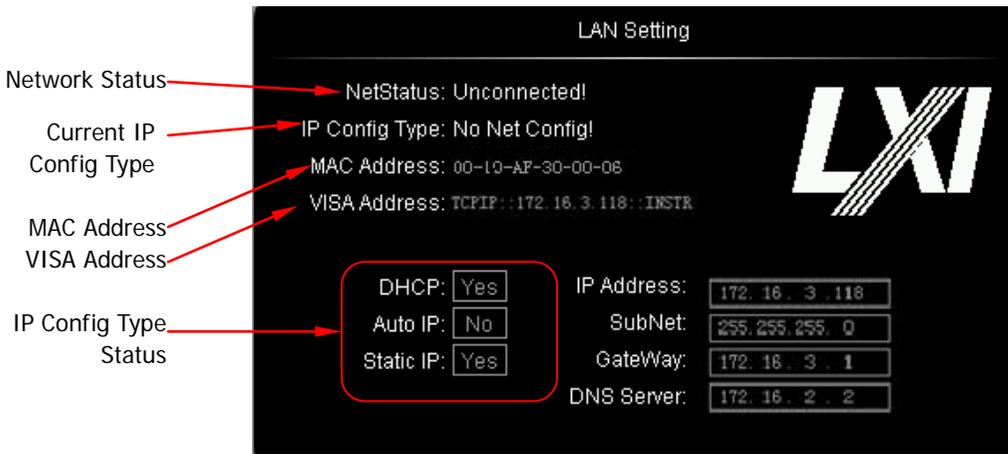
- [Remote Interface Configuration](#)
- [System-related](#)

Remote Interface Configuration

The MSO4000/DS4000 can communicate with PC via LAN, USB and GPIB (with the USB-GPIB interface converter provided by **RIGOL**) buses. Please refer to the introduction below to configure the corresponding interface before using the remote interfaces.

LAN Setting

Press **Utility** → **IO Setting** → **LAN Set** to turn on 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 DHCP 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 (DHCP is turned off), **Gate** and **DNS** are added to the right side of the screen. 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, turn DHCP and auto IP off manually and **IP Address**, **Subnet 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 IP address is nnn.nnn.nnn.nnn; wherein, 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-off Recall** 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; wherein, 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-off Recall** 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 parameter in **Auto IP** and **Static IP** mode.

The format of the gate is nnn.nnn.nnn.nnn; wherein, 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-off Recall** 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 parameter In **Auto IP** and **Static IP** mode.

The address format of the domain name server is nnn.nnn.nnn.nnn; wherein, 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 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 PC or 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 **Utility** → **IO Setting** → **USB Device** and use  to select the desired device type.

- Computer: in this mode, the oscilloscope can communicate with the PC.
- PictBridge: in this mode, the oscilloscope can communicate with a PictBridge printer.

To Set the GPIB Address

When using the GPIB mode to control the oscilloscope, you need to extend a GPIB interface for the oscilloscope using the USB-GPIB interface converter (need to be ordered separately).

To set the GPIB address of this interface, press **Utility** → **IO Setting** → **GPIB** and use  to input the desired address. The default is 1 and the range is from 1 to 30.

System-related

Sound

When sound is enabled, you can hear the sound of a beeper when you press a function key or a menu softkey or when the prompt message pops up.

Press **Utility** → **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.

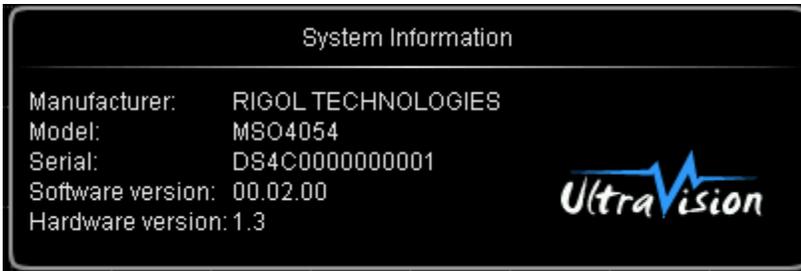
Language

This oscilloscope supports multiple language menus, Chinese/English help and prompt messages.

Press **Utility** → **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 **Utility** → **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 **Utility** → **System** → **Power On** to select "Last" (default) or "Default".

- Last: return to the setting of the system at 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 **Utility** → **System** → **System Time** → **System Time** to turn on the setting interface as follows:



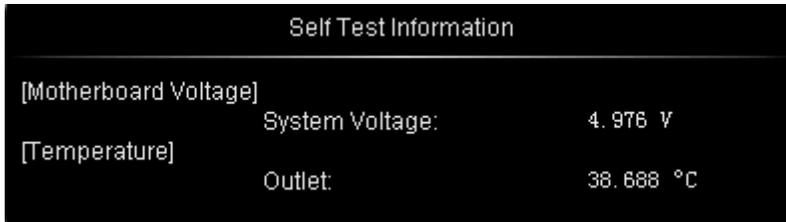
In the figure, the item in green (such as 2013) is the item that can be modified currently. Use  to modify and press down  to complete the input. The order of time modifications is: year (2013) → month (07) → date (08) → hour (14) → minute (34) → second (19). The setting range of each item conforms to 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-test Information

Press **Utility** → **System** → **SelfTestInfo** to view the result of the last self-test of the oscilloscope. The self-test result usually contains the contents as shown in the figure below.



Self Test Information		
[Motherboard Voltage]	System Voltage:	4.976 V
[Temperature]	Outlet:	38.688 °C

Screen

When the oscilloscope enters the idle state and holds for a certain period of time, the screen saver program will be enabled.

Press **Utility** → **System** → **Screensaver** to open the screen saver setting menu. "Default" means using the icon **RIGOL** as the screen saver icon.

Press **Time** to select the screen saver time. When "OFF" is selected, the screen saver program is disabled. The screen saver time 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

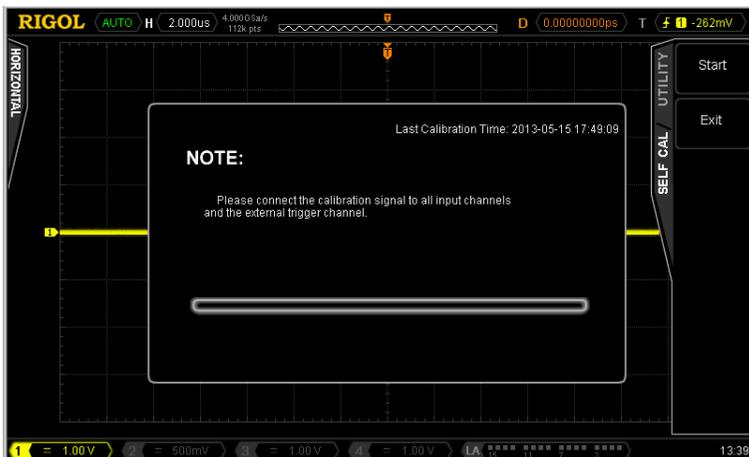
If this menu is displayed in gray, it means that the circuit board of the oscilloscope is working normally. Otherwise, an exclamation mark will be displayed in the status bar at the lower right corner of the screen, indicating that there is error information currently and at this point, press this key (press **Utility** → **System** → **ErrorInfo**) to view the error information.

The error information that might appear includes 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

The self-calibration routine optimizes performance of the oscilloscope and helps maintaining measurement accuracy. You can perform self-calibration at any time, but it is especially recommended when the temperature changes up to or more than 5 °C. Make sure that the oscilloscope has been warmed up or operated for more than 30 minutes before performing the self-calibration.

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 **Utility** → **Self-Cal** and the self-calibration interface as shown in the figure below is displayed.



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. At this point, the oscilloscope is powered on.

Press **Utility** → **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 that at this point, you can also 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 **Utility** → **Extimpedance** to select "1 M Ω " (default) or "50 Ω ".

Aux Output

Users can set the type of the signal output from the **[Trig Out/Calibration]** connector at the rear panel.

Press **Utility** → **AuxOutput** to select the desired output type.

1. TrigOut

After this type is selected, the oscilloscope outputs a signal that can reflect the current capture rate of the oscilloscope at each trigger. Connect the signal to the waveform display device and measure the frequency of the signal, the measurement result is the same as the current capture rate.

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 TTL high level when failed waveforms are detected and low level at the beginning of the next waveform. This signal can be connected 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 **Utility** → **RefClock** to select the desired clock type.

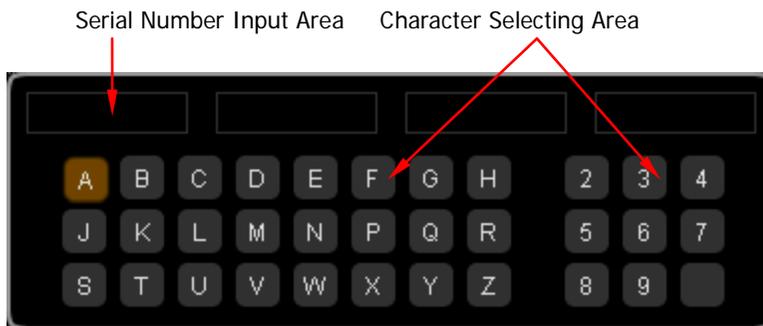
- ClockOutput: configure the **[10MHz In/Out]** connector as an output to output the internal 10 MHz clock.
- ClockInput: configure the **[10MHz In/Out]** connector as an input and the oscilloscope will accept a 10 MHz clock input from this connector.

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 this menu.

Press **Utility** → **Options** → **Installed** to view the options currently installed on the oscilloscope and the related information. Press **Setup** to enter the serial number activation operation menu.

- Editor: press this softkey to turn on 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.



- 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.

15 Remote Control

The oscilloscope can be controlled remotely mainly through 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** of **RIGOL**, **Measurement & Automation Explorer** of **NI** (National Instruments Corporation) or **Agilent IO Libraries Suite** of **Agilent** (Agilent Technologies, Inc.) to send commands to control the oscilloscope remotely.

This oscilloscope can communicate with PC through USB, LAN and GPIB (with the USB-GPIB interface converter provided by **RIGOL**) instrument buses. This chapter will give a detailed introduction of how to use **Ultra Sigma** to control the oscilloscope remotely through various interfaces. For the **Ultra Sigma** software, please contact **RIGOL** salesmen or technical support.

The contents of this chapter:

- [Remote Control via USB](#)
- [Remote Control via LAN](#)
- [Remote Control via GPIB](#)

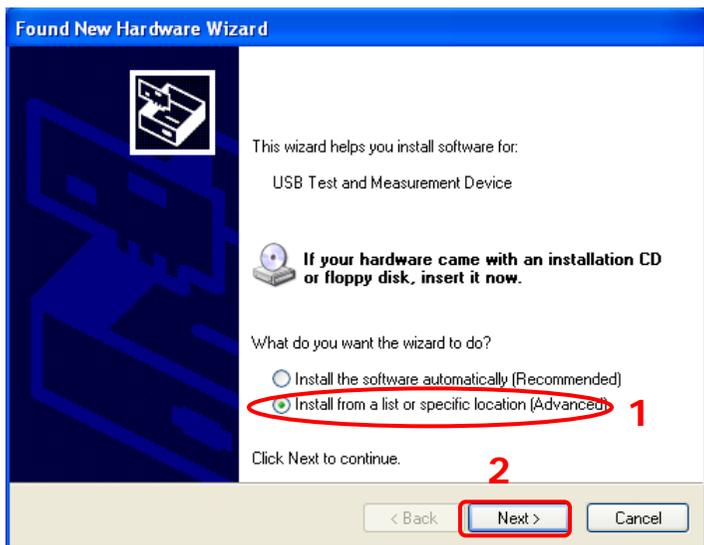
Remote Control via USB

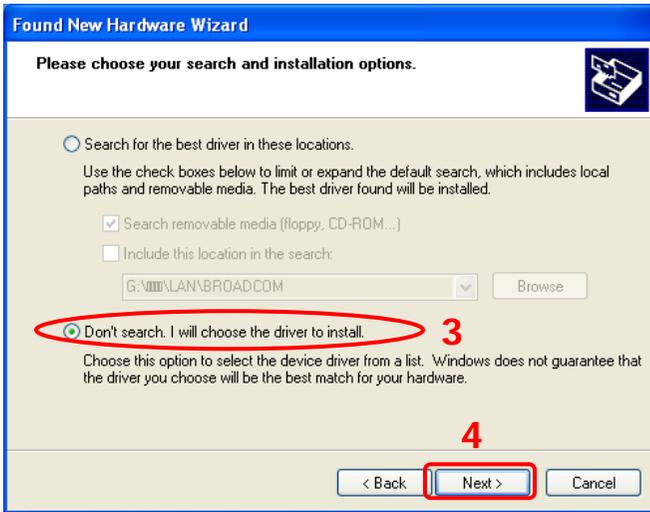
1. Connect the device

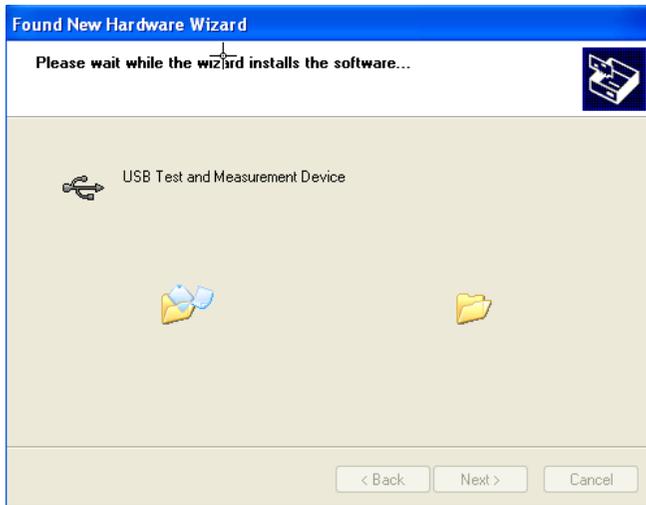
Connect the oscilloscope (USB DEVICE) with your PC using a USB cable.

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 is automatically configured to USB interface), the **New Hardware Wizard** as shown in the figure below is displayed on the PC. 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

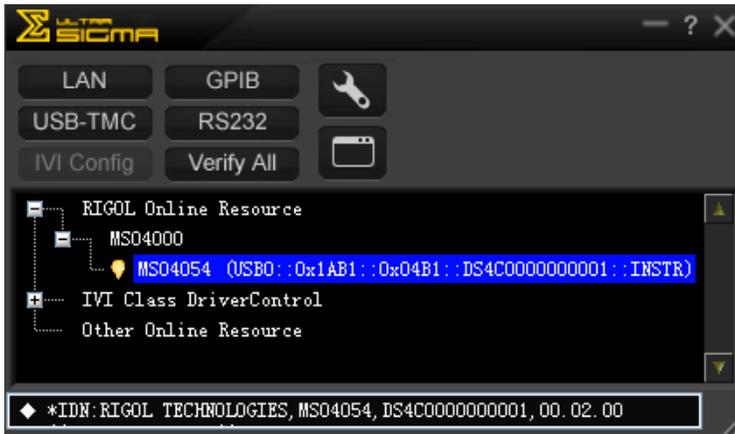
USB-TMC to search the resources. During the search, the status bar of the software is as shown in the figure below:



4. View the device resource

The resources found will appear under the "RIGOL Online Resource" directory

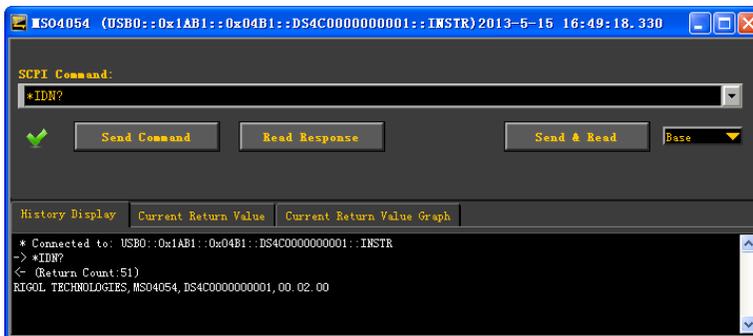
and the model number and USB interface information of the instrument will also be displayed as shown in the figure below.



5. Communication test

Right click the resource name

“MSO4054 (USB0::0x1AB1::0x04B1::DS4C0000000001::INSTR)” to select “SCPI Panel Control” to turn on the remote command control panel (as shown in the figure below) through which you can send commands and read data.



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** (as shown in Figure (c)).

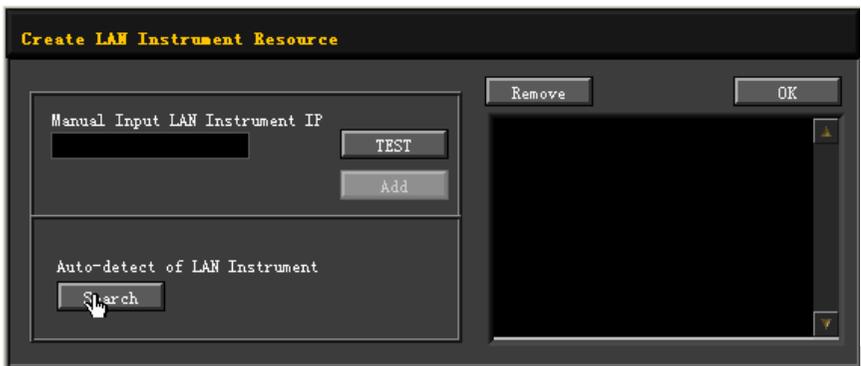


Figure (a)

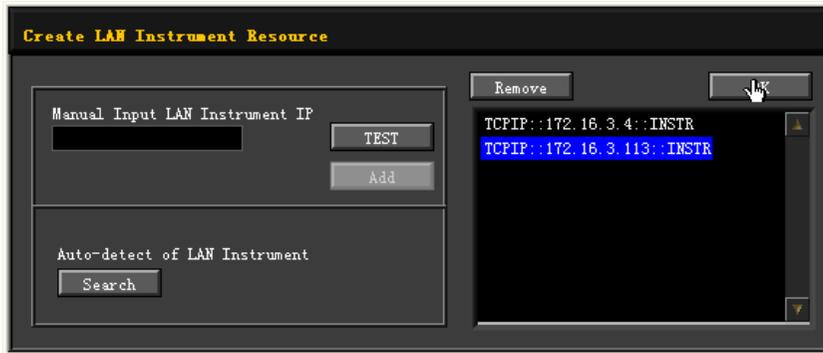


Figure (b)

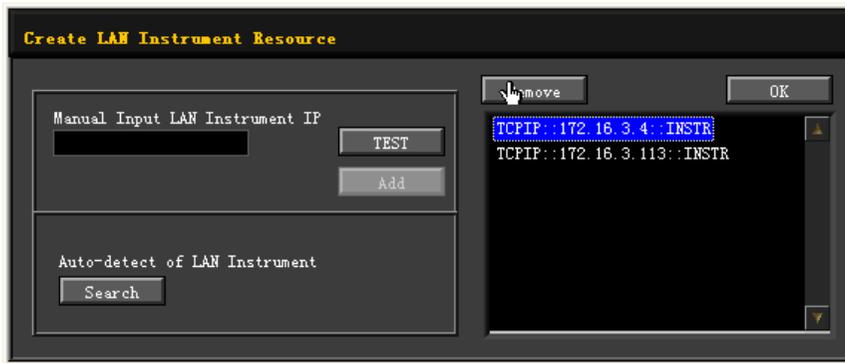


Figure (c)

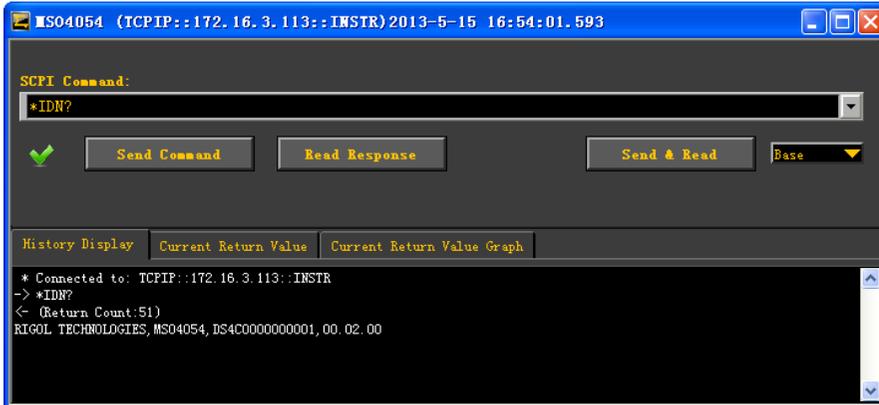
4. View device resource

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 “MSO4054 (TCPIP::172.16.3.113::INSTR)” to select “SCPI Panel Control” to turn on the remote command control panel (as shown in the figure below) through which you can send commands and read data.



6. Load LXI webpage

As this oscilloscope conforms to LXI-C standards, you can load LXI webpage through **Ultra Sigma** (right-click the resource name and select 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.

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 "System Function Setting→ Remote Interface Configuration→ 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.

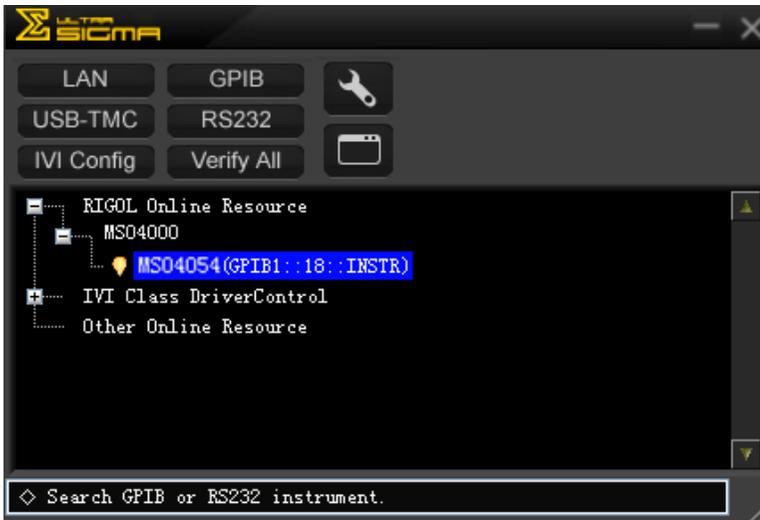


If resources can not be found automatically:

- 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.

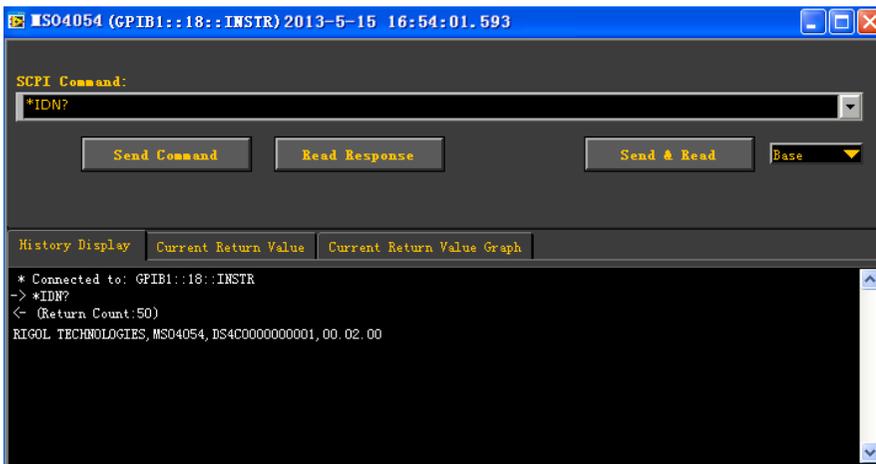
5. View device resource

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 "MSO4054 (GPIB0::18::INSTR)" to select "SCPI Panel Control" to turn on the remote command control panel through which you can send commands and read data as shown in the figure below.



16 Troubleshooting

Some 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 (**Utility** → **System** → **System Info**).

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

- (1) Check whether the power is correctly connected.
- (2) Check whether the power switch is really on.
- (3) Check whether the fuse is burned out. If the fuse needs to be changed, please return the instrument to the factory and the **RIGOL** authorized personnel will change the fuse for you.
- (4) Restart the instrument after finishing the above inspections.
- (5) 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 item to be tested.
- (2) Check whether there are signals generated from the item to be tested (you can connect the probe compensation signal to the problematic channel to determine which has problem, the channel or the item to be tested).
- (3) 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 trigger 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) Check the trigger level: adjust the trigger level to the middle position of the

signal.

- (4) 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.
- (5) Change the trigger holdoff setting.

5. No display after pressing RUN/STOP:

Check whether **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 **MODE** to "Auto".

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

6. The waveform displayed 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 the display **Type** to "Dots" to solve the problem.

7. Digital waveform is not stable:

- (1) Check whether the signal input from the selected trigger source is synchronized with the digital signal.
- (2) Adjust the oscilloscope to select the appropriate trigger mode, time base and other general settings.
- (3) If the display is still not stable, please check the electrical connections and parameter settings again.
- (4) Try to use other probe (such as analog probe) to verify the signal status of the test point.

8. Fail to connect PC or PictBridge printer through USB:

- (1) Check the **IO Setting** in **Utility** to make sure whether the setting in **USB Device** matches the device currently connected.
- (2) Check whether the USB cable is properly connected to the oscilloscope and PC.
- (3) Check whether the USB cable is in good condition. If necessary, restart the oscilloscope.

9. 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 8 GBytes.
- (4) Restart the instrument and then insert the USB storage device to check it.
- (5) If the USB storage device still can not be used normally, please contact **RIGOL**.

17 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
Real Time Sample Rate	Analog channel: 4.0 GSa/s (single-channel), 2.0 Gsa/s (dual-channel) Digital channel: 1.0 Gsa/s
Peak Detect	Analog channel: 250 ps (single-channel); 500 ps (dual-channel) Digital channel: 1 ns
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 $\geq 5 \mu\text{s}/\text{div}$ @ 4 GSa/s (or $\geq 10 \mu\text{s}/\text{div}$ @ 2 GSa/s).
Minimum Detectable Pulse Width	Digital channel: 5 ns
Memory Depth	Analog channel: 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 Digital channel: maximum 28M pts

Input

Number of Channels	MSO40X4: four-analog-channel + 16-digital-channel MSO40X2: dual-analog-channel + 16-digital-channel DS40X4: four-channel DS40X2: dual-channel
Input Coupling	DC, AC or GND
Input Impedance	Analog channel: $(1\text{ M}\Omega \pm 1\%) \parallel (14\text{ pF} \pm 3\text{ pF})$ or $50\ \Omega \pm 1.5\%$ Digital channel: $(101\text{ k}\Omega \pm 1\%) \parallel (9\text{ pF} \pm 1\text{ pF})$
Probe Attenuation Coefficient	Analog channel: 0.01X to 1000X, in 1-2-5 step
Maximum Input Voltage (1M Ω)	Analog channel (1 M Ω) : CAT I 300 Vrms, CAT II 100 Vrms, Transient Overvoltage 1000 Vpk 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 Digital channel: CAT I 40 Vrms, transient overvoltage 800 Vpk

Horizontal

Time Base Scale	MSO405X/DS405X: 1 ns/div to 1000 s/div MSO403X/DS403X: 2 ns/div to 1000 s/div MSO402X/DS402X: 2 ns/div to 1000 s/div MSO401X/DS401X: 5 ns/div to 1000 s/div
Time Base Accuracy	$\leq \pm 4\text{ ppm}$
Time Base Drift	$\leq \pm 2\text{ ppm/year}$
Delay Range	Pre-trigger (negative delay): ≥ 1 screen width Post-trigger (positive delay): 1 s to 1000 s
Time Base Mode	Y-T, X-Y, Roll, Delayed
Number of X-Ys	2 paths at the same time (four-channel model)
Waveform Capture Rate ¹	110,000 wfms/s (digital channels are closed) or 85,000 wfms/s (digital channels are opened) (dots display)

Vertical

Bandwidth (-3dB)	MSO405X/DS405X: DC to 500 MHz MSO403X/DS403X: DC to 350 MHz MSO402X/DS402X: DC to 200 MHz MSO401X/DS401X: DC to 100 MHz
Single Bandwidth	MSO405X/DS405X: DC to 500 MHz MSO403X/DS403X: DC to 350 MHz MSO402X/DS402X: DC to 200 MHz MSO401X/DS401X: DC to 100 MHz
Vertical Resolution	Analog channel: 8 bits, two channels sample at the same time Digital channel: 1 bit
Vertical Scale	1 mV/div to 5 V/div (1 M Ω) 1 mV/div to 1 V/div (50 Ω)
Offset Range	1 mV/div to 124 mV/div: $\pm 1.2V$ (50 Ω) 126 mV/div to 1 V/div: $\pm 12V$ (50 Ω) 1 mV/div to 225 mV/div: $\pm 2V$ (1M Ω) 230 mV/div to 5 V/div: $\pm 40V$ (1M Ω)
Dynamic Range	± 5 div
Bandwidth Limit ²	MSO405X/MSO403X/DS405X/DS403X: 20 MHz/100 MHz/200 MHz MSO402X/DS402X: 20 MHz/100 MHz MSO401X/DS401X: 20 MHz
Low Frequency Response (AC Coupling -3dB)	≤ 5 Hz (on BNC)
Calculated Rise Time ²	MSO405X/DS405X: 700 ps MSO403X/DS403X: 1 ns MSO402X/DS402X: 1.8 ns MSO401X/DS401X: 3.5 ns
DC Gain Accuracy	$\pm 2\%$ full scale
DC Offset Accuracy	200 mV/div to 5 V/div: 0.1 div ± 2 mV $\pm 0.5\%$ offset 1 mV/div to 195 mV/div: 0.1 div ± 2 mV $\pm 1.5\%$ offset
ESD Tolerance	± 2 kV
Channel to Channel Isolation	DC to maximum bandwidth: > 40 dB

Vertical (Digital Channel)

Threshold	1 group with 8 channels adjustable threshold
LevelType	TTL (1.4 V)
	5.0 V CMOS (+2.5 V), 3.3 V CMOS (+1.65 V)
	2.5 V CMOS (+1.25 V), 1.8 V CMOS (+0.9 V)
	ECL (-1.3 V)
	PECL (+3.7 V)
	LVDS (+1.2 V)
	0 V
	User
Threshold range	$\pm 20.0V$, with 10 mV step
Threshold accuracy	$\pm (100 \text{ mV} + 3\% \text{ of threshold setting})$
Dynamic range	$\pm 10 \text{ V} + \text{threshold}$
Min Voltage Swing	500 mVpp
Vertical resolution	1 bit

Trigger

Trigger Level Range	Internal	± 6 div from center of the screen
	EXT	$\pm 0.8 \text{ V}$
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
Runt Trigger	
Pulse Polarity	Positive, Negative
Qualifier	None, >, <, <>
Nth Edge Trigger	
Edge Type	Rising, Falling
Idle Time	40 ns to 1 s
Number of Edges	1 to 65535
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	
Polarity	Positive, Negative
Synchrony	All Lines, Line Num, Odd Field, Even Field
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	
Polarity	Normal, Invert
Trigger Condition	Start, Error, Check Error, Data
Baud	2400bps, 4800bps, 9600bps, 19200bps, 38400bps, 57600bps, 115200bps, 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 (Chip Select), Timeout
Timeout Value	100 ns to 1 s

Data Bits	4 bit to 32 bit
Data Line Setting	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, Frame Error
Baud	10kbps, 20kbps, 33.3kbps, 50kbps, 62.5kbps, 83.3kbps, 100kbps, 125kbps, 250kbps, 500kbps, 800kbps, 1Mbps, User
Sample Point	5% to 95%
Frame Type	Data, Remote, Error, OverLoad
FlexRay Trigger	
Baud	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, Suspend, Exit Suspend

Measure

Cursor	Manual Mode	Voltage Deviation between Cursors (ΔV) Time Deviation between Cursors (ΔT) Reciprocal of ΔT (Hz) ($1/\Delta T$)
	Track Mode	Voltage and Time Values of the Waveform Point
	Auto Mode	Allow to display cursors during auto measurement
Auto Measurement	Analog channel: 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 $A \rightarrow B_f$, Delay $A \rightarrow B_t$, Phase $A \rightarrow B_f$, Phase $A \rightarrow B_t$ Digital channel:	

	Frequency, Period, Positive Pulse Width, Negative Pulse Width, Positive Duty Cycle, Negative Duty Cycle, Delay $A \rightarrow B f$, Delay $A \rightarrow B t$, Phase $A \rightarrow B f$, Phase $A \rightarrow B t$
Number of Measurements	Display 5 measurements at the same time.
Measurement Range	Screen Region or Cursor Region
Measurement Statistic	Average, Max, Min, Standard Deviation, Number of Measurements
Frequency Counter	Hardware 6 bits frequency counter (channels are selectable)

Math Operation

Waveform Operation	$A+B$, $A-B$, $A \times B$, A/B , FFT, Editable Advanced Operation, Logic Operation
FFT Window	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 (MSO4XX4/DS4XX4 option), CAN (option), FlexRay (option)

Display

Display Type	9 inches (229 mm) TFT LCD display
Display Resolution	800 Horizontal \times RGB \times 480 Vertical Pixel
Display Color	160,000 Color
Persistence Time	Min, 50ms, 100ms, 200ms, 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, PassFail, GND)
----------------	--

General Specifications**Probe Compensation Output**

Output Voltage ²	About 3 V, peak-peak
Frequency ²	1 kHz

Power

Power Voltage	100 to 127 V, 45 to 440Hz 100 to 240 V, 45 to 65Hz
Power	Maximum 120W
Fuse	3 A, T Degree, 250 V

Environment

Temperature Range	Operating: 0 °C to +50 °C
	Non-operating: -40 °C to +70 °C
Cooling Method	Fan
Humidity Range	0 °C to +30 °C: ≤95% relative humidity
	+30 °C to +40 °C: ≤75% relative humidity
	+40 °C to +50 °C: ≤45% relative humidity
Altitude	Operating: under 3,000 meters
	Non-operating: under 15,000 meters

Physical Characteristics

Size ³	Width×Height×Depth = 440.0 mm× 218.0 mm×130.0 mm	
Weight	Package Excluded	4.8 kg ± 0.2 kg
	Package Included	7.1 kg ± 1.0 kg

Adjustment 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

- 1** Maximum value. In single-channel mode, sine signal with 10 ns horizontal scale, 4 div input amplitude and 10 MHz frequency, edge trigger.
- 2** Typical.
- 3** Supporting legs and handle folded, knob height included, front panel cover excluded.

18 Appendix

Appendix A: Accessories and Options

	Description	Order Number
Model	MSO4054 (500 MHz, 4+16 channels mixed signal oscilloscope)	MSO4054
	MSO4052 (500 MHz, 2+16 channels mixed signal oscilloscope)	MSO4052
	MSO4034 (350 MHz, 4+16 channels mixed signal oscilloscope)	MSO4034
	MSO4032 (350 MHz, 2+16 channels mixed signal oscilloscope)	MSO4032
	MSO4024 (200 MHz, 4+16 channels mixed signal oscilloscope)	MSO4024
	MSO4022 (200 MHz, 2+16 channels mixed signal oscilloscope)	MSO4022
	MSO4014 (100 MHz, 4+16 channels mixed signal oscilloscope)	MSO4014
	MSO4012 (100 MHz, 2+16 channels mixed signal oscilloscope)	MSO4012
	DS4054 (500 MHz, 4-channel)	DS4054
	DS4052 (500 MHz, dual-channel)	DS4052
	DS4034 (350 MHz, 4-channel)	DS4034
	DS4032 (350 MHz, dual-channel)	DS4032
	DS4024 (200 MHz, 4-channel)	DS4024
	DS4022 (200 MHz, dual-channel)	DS4022
	DS4014 (100 MHz, 4-channel)	DS4014
	DS4012 (100 MHz, dual-channel)	DS4012

Accessories and Options (Continued):

Standard Accessories	Power Cord conforming to the standard of the country	--
	Front Panel Cover	FPCS-DS4000
	USB Cable	CB-USBA-USBB-FF-150
	2 or 4 Passive Probes (500 MHz)	RP3500A
	1 set of logic analysis probe (only for MSO4000 model)	RPL2316
	Quick Guide	--
	Resource CD (User's Guide and Application Software)	--
Optional Accessories	Active Differential Probe (1.5 GHz)	RP7150
	Rack Mount Kit	RM-DS4000
Decoding Options	RS232/UART Decoding Kit	SD-RS232-DS4000
	I2C Decoding Kit	SD-I2C-DS4000
	SPI Decoding Kit	SD- SPI-DS4000
	CAN Decoding Kit	SD-CAN-DS4000
	FlexRay Decoding Kit	SD-FlexRay-DS4000

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. To get repair service, 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.

Index

- Duty 6-15
- Width 6-15
- + Duty 6-15
- + Width 6-15
- Acquisition Mode 4-2
- Addition 6-2
- AND 6-8
- Antialiasing 4-1, 4-10
- Auto 5-3
- Auto IP 14-3
- Auto Mode 6-34
- Aux Output 14-14
- Average 4-2
- Bandwidth Limit 2-4
- Blackman 6-6
- CAN Decoding 8-1, 8-16
- Channel Coupling 2-3
- Channel Label 2-10
- CSV 13-4
- Cursors 6-26
- Decoding Table 8-4
- Delay 6-16
- Delay Calibration 2-11
- Delayed Sweep 3-2
- DHCP 14-3
- Division 6-4
- Domain Name Serve 14-5
- Dots 12-2
- Edge Trigger 5-9
- Edge Type 5-15
- Expression 6-10
- Factory 13-19
- Fall Time 6-15
- FFT 6-5
- FlexRay Decoding 8-1, 8-19
- Frequency 6-15
- Frequency Counter 6-19
- GPIOB 14-6
- Hamming 6-6
- Hanning 6-6
- High Resolution 4-4
- Horizontal Reference 3-10
- Horizontal Scale 3-9
- I2C Decoding 8-1, 8-10
- Idle Time 5-15
- Input Impedance 2-7
- IP Address 14-4
- LAN setting 14-2
- MAC 14-5
- Manual Mode 6-27
- Memory Depth 4-9
- Multiplication 6-3
- Network Status 14-2
- Normal 4-2, 5-4
- NOT 6-8
- Nth Edge Trigger 5-15
- Operator 6-12
- OR 6-8
- Overshoot 6-17
- Parallel Decoding 8-2
- Pattern Setting 5-22
- Pattern Trigger 5-22
- Peak Detect 4-4
- Period 6-15
- Persistence Time 12-3
- Phase 6-16
- Picture 13-3
- Preshoot 6-17

Pulse Condition	5-11	Trigger Position	3-10
Pulse Polarity	5-13	Trigger Source	5-2
Pulse Trigger	5-11	USB	14-6
Pulse Width Setting.....	5-12	Vamp	6-17
Qualifier	5-13	Variable.....	6-11
Record Constant On.....	11-4	Vavg	6-17
Rectangle	6-6	Vbase	6-17
Reference Clock.....	1-14	Vectors	12-2
Rise Time	6-15	Vertical Expansion	2-9
Roll	3-8	Vertical Scale	2-8
RS232 Decoding.....	8-1, 8-5	Vertical Window	5-14, 5-18
Runt Trigger.....	5-13	Video Polarity	5-20
Sample Rate	4-6	Video Standard	5-20
Set the Gate	14-4	Video Trigger	5-20
Setups.....	13-3	VISA	14-5
Single.....	5-4	Vmax	6-17
Slope Condition	5-17	Vmin.....	6-17
Slope Trigger.....	5-17	Vpp	6-17
SPI Decoding	8-1, 8-13	Vrms.....	6-17
Static IP	14-3	Vtop	6-17
Subnet Mask	14-4	Waveform Analysis	11-7
Substraction.....	6-3	Waveform Confusion	4-7
Sync.....	5-20	Waveform Distortion.....	4-6
The Center of the Screen	3-10	Waveform Invert.....	2-7
Threshold	6-21	Waveform Leakage.....	4-7
Time Setting	5-17	Waveform Playback	11-5
Traces	13-3	Waveform Record.....	11-1
Track Mode	6-31	Waveforms	13-3
Trigger Coupling	5-5	Window Function	6-5
Trigger Holdoff	5-6	XOR.....	6-8
Trigger Level	5-9	X-Y	3-5
Trigger Mode.....	5-3, 5-9	Y-T	3-4