

User Manual

SDS2000 Series Digital oscilloscope

UM01020-E03A



2015 SIGLENT TECHNOLOGIES CO., LTD.

Copyright and Declaration

Copyright

SIGLENT TECHNOLOGIES CO., LTD. All Rights Reserved.

Trademark Information

SIGLENT is the registered trademark of **SIGLENT TECHNOLOGIES CO., LTD.**

Declaration

SIGLENT products are protected by patent law in and outside of P.R.C.

SIGLENT reserves the right to modify or change parts of or all the specifications or pricing policies at company's sole decision.

Information in this publication replaces all previously corresponding material.

Any way of copying, extracting or translating the contents of this manual is not allowed without the permission of **SIGLENT**.

Product Certification

SIGLENT guarantees this product conforms to the national and industrial stands in China and other international stands conformance certification is in progress.

Contact Us

If you have any problem or requirement when using our products, please contact

SIGLENT TECHNOLOGIES CO., LTD

Add: 3//F, Bldg No.4, Antongda Industrial Zone, 3rd Liuxian Road, Bao'an District, Shenzhen, 518101, P.R.China

Tel: 400-878-0807

E-mail: sales@siglent.com

<http://www.siglent.com>

Safety Information

General Safety Summary

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

Use Proper Power Line

Only the power cord designed for the instrument and authorized by local country could be used.

Ground the Instrument

The instrument is grounded through the protective earth conductor of the power line. To avoid electric shock, please make sure the instrument is grounded correctly before connecting its input or output terminals.

Connect the Signal Wire Correctly

The potential of the signal wire is equal to the earth, so do not connect the signal wire to a high voltage.

Look Over All Terminals' Ratings

To avoid fire or electric shock, please look over all ratings and sign instruction of the instrument. Before connecting the instrument, please read the manual carefully to gain more information about the ratings.

Use Proper Overvoltage Protection

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

Electrostatic Prevention

Operate in an electrostatic discharge protective area environment to avoid damages induced by static discharge. Always ground both the internal and external conductors of the cable to release static before connecting.

Keep Well Ventilation

Inadequate ventilation may cause increasing of temperature, which will eventually damage the instrument. So keep well ventilation and inspect the intake and fan regularly.

Avoid Circuit or Components Exposed

Do not touch exposed contacts or components when the power is on.

Use proper Fuse

Use only the specified fuse.

Do Not Operate Without Covers

Do not operate the instrument with covers or panels removed.

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 **SIGLENT** authorized personnel.

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 injuries, 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 device clean and dry.

Handling Safety

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

Safety Terms and Symbols

Terms in this Manual. These terms may appear in this manual:



WARNING

Warning statements indicate the conditions or practices that could result in injury or loss of life.



CAUTION

Caution statements indicate the 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 direct injuries or hazards that may happen.

WARNING Indicates potential injuries or hazards that may happen.

CAUTION Indicates potential damages to the instrument or other property that may happen.

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



**Hazardous
Voltage**



**protective Earth
Terminal**



Warning



**Test
Ground**



Power Switch

Measurement Category

Measurement Categories

SDS2000 series digital oscilloscopes can make measurements in measurement category I.



WARNING

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

Measurement Category Definitions

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

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

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

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

Working Environment

Temperature

Operating: 10°C to +40°C

Non-operation: -20°C to +70°C

Humidity

Under +35°C: ≤90% relative humidity

+35°C to +40°C: ≤60% relative humidity



WARNING

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

Altitude

Operating: less than 3 Km

Non-operation: less than 15 Km

Installation (overvoltage) Category

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



WARNING

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

Installation (overvoltage) Category Definitions

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

Installation (overvoltage) category II refers to the local power distribution level which is applicable to equipment connected to the AC line (AC power).

Ventilation Requirement

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



WARNING

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

General Care and Cleaning

Care

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



WARNING

To avoid damages to the instrument or probe, please do not leave them in fog, liquid, or solvent.

Cleaning

Please perform the following steps to clean the instrument and probe regularly according to its operating conditions.

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



WARNING

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



WARNING

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

Document Overview

This manual introduces how to use SDS2000 serial digital oscilloscope in details.

Quick Start	Provide information about preparations before using the instrument and a brief introduction of the instrument.
To Set the Vertical System	Introduce the functions of the vertical system of the oscilloscope.
To Set the Horizontal System	Introduce the functions of the horizontal system of the oscilloscope.
To Set the Sample System	Introduce the functions of the sample system of the oscilloscope.
To Trigger the Oscilloscope	Introduce the trigger mode, trigger coupling, trigger holdoff, external trigger and various trigger types of the oscilloscope.
Serial Trigger and Decode	Introduce how to trigger and decode the input signal.
To Save Reference Waveform	Introduce how to save and display REF waveform.
To Make Math Operation	Introduce the math operation function of the oscilloscope.
To Make Cursor Measurements	Introduce how to use cursors to make measurements.
To Make Measurements	Introduce how to use measure function to measure the waveform parameters.
Display Setting	Introduce how to set the display of the oscilloscope.
Save and Recall	Introduce how to save and recall the measurement result and the setting of the oscilloscope.
System Setting	Introduce how to set the system setup.
Arbitrary Waveform Generation	Introduce how to use Arbitrary waveform generation function.
Default	Introduce the Default setup of the oscilloscope
Troubleshooting	Introduce how to deal with common failures of the oscilloscope.

Content Conventions in this Manual:

This manual takes SDS2304 for example and the descriptions here have contained all the functions and performances of other models. SDS2000 series includes the following models:

Model	Analogy Bandwidth	channels
SDS2072/SDS2074	70MHz	2/4
SDS2102/SDS2104	100 MHz	2/4
SDS2202/SDS2204	200 MHz	2/4
SDS2302/SDS2304	300 MHz	2/4

Table of Content

Copyright and Declaration	I
Safety Information.....	II
General Safety Summary	II
Safety Terms and Symbols	IV
Measurement Category	V
Working Environment	VI
Ventilation Requirement	VII
General Care and Cleaning.....	VIII
Document Overview	IX
Quick Start	1
General Inspection	2
Appearance and Dimensions	3
To Prepare the Oscilloscope for Use	4
To Adjust the Supporting Legs.....	4
To Connect to Power Supply	5
Power-on Inspection	6
To Connect the Probe	7
Function Inspection	8
Probe Compensation	9
Front Panel Overview.....	10
Rear Panel Overview	11
Front Panel Function Overview.....	12
Horizontal	12
Vertical.....	13
Trigger	15
Run Control	16
Universal Knob	17
Menu.....	18
Help	19
User Interface	20
To Use the Security Lock	22
To Set the Vertical System	23
To Enable the Channel.....	24
To Adjust the Vertical Scale	25
To Adjust the Vertical Position	25
To Specify Channel Coupling.....	26
To Specify Bandwidth Limit.....	26
To Specify Probe Attenuation Factor	27
To Specify channel Input Impedance.....	27
To Specify Amplitude Unit	28
To Invert a Waveform	28
To Set the Horizontal System.....	29

To Adjust the Horizontal Scale	30
To Adjust the Horizontal Trigger Position.....	31
Panning and Zooming Single or Stopped Acquisitions.....	32
To Change the Horizontal Format.....	33
To Use the Zoom Function.....	35
To Set the Sample System	36
Run Control.....	37
Overview of Sampling.....	38
Sampling Theory	38
Sample Rate.....	39
Oscilloscope Bandwidth and Sample Rate.....	40
To Specify Memory Depth.....	41
To Select Sampling Mode	42
To Specify Waveform Interpolation Method	43
To Select Acquisition Mode.....	45
Normal.....	46
Peak Detect.....	47
Average.....	48
High Resolution.....	49
To Use Sequence Mode.....	50
To Trigger the Oscilloscope	53
Trigger Source	55
Trigger Mode	56
Trigger Level.....	57
Trigger Coupling	58
Trigger Holdoff.....	59
Noise Rejection	60
Trigger Type	62
Edge Trigger.....	63
Slope Trigger.....	64
Pulse Trigger.....	66
Video Trigger.....	68
Window Trigger	71
Interval Trigger	74
DropOut Trigger	76
Runt Trigger	78
Pattern Trigger	79
Serial trigger and decode.....	81
IIC Trigger and Serial Decode.....	82
Setup for IIC Signals	82
IIC Triggering.....	83
IIC Serial Decode	86
SPI Triggering and Serial Decode.....	88
Setup for SPI Signals.....	88

SPI Triggering.....	91
SPI Serial Decode	93
UART/RS232 Triggering and Serial Decode.....	95
Setup for UART/RS232 Signals.....	95
UART/RS232 Triggering	96
UART/RS232 Serial Decode.....	97
CAN Trigger and Serial Decode	99
Setup for CAN Signals	99
CAN Triggering.....	100
CAN Serial Decode	102
LIN Triggering and Serial Decode.....	104
Setup for LIN Signals	104
LIN Triggering.....	105
LIN Serial Decode	106
Digital Channels.....	108
To Connect the Digital Probes to the Device under Test.....	109
Acquiring Waveform Using the Digital Channels	110
To Change the Display Type of the Digital Channels	110
To Switch a Single Channel On or Off	111
To Switch All Digital Channels On or Off	111
To Change the Logic Threshold for Digital Channels	112
To Reposition a Digital Channel.....	112
To Display Digital Channels as a Bus	113
To Save Reference Waveform.....	114
To Save REF Waveform to Internal Memory.....	115
To Display REF Waveform	115
To Adjust REF Waveform Display	116
To Clear REF Waveform Display.....	117
To Make Math Operation	118
Units for Math Waveforms.....	119
Math Operators.....	120
Addition or Subtraction.....	120
Multiplication and Division.....	121
FFT Operation	122
Math Function Operation	125
Differentiate	125
Integrate	126
Square Root	128
To Adjust the Math Waveform Scale and Offset.....	129
To Adjust the Math Waveform Horizontal Scale and Offset.....	129
To Adjust the Math Waveform Vertical Scale and Offset	129
To Make Cursors Measurements.....	130
X Cursors.....	130
Y Cursors.....	130

To Make Cursor Measurements.....	131
To Make Measurements	133
Type of Measurement.....	134
Voltage Measurements.....	134
Time Measurements.....	136
Delay Measurements	137
To Make Automatic Measurement.....	138
To Clear Measurement Parameters	140
To Make Statistics Measurements.....	141
To Make All Parameters Measurement	142
Display Setting	143
To Set Display Type.....	144
To Set Color Display.....	145
To Set and Clear Persistence.....	146
To clear the display.....	147
To Select Grid Type.....	147
To Adjust Waveform Intensity	148
To Adjust Grid Brightness	150
To Adjust Transparence	150
Save and Recall.....	151
Save Type.....	152
Internal Save and Recall	153
External save and recall	154
Disk Management.....	157
To Create a New File or Folder	158
To delete a file or folder.....	159
To rename a file or a folder	159
System Function Setting.....	160
To View the System Status	161
To Do Self Calibration.....	162
To Use the History Function	163
To Enable or Disable the Sound	165
To Specify the Language	165
To Do Pass/Fail Test.....	166
To Set and Perform Pass/Fail Test.....	167
To Save and Recall Test Mask.....	168
IO Set.....	170
To Set the USB Device	170
To Set the LAN	171
To Set Aux Output.....	172
To Do Power Analysis	173
Perform Channel Deskew	174
Perform Power Analysis.....	177
To Print the Screen Image	202

To Enable Quick-Cal.....	203
To Update Firmware and Configuration	204
Do Self Test	205
Screen Test.....	205
Keyboard Test	206
LED Test.....	207
To specify Screen Saver Time.....	208
Option Management.....	209
To Set and Display Data /Time.....	211
Arbitrary Waveform Generator	212
To Set Wave Type and Parameters.....	213
To Output Arbitrary Waveform	215
To Set Output Load.....	216
Recover the Default Values of Wave Gen.....	216
AWG Self Cal.....	217
Default Setup	218
Troubleshooting	224

Content of Figure

Figure 1 Front View	3
Figure 2 Top View	3
Figure 3 Adjust the Supporting Legs.....	4
Figure 4 To Connect to Power Supply.....	5
Figure 5 Function Inspection	8
Figure 6 Front Panel Overview.....	10
Figure 7 Rear Panel Overview	11
Figure 8 Help Message	19
Figure 9 User Interface.....	20
Figure 10 To Use the Security Lock	22
Figure 11 Display Type Set to Dots	43
Figure 12 x Interpolation.....	44
Figure 13 Sinx Interpolation	44
Figure 14 Acquisition System	45
Figure 15 Sine With Glitch, Normal Mode.....	47
Figure 16 Sine With Glitch, Peak Detect Mode.....	47
Figure 17 Sine With Random Noise, Normal Mode.....	48
Figure 18 Sine With Random Noise, Average Mode	49
Figure 19 SEQUENCE Function Menu	50
Figure 20 Waterfall Display Mode	50
Figure 21 Stop under Sequence Mode	51
Figure 22 HISTORY Function Menu	52
Figure 23 Turn off the Noise Reject	60

Figure 24 Turn on the Noise Reject	61
Figure 25 Edge Trigger.....	63
Figure 26 Slope Trigger	65
Figure 27 Pulse Trigger.....	67
Figure 28 Video Trigger.....	70
Figure 29 Absolute Window Trigger	72
Figure 30 Relative Window Trigger	73
Figure 31 Interval Trigger	75
Figure 32 Edge DropOut Trigger.....	77
Figure 33 State DropOut Trigger	77
Figure 34 Runt Trigger	78
Figure 35 Pattern Trigger	80
Figure 36 IIC SINGAL Menu.....	82
Figure 37 IIC TRIGGER Menu	83
Figure 38 IIC Decode Menu	86
Figure 39 IIC Decode Result.....	86
Figure 40 SPI SINGAL Menu	88
Figure 41 SPI Clock Menu	88
Figure 42 MOSI Menu	88
Figure 43 MOSI Menu	89
Figure 44 SPI CS Menu	89
Figure 45 SPI TRIG SET Menu.....	91
Figure 46 SPI Trigger	92
Figure 47 SPI Decode function menu	93
Figure 48 SPI Decode	93
Figure 49 UART/RS232 SIGNAL Menu	95
Figure 50 BUS CONFIG Menu.....	95
Figure 51 UART TRIG SET Menu.....	96
Figure 52 UART/RS232.....	96
Figure 53 UART/RE232 DECODE Function Menu.....	97
Figure 54 UART/RE232 Decode.....	97
Figure 55 CAN SIGNAL Menu	99
Figure 56 CAN Trigger	101
Figure 57 CAN Decode	102
Figure 58 LIN SIGNAL Menu	104
Figure 59 LIN Trigger.....	105
Figure 60 LIN DECODE Function Menu	106
Figure 61 LIN Decode	106
Figure 62 Connect the Digital Probes	109
Figure 63 Digital Function Menu	110
Figure 64 Low Display Type	110
Figure 65 High Display Type	111
Figure 66 THRESHOLDS Function Menu.....	112
Figure 67 DIGITALBUS Function Menu	113

Figure 68 Digital Bus	113
Figure 69 Reference Waveform	116
Figure 70 FFT Waveform In Split Mode	124
Figure 71 Differential Function Operation	125
Figure 72 Integral without Offset	126
Figure 73 Integral with Offset	127
Figure 74 Square Root	128
Figure 75 Measure Pulse Width	131
Figure 76 Check the Change of the Pulse Width	132
Figure 77 Voltage Measurements	134
Figure 78 Overshoot	135
Figure 79 Preshoot	135
Figure 80 Time Measurements	136
Figure 81 Before Adding the Duty Measurement	138
Figure 82 After Adding Duty Measurement	139
Figure 83 Statistics Measurement	141
Figure 84 All Parameters Measurement	142
Figure 85 Vectors Display	144
Figure 86 Dots Display	144
Figure 87 Color Temperature	145
Figure 88 Persist Set to Infinite	146
Figure 89 Intensity Set to 100%	148
Figure 90 Intensity Set to 20%	149
Figure 91 SAVE/RECALL File System	154
Figure 92 Select Save Location	155
Figure 93 File Name Dialogue	155
Figure 94 Input Keyboard	158
Figure 95 System Status	161
Figure 96 Do Self Cal	162
Figure 97 History	163
Figure 98 Pass/Fail Test	166
Figure 99 LAN Setting Interface	171
Figure 100 DF2001A Deskew Fixture	175
Figure 101 Second Page of the UTILITY Function Menu	175
Figure 102 UTILITY Function Menu	175
Figure 103 Power Analysis SIGNAL Menu	175
Figure 104 DESKEW Menu	176
Figure 105 Deskew Success	176
Figure 106 ANALYSIS Function Menu	178
Figure 107 Power Quality SIGNAL Menu	178
Figure 108 Perform Power Quality Analysis	179
Figure 109 STATISTIC Menu	179
Figure 110 Turn on Statistic Function	180
Figure 111 ANALYSIS Function Menu	181

Figure 112 Harmonics SINGAL Menu	181
Figure 113 Harmonics SETTING Menu	182
Figure 114 Perform Harmonics Analysis	182
Figure 115 ANALYSIS Function Menu	184
Figure 116 Inrush Analysis SIGNAL Menu	184
Figure 117 Perform Inrush Analysis	185
Figure 118 ANALYSIS Function Menu	186
Figure 119 Switching Analysis Signal Menu.....	186
Figure 120 Switching Analysis SETTINGS Menu	187
Figure 121 Perform Switching Analysis.....	188
Figure 122 Slew Rate ANALYSIS Function Menu	189
Figure 123 Slew Rate Signal Menu.....	189
Figure 124 Perform Slew Rate Analysis.....	190
Figure 125 Modulation ANALYSIS Function Menu	191
Figure 126 Modulation SIGNAL Menu	191
Figure 127 Perform Modulation Analysis	192
Figure 128 Output Ripple ANALYSIS Function Menu.....	193
Figure 129 Output Ripple SIGNAL Menu	193
Figure 130 Perform Output Ripple Analysis.....	194
Figure 131 Output Ripple STATISTIC Menu	194
Figure 132 Turn on Statistic Function.....	195
Figure 133 Turn On/Off ANALYSIS Function Menu	196
Figure 134 Turn On/Off SIGNAL Menu	196
Figure 135 Perform Turn On/Off Analysis	197
Figure 136 Transient ANALYSIS Function Menu	198
Figure 137 Transient SIGNAL Menu	198
Figure 138 Transient SETTING Menu.....	199
Figure 139 Perform Transient Analysis	199
Figure 140 Efficiency ANALYSIS Function Menu	200
Figure 141 Efficiency SIGNAL Menu.....	200
Figure 142 Perform Efficiency Analysis.....	201
Figure 143 Screen Test	205
Figure 144 Keyboard Test	206
Figure 145 LED Test.....	207
Figure 146 Screen Saver Interface	208
Figure 147 OPTION Function Menu	209
Figure 148 LABEL Function Menu	209
Figure 149 Option Information.....	210
Figure 150 DATE/TIME Function Menu	211
Figure 151 Send Wave Dialogue	215
Figure 152 Save Waveform Successfully	215
Figure 153 SETTING Function Menu.....	216
Figure 154 AWG Self Cal	217

Quick Start

This chapter introduces the preparations when using the oscilloscope for the first time, the front panel, rear panel and user interface of the oscilloscope,

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
- ◆ Back Panel Function Overview
- ◆ User Interface
- ◆ To Use the Security Lock

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. **SIGLENT** 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 **SIGLENT** 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 **SIGLENT** sales representative.

Appearance and Dimensions

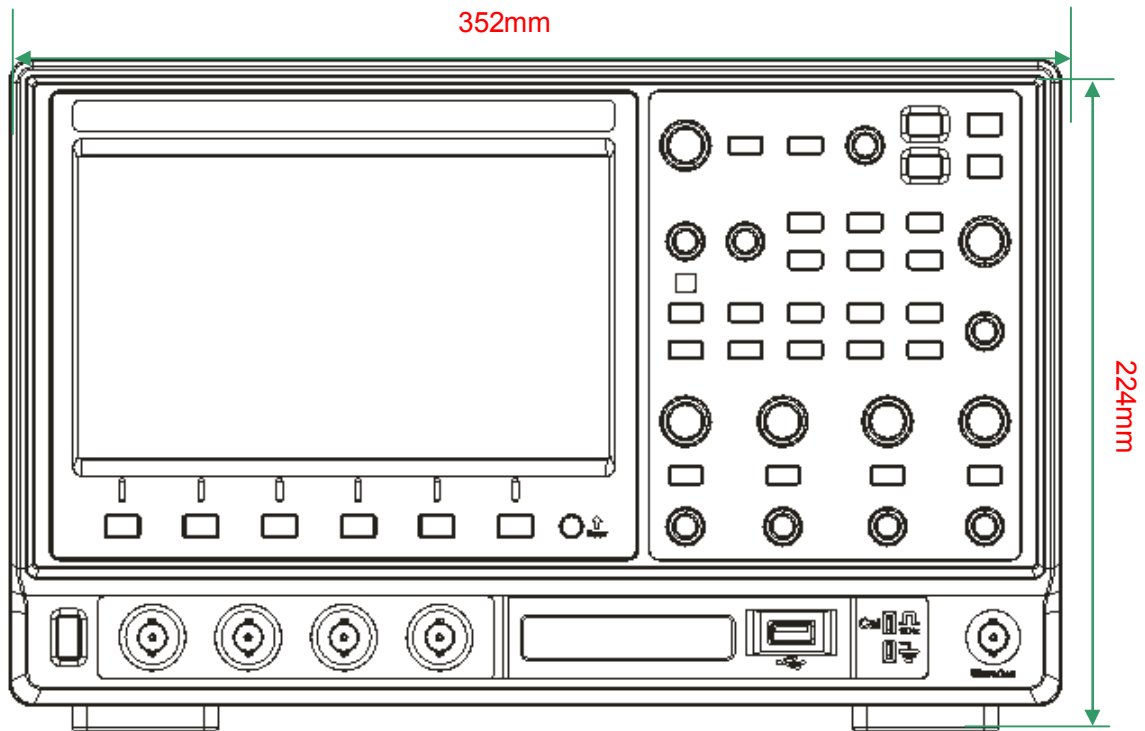


Figure 1 Front View

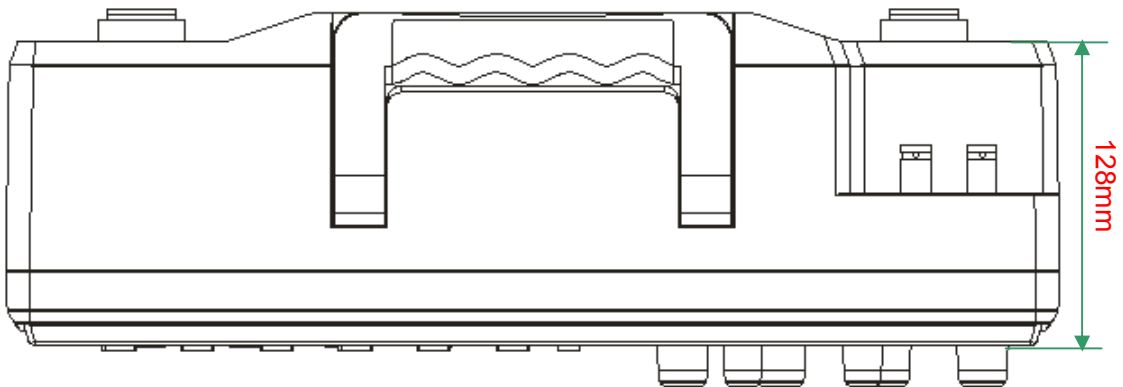


Figure 2 Top View

To Prepare the Oscilloscope for Use

To Adjust the Supporting Legs

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

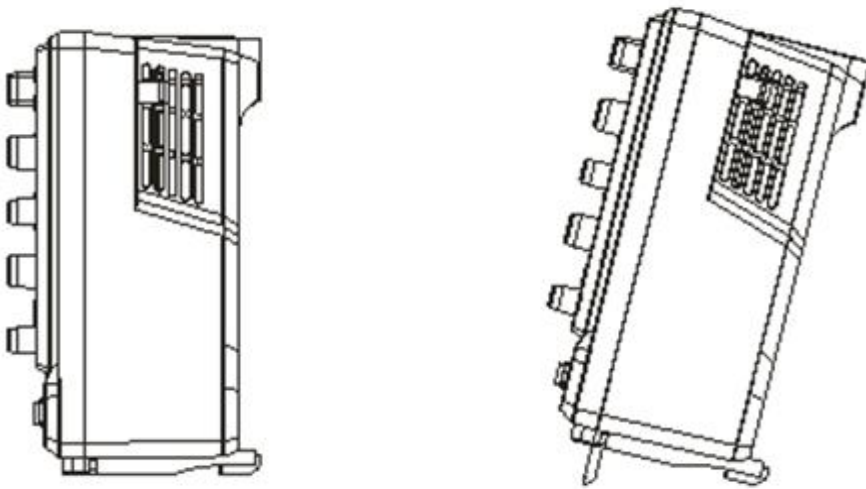


Figure 3 Adjust the Supporting Legs

To Connect to Power Supply

The power requirements of SDS2000 are 100-240 V, 45-440 Hz. Please use the power cord supplied with the accessories to connect the oscilloscope to the power source.

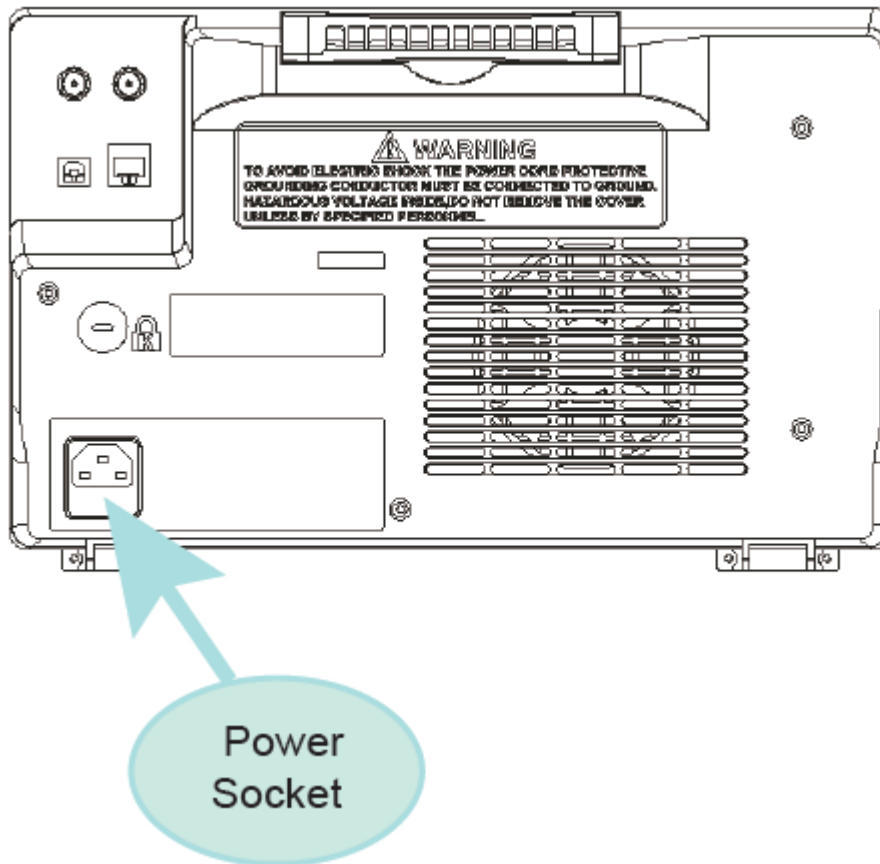


Figure 4 To Connect to Power Supply

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 sound of relay switching. After the self-test is finished, the welcome screen is displayed.

To Connect the Probe

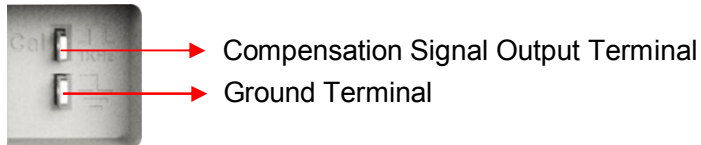
SIGLENT provides passive probes for the SDS2000 series oscilloscopes. For detailed technical information of the probes, please refer to the corresponding Probe User's Guide.

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.

Function Inspection

1. Press the **Default** button on the front panel to restore the instrument 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.
4. Press the **Auto Setup** softkey.
5. Observe the waveform on the display. In normal condition, the display should be a square waveform as shown in the figure below:

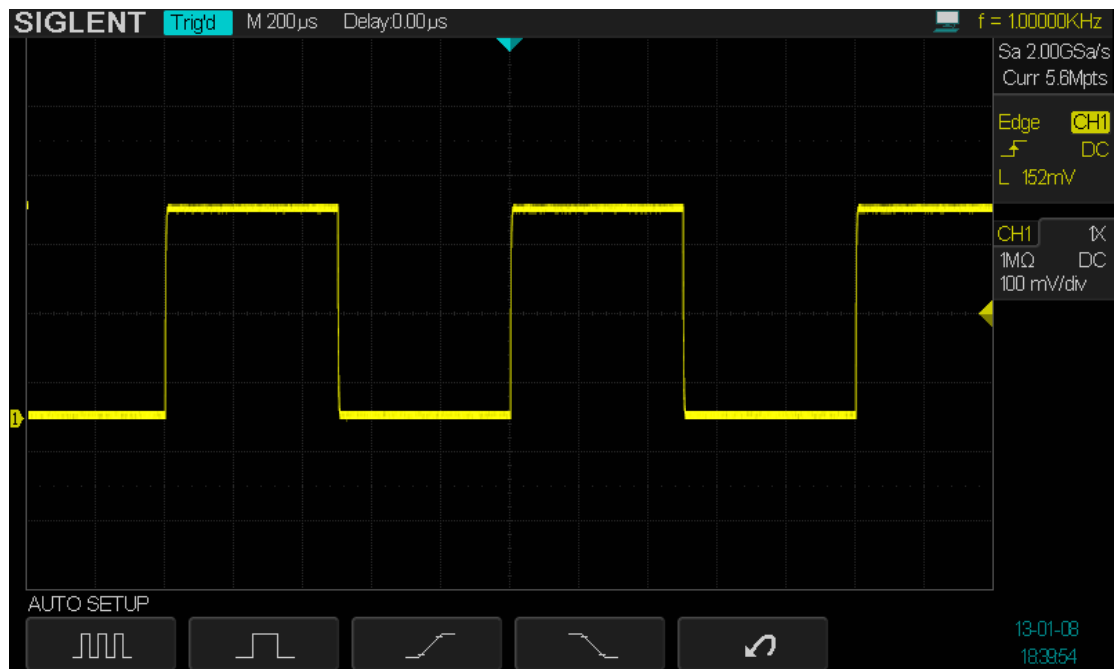


Figure 5 Function Inspection

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



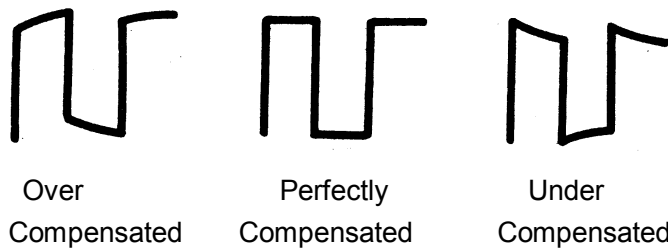
WARNING

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

Probe Compensation

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

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



4. Use a nonmetallic driver to adjust the low-frequency compensation adjustment hole on the probe until the waveform displayed is as the “Perfectly compensated” in the figure above.

Front Panel Overview

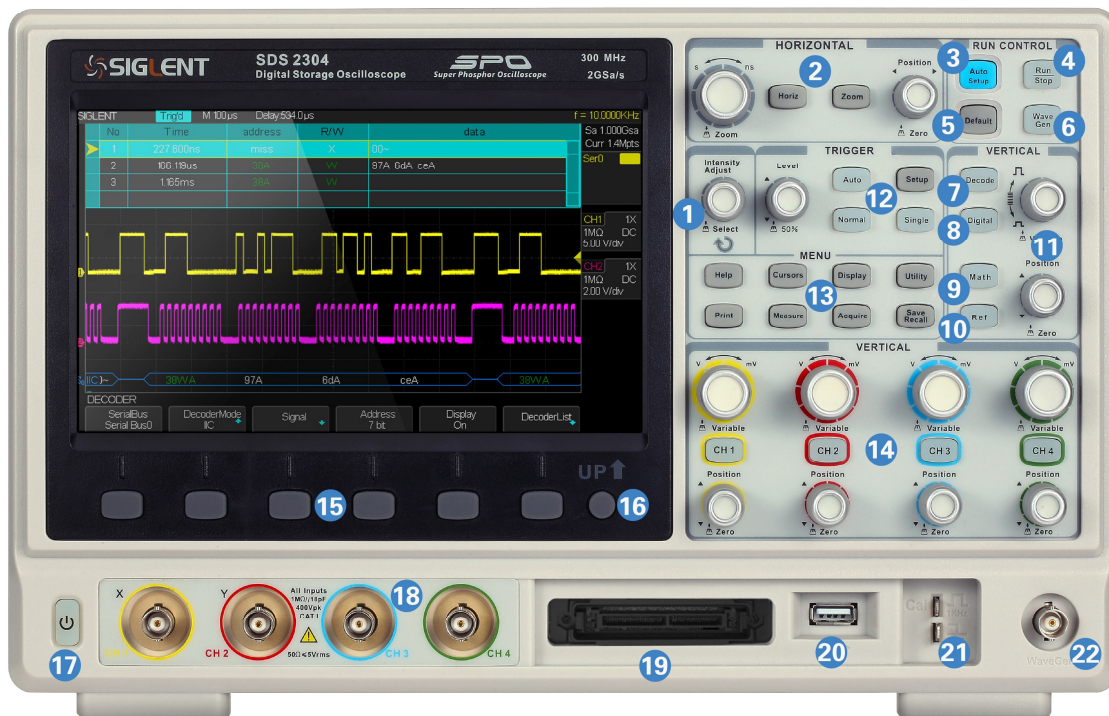


Figure 6 Front Panel Overview

NO.	Description	NO.	Description
1	Universal Knob	12	Trigger Control
2	Horizontal Control	13	Common Function Menus
3	Auto Setup	14	Vertical Control
4	Run/Stop	15	Menu Softkey
5	Default Setup	16	Up Botton
6	Wave Gen	17	Power Botton
7	Decode Control	18	Analog Channel Input
8	Digital Channel Control	19	Digital Channel Input
9	Math Function	20	USB Host
10	Ref waveform	21	Probe Comp
11	Vertical control for (Decode/Digital/Math/Ref)	22	WaveGen Output Terminal

Rear Panel Overview

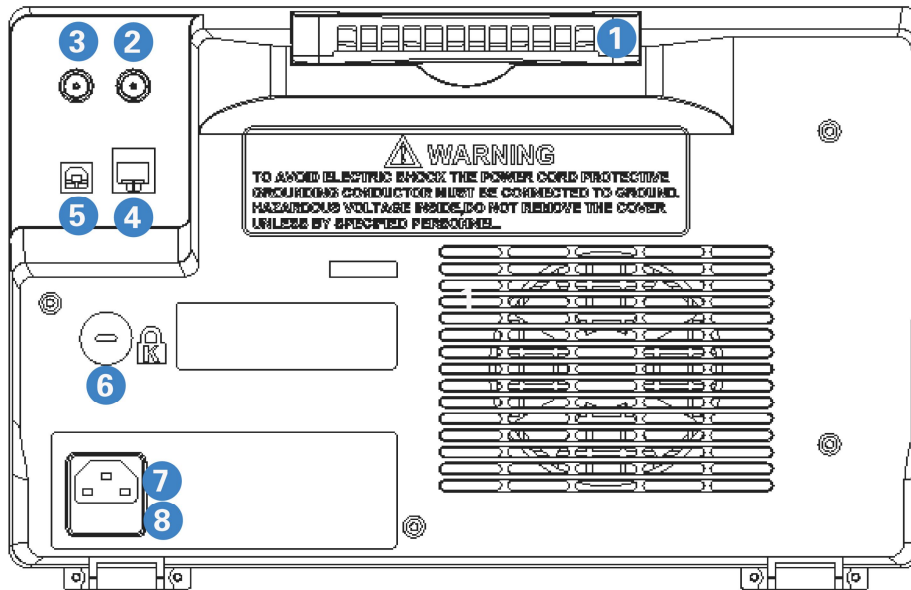



Figure 7 Rear Panel Overview


1. **Handle**
Pull up the handle vertically for easy carrying of the instrument. When you do not need the handle, press it down.
2. **EXT TRIG**
External trigger signal input terminal.
3. **Pass/Fail or Trigger Out**
The oscilloscope outputs a signal that can reflect the current capture rate of the oscilloscope at each trigger via this interface or outputs the pass/fail test pulse.
4. **LAN**
Connect the instrument to the network via this interface for remote control.
5. **USB Device**
PictBridge printer or PC can be connected via this interface to print waveform data or control the instrument using PC software.
6. **Lock Hole**
You can lock the instrument to a fixed location using the security lock (please buy it yourself) via the lock hole.
7. **AC Power Socket**
AC power input terminal. The power requirements of this oscilloscope are 100-240 V, 45-440 Hz. Use the power cord provided with the accessories to connect the instrument to AC power. Then, you can press the power key at the front panel to start the instrument.
8. **Fuse**
If a new fuse is required, please use the specified fuse (250V, T1.25A).


Front Panel Function Overview


Horizontal



 : press the button to enter the HORIZONTAL function menu. Available display formats have YT, XY and ROLL.

 : press the button to turn on or off the zoom function.

Horizontal Position Knob  : adjust horizontal 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 at the upper-right corner of the screen would change accordingly. Press down this knob to quickly reset the trigger position.

Horizontal Scale Knob  : adjust the horizontal time base. Turn clockwise to reduce the time base and turn counterclockwise to increase the time base. During the modification, waveforms of all the channels will be displayed in expanded or compressed mode and the time base message at the upper-left side of the screen would change accordingly. Press down this knob to quickly turn on zoom function.

Vertical



Decode

: press the button to enter the DECODE function menu. SDS2000 supports IIC, SPI, UART/RS232, CAN and LIN serial bus decode.

Digital

: press the button to enter the DIGITAL function menu. SDS2000 supports 8 integrated digital channels.

Math

: press the button to enter the MATH function menu. SDS2000 provides addition, subtraction, multiplication, FFT, differential, integral and square root operations.

Ref

: press the button to enter the REF function menu. A reference waveform can be displayed and compared against other waveforms.

Ref/Math Waveform Vertical Position Knob

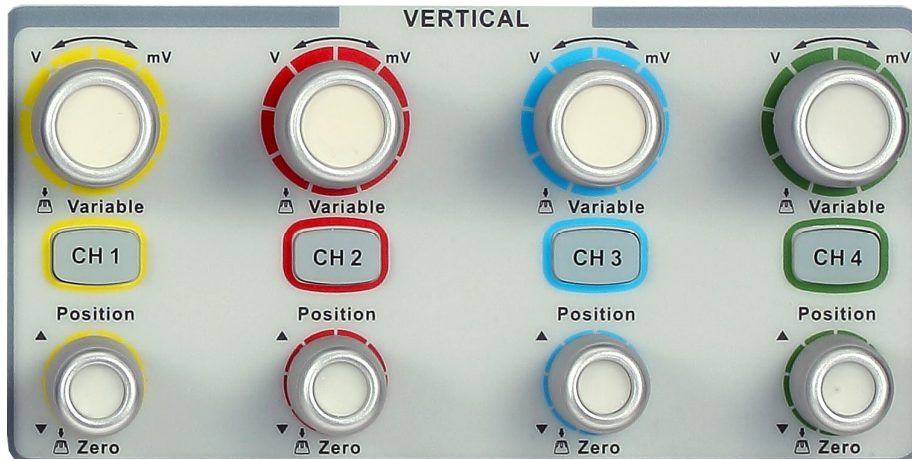



: adjust the vertical position of the Ref/Math waveform. Turn clockwise to increase the position and turn counterclockwise to decrease. During the modification, the waveform would move up and down and the position message at the center of the screen would change accordingly. Press down this knob to quickly reset the vertical position to zero.


Ref/Math Waveform Vertical Variable Knob




: adjust the vertical scale of the Ref/Math waveform. Turn clockwise to decrease the scale and turn counterclockwise to increase. During the modification, the amplitude of the waveform would enlarge or reduce and the scale information at the center of the screen would change accordingly.




 : Analog input channels. The 4 channels are marked by different colors which are also used to mark both 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.


Vertical Position Knob  : adjust the vertical position of the current channel waveform. Turn clockwise to increase the position and turn counterclockwise to decrease. During the modification, the waveform would move up and down and the position message at the lower-left corner of the screen would change accordingly. Press down this knob to quickly reset the vertical position to zero.


Vertical Variable Knob  : adjust the vertical scale of the current channel. Turn clockwise to decrease the scale and turn counterclockwise to increase. During the modification, the amplitude of the waveform would enlarge or reduce and the scale information at the right side of the screen would change accordingly. Press down this knob to quickly switch the vertical scale adjustment modes between “Coarse” and “Fine”.


Trigger



 : press the button to enter the TRIGGER function menu. SDS2000 provides abundant advanced trigger functions

 : press the button to set the trigger mode to Auto.

 : press the button to set the trigger mode to Normal.

 : press the button to set the trigger mode to Single.


Trigger Level Knob





: adjust the trigger level. Turn clockwise to increase the level and turn counterclockwise to reduce the level. During the modification, the trigger level line would move up and down and the value in the trigger level message box at the up-right corner of the screen would change accordingly. Press down the knob to quickly reset the trigger level to center of the waveform.


Run Control



 : 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.

 : press the button to set the acquisition state to Run or Stop.
In RUN state, the key is illuminated in yellow.
In STOP state, the key is illuminated in red.

 : press the button to reset the oscilloscope to default setting.

 : press the button to enable the Wave Gen function and open the WAVEFORM function menu.

Universal Knob



1. Adjust the waveform intensity.

When there is no menu operation, turn the knob to adjust the intensity of the displayed waveforms. The adjustable range is from 0% to 100%. Turn clockwise to increase the intensity and counterclockwise to reduce. You can also press the **Display** button; press the **Next Page** softkey to go to the second page of the DISPLAY function menu; press the **Intensity** softkey and then turn the **Universal Knob** to adjust the waveform intensity.

2. Select the desired submenu.

In menu operation, press any menu softkey and turn the **Universal Knob** to select the desired submenu under the menu and push down the knob to confirm the current submenu. Turn clockwise to up the intensity and counterclockwise to down.

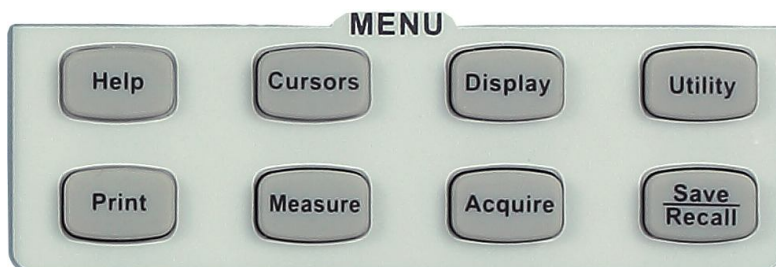
3. Modify parameters.


After having chosen a parameter, turn the **Universal Knob** to modify the value. Turn clockwise to increase the intensity and counterclockwise to reduce.


4. Choose file or directory or input filename.

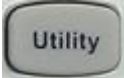
After having entered the file system, turn the **Universal Knob** to select the desired file or directory. When inputting filename, turn the **Universal Knob** to select the desired character and the push the knob to confirm.


Menu

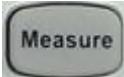



 : press the button to enter the CURSOR function menu. SDS2000 provides X1, X2, X1-X2, Y1, Y2, Y1-Y2 cursor types.


 : press the button to enter the DISPLAY function menu to set the display type, color, persistence time, grid type and so on.

 : Press the button to enter the UTILITY function menu to look at the system status, do self calibration, set the sound, language and so on.

 : press the button to print the current display interface or save the current display interface to USB flash driver.

 : press the button to enter the MEASURE function menu to set the measurement parameters, all measurement and statistics.

 : press the button to enter the ACQUIRE function menu to set the acquisition mode, memory depth, sinx/x interpolation and so on.

 : press the button to enter the SAVE/RECALL function menu to save setups, waveforms, pictures, or CSV files to internal memory or USB flash driver.

Help

SDS2000 has an on line help function that supplies multi-language help information, and you can recall then to help you operate the oscilloscope when you need.

Press the **Help** button on the front panel to enter the on line help status, then press the button or the softkey to obtain the corresponding help information.

If you want to recall the help information of the **Single** button or the **Run/Stop** button, you need to press the **Single** or the **Run/Stop** button as soon as you enter the on line help status. Because that press the **Single** button can go to the next page of the help information while press the **Run/Stop** button will return to the first page.

All submenus of every main menu have their help information. If you want to see the help information of the second page, first you need to press the **Help** button to exit the on line help status; and then press the appointed softkey to enter the second page of the menu; press the **Help** button to enter the on line help status again; press the desired softkey to see its help information.

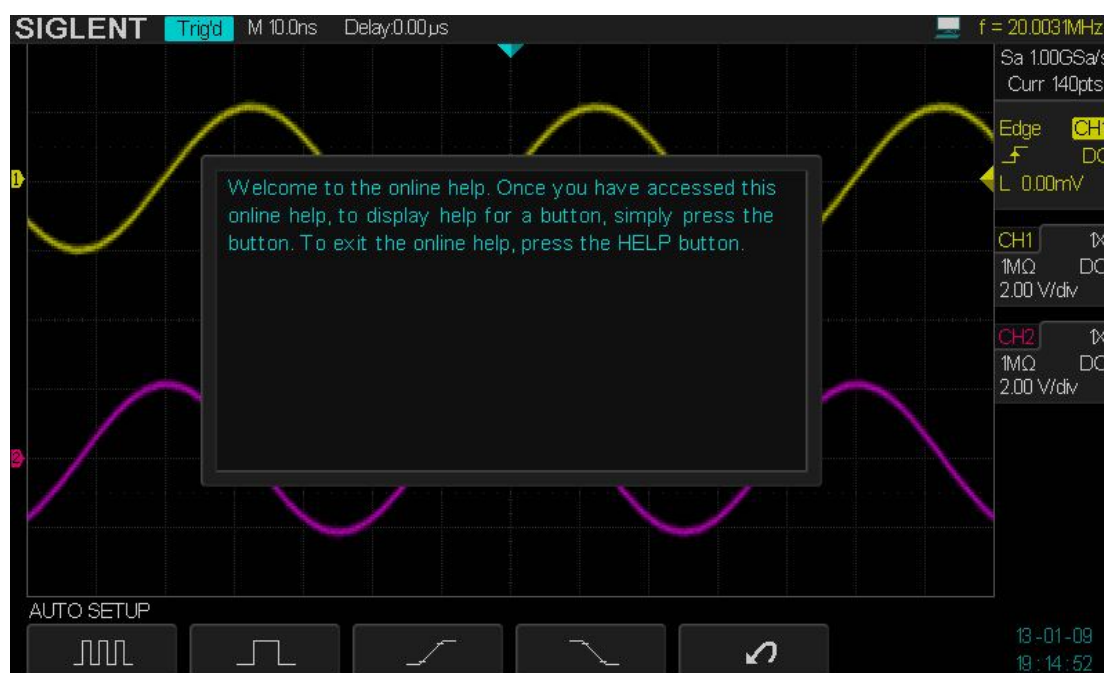


Figure 8 Help Message

User Interface

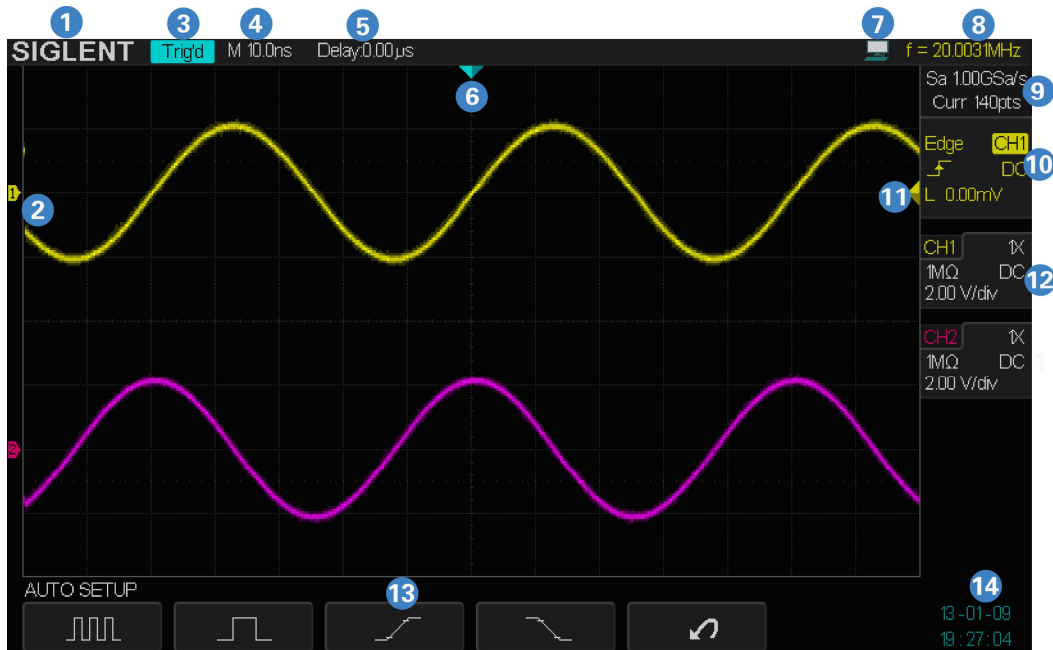


Figure 9 User Interface

1. **Product Logo**
SIGLENT is the registered trademark of **SIGLENT TECHNOLOGIES CO., LTD.**
2. **Channel Label/Waveform**
 Different channels are marked by different colors and the color of the waveform complies with the color of the channel.
3. **Trigger Status**
 Available trigger status includes Ready, Auto, Stop, Arm, Trig'd.
4. **Horizontal Time Base**
 - Represent the time per grid on the horizontal axis on the screen.
 - Use the **HORIZONTAL SCALE Knob** to adjust the parameter. The available range is from 1.000 ns to 50 s.
5. **Trigger Position**
 Turn the Horizontal Position Knob to adjust the parameter. Push the knob to set the value to 0 automatically.
6. **Trigger Position Label**
 Indicate the trigger position on the waveform.

7. USB Device

Indicate the USB device setup.

 Indicate that the **USB Device** is set to **USBTMC**.

 Indicate that the **USB Device** is set to **Printer**.


8. Frequency Counter


Display the frequency value of the trigger channel.

9. Sampling Rate/ Memory Depth


Display the current sampling rate and memory depth. Sa means the current sampling rate and Curr means the current memory depth. Turn the **Universal Knob** to adjust the parameter


10. Trigger Setup

Trigger Type  : display the current trigger type. The trigger type names display by the abbreviation when the name is too long to display.

Trigger source  : display the currently trigger source. Different channels display in different color.

Trigger condition  : display the current trigger condition.


Trigger coupling  : display the current trigger coupling. Available trigger coupling mode: DC, AC, HF Reject, LF Reject.


Trigger level  : display the current value of trigger level. Push the knob to set the trigger to the 50% of the waveform amplitude automatically.


11. Trigger Level Label


Display the position of trigger level, the color is the same to the trigger channel. It can move from +4.5div to -4.5div of the screen center.

12. Channel Setup

Probe attenuation factor  : display the current probe attenuation factor of the channel. Available probe attenuation factors: 1X, 5X, 10X, 50X, 100X, 500X, 1000X.

Input impedance  : display the current input impedance of the channel. Input impedance that available: 1MΩ or 50Ω.

Channel coupling  : display the current channel coupling of the channel. Channel coupling that available: DC, AC, and GND.

Vertical Scale  : display the current vertical scale of the channel. Turn the Vertical Scale Knob to adjust the value.

13. Menu

Display the corresponding function menu of the selected button. Press the corresponding softkey to set the oscilloscope.

14. Data/Time

Display the current data and time.

To Use the Security Lock

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

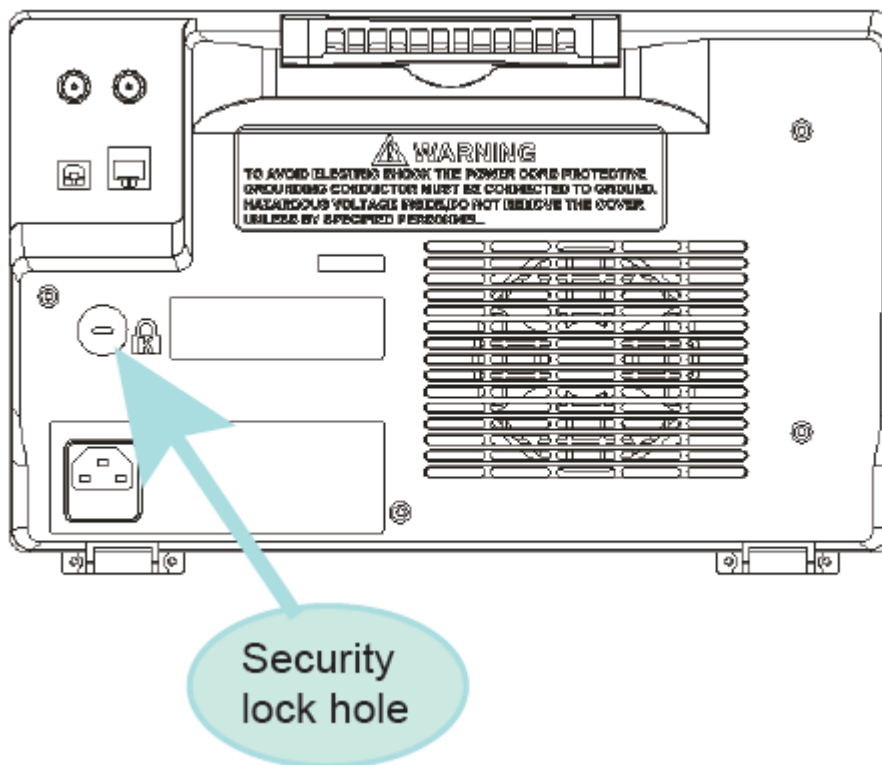


Figure 10 To Use the Security Lock

To Set the Vertical System

This chapter introduces how to set the vertical system of the oscilloscope.

The contents of this chapter:

- ◆ To Enable the Channel
- ◆ To Adjust the Vertical Scale
- ◆ To Adjust the Vertical Position
- ◆ To Specify Channel Coupling
- ◆ To Specify Bandwidth Limit
- ◆ To Specify Probe Attenuation Factor
- ◆ To Specify channel Input Impedance
- ◆ To Specify Amplitude Unit
- ◆ To Invert a Waveform

To Enable the Channel

SDS2000 provides 4 analog input channels (CH1, CH2, CH3 and CH4) and provides independent vertical control system for each channel. As the vertical system setting methods of the 4 channels are the same, this chapter takes CH1 as an example to introduce the setting method of the vertical system.

Connect a signal to the CH1 channel connector; and then press the **CH1** button in the vertical control area (VERTICAL) at the front panel to enable CH1.

The channel setting menu is displayed at the bottom of the screen and the channel label at the right side of the screen. 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.

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

To Adjust the Vertical Scale

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

- **Coarse** adjustment (take counterclockwise as an example): set the vertical scale in 1-2-5 step namely 2 mV/div, 5 mV/div, 10 mV/div, 20 mV/div ... 10 V/div.
- **Fine** adjustment: further adjust the vertical scale within a relatively smaller range to improve vertical resolution. For example: 2 V/div, 1.98V/div, 1.96V/div, 1.94 V/div ... 1 V/div.

If the amplitude of the input waveform is a little bit greater than the full scale under the current 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.

Press the **CH1** button on the front panel; then press the **Adjust** softkey to select the desired mode. Turn the **VERTICAL Variable Knob** to adjust the vertical scale (clockwise to reduce the scale and counterclockwise to increase).

The scale information in the channel label at the right side of the screen will change accordingly during the adjustment. The adjustable range of the vertical scale is related to the probe ratio currently set. By default, the probe attenuation factor is 1X and the adjustable range of the vertical scale is from 2mV/div to 10 V/div.

Note: push the **VERTICAL Variable Knob** to quickly switch between **Coarse** and **Fine** adjustments.

To Adjust the Vertical Position

Turn the **VERTICAL Position Knob** to adjust the vertical position of the channel waveform. Turn the knob clockwise to increase the vertical position and the channel waveform moves up while counterclockwise to reduce the vertical position and the waveform moves down. Push the knob to set the vertical position of the channel waveform to zero.

During the adjustment, the vertical position information Volts Pos displays at the bottom of the screen. The table below shows the range of vertical position according to the volt scale.

Volt Scale	Range of Vertical Position
2 mV/div ~ 100 mV/div	±1V
102 mV/div ~ 1 V/div	±10 V
1.02 V/div ~ 10 V/div	±100 V

To Specify 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 set to **DC**: the DC and AC components of the signal under test can both pass the channel.
- When the coupling mode set to **AC**: the DC components of the signal under test are blocked.
- When the coupling mode set to **GND**: the DC and AC components of the signal under test are both blocked.

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

The current coupling mode is displayed in the channel label at the right side of the screen. You can also press the **Coupling** softkey continuously to switch the coupling mode.

To Specify Bandwidth Limit

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

- When the bandwidth limit set to **Full**, the high frequency components of the signal under test can pass the channel.
- When the bandwidth limit set to **20M**, the high frequency components that exceed 20 MHz are attenuated.

Press the **CH1** button on the front panel; then press the **BW Limit** softkey to select **Full** or **20M**. The default setup is **Full**. When bandwidth limit is enabled, the character **B** will be displayed in the channel label at the right side of the screen.

At 2 mV/div volt scale, the **BW Limit** is set to **20M** automatically, and cannot be turned off.

To Specify Probe Attenuation Factor

Set the probe attenuation factor to match the type of the probe that you are using to ensure correct vertical readouts.

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

The current probe attenuation factor is displayed in the channel label at the right side of the screen. You can also press the **Probe** softkey continuously to switch the probe attenuation factor.

The table shows the probe attenuation factor

Menu	Attenuation Factor
1X	1 : 1
5X	5 : 1
10X	10 : 1
50X	50 : 1
100X	100 : 1
500X	500 : 1
1000X	1000 : 1

To Specify channel Input Impedance

The channel input impedance matching gives you the most accurate measurements because reflections are minimized along the signal path.

- Impedance setting to $1M\Omega$ is for use with many passive probes and for general-purpose measurements. The higher impedance minimizes the loading effect of the oscilloscope on the device under test.
- Impedance setting to 50Ω matches 50 ohm cables commonly used in making high frequency measurements, and 50 ohm active probes.

Press the CH1 button on the front panel; then press the **Impedance** softkey to select the desired impedance.

The current channel input impedance is displayed in the channel label at the right side of the screen.

To Specify Amplitude Unit

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

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

The default setup is **V**.

To Invert a Waveform

When **Invert** is set to **On**, the voltage values of the displayed waveform are inverted. Invert affects how a channel is displayed and it keeps the trigger settings.

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

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

To Set the Horizontal System

This chapter introduces how to set the horizontal system of the oscilloscope.

The contents of this chapter:

- ◆ To Adjust the Horizontal Scale
- ◆ To Adjust the Horizontal Trigger Position
- ◆ To Adjust Vertical Position
- ◆ To Change the Horizontal Time Mode
- ◆ To Use the Zoom Function

To Adjust the Horizontal Scale

Turn the **HORIZONTAL Scale Knob** on the front panel to adjust the horizontal time base. Turn clockwise to reduce the horizontal time base and turn counterclockwise to increase.

The time base information at the upper left corner of the screen will change accordingly during the adjustment. The range of the horizontal scale is from 1ns/div to 50s/div.

The Horizontal Scale Knob works (in the Normal time mode) while acquisitions are running or when they are stopped. When running, adjusting the horizontal scale knob changes the sample rate. When stopped, adjusting the horizontal scale knob lets you zoom into acquired data.

To Adjust the Horizontal Trigger Position

Turn the Horizontal Position Knob on the front panel to adjust the horizontal trigger position (trigger delay) of the waveform. During the modification, waveforms of all the channels would move left or right and the trigger position message at the upper-right corner of the screen would change accordingly. Press down this knob to quickly reset the trigger position.

Changing the delay time moves the trigger point (solid inverted triangle) horizontally and indicates how far it is from the time reference point. These reference points are indicated along the top of the display grid.

All events displayed left of the trigger point happened before the trigger occurred. These events are called pre-trigger information, and they show events that led up to the trigger point.

Everything to the right of the trigger point is called post-trigger information. The amount of delay range (pre-trigger and post-trigger information) available depends on the time/div selected and memory depth.

The horizontal position knob works (in the Normal time mode) while acquisitions are running or when they are stopped. When running, adjusting the horizontal scale knob changes the sample rate. When stopped, adjusting the horizontal scale knob lets you zoom into acquired data.

Panning and Zooming Single or Stopped Acquisitions

When the oscilloscope is stopped, use the horizontal scale and position knobs to pan and zoom in or out your waveform. The stopped display may contain several acquisitions worth of information, but only the last acquisition is available for pan and zoom.

The ability to pan (move horizontally) and scale (expand or compress horizontally) an acquired waveform is important because of the additional insight it can reveal about the captured waveform. This additional insight is often gained from seeing the waveform at different levels of abstraction.

To Change the Horizontal Format

Press the **Horiz** button on the front panel; then press the **Format** softkey and turn the **Universal Knob** to select the desired display format and push the knob to confirm. The default setup is **YT**.

YT

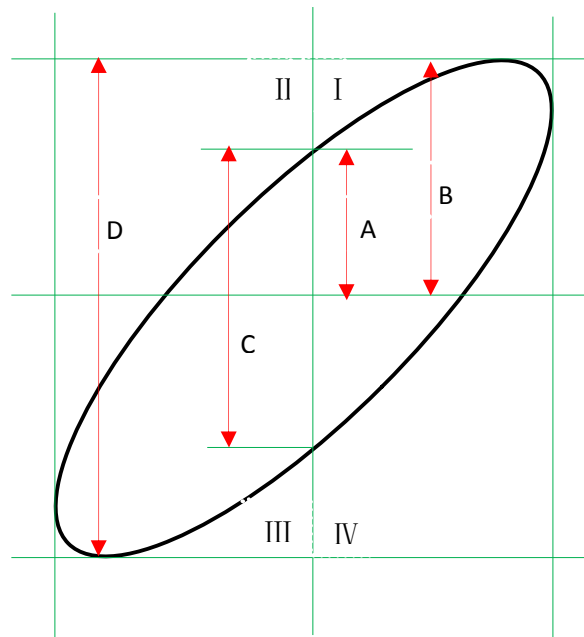
It is the normal viewing mode for the oscilloscope. In the Normal time mode, signal events occurring before the trigger are plotted to the left of the trigger point and signal events after the trigger plotted to the right of the trigger point.

XY

XY mode changes the display from a volt- versus- time display to a volt- versus- volt display. Channel 1 and channel 3 amplitude are plotted on the X- axis and Channel 2 and channel 3 amplitude are plotted on the Y- axis. Channel 1 and channel 2 is a combination while channel 3 and channel 4 is a combination. In each combination, the two channels will be turned on or off together.

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

The phase deviation between two signals with the same frequency can be easily measured via 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: $\theta=\pm\arcsin (A/B)$ 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 $(\pi \text{ to } 3\pi/2)$.

X-Y function can be used to measure the phase deviation occurred 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.

Roll

In Roll mode the waveform moves slowly across the screen from right to left. It only operates on time base settings of 100 ms/div and slower. If the current time base setting is faster than the 100 ms/div limit, it will be set to 100 ms/div when Roll mode is entered.

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

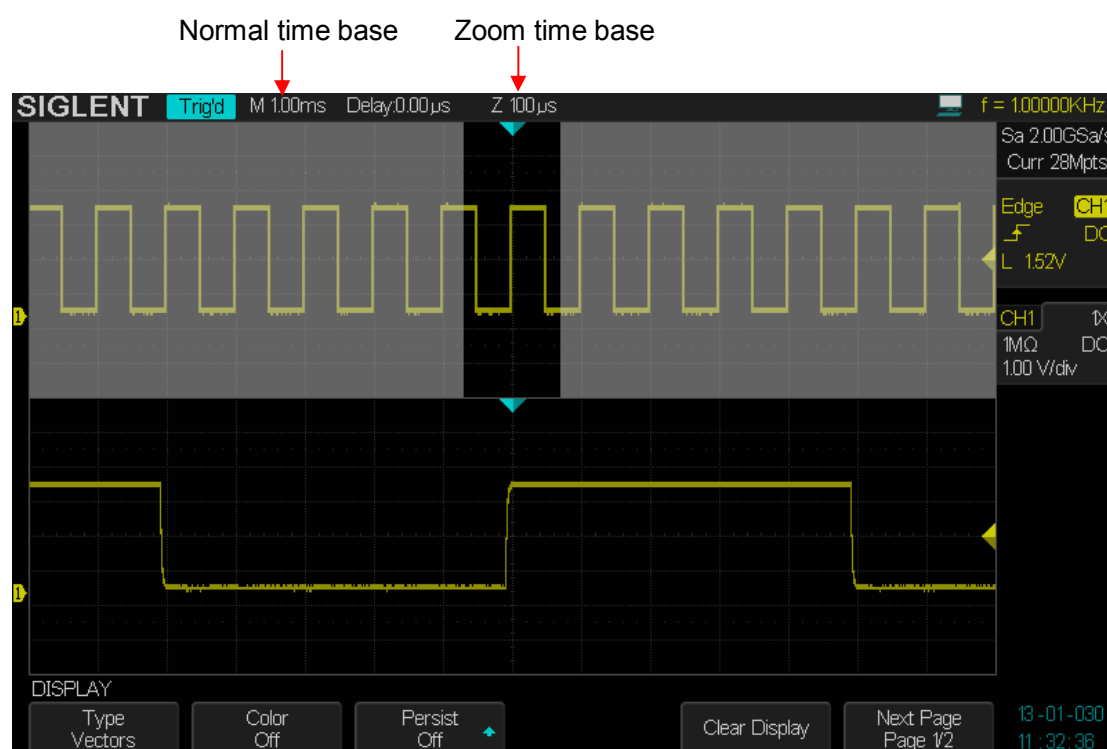
If you would like to stop the display in Roll mode, press the **Run/Stop** button. To clear the display and restart an acquisition in Roll mode, press the **Run/Stop** button again.

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

To Use the Zoom Function

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

Press the **Zoom** button on the front panel to turn on the zoom function, and press the button again to turn off the function. When Zoom function is on, the display divides in half. The top half of the display shows the normal time base window and the bottom half displays a faster Zoom time base window. Also push the **Horizontal Scale Knob** can quickly turn on or off the Zoom function.



The area of the normal display that is expanded is outlined with a box and the rest of the normal display is ghosted. The box shows the portion of the normal sweep that is expanded in the lower half.

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

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

To Set the Sample System

This chapter introduces how to use the run control and set the sampling system of the oscilloscope.

The contents of this chapter:

- ◆ Run Control
- ◆ Overview of Sampling
- ◆ To Specify Memory Depth
- ◆ To Select Sampling Mode
- ◆ Waveform Interpolation Method

Run Control

Press the **Run/Stop** or **Single** button on the front panel to run or stop the sampling system of the scope.

- When the **Run/Stop** b is green, the oscilloscope is running, that is, acquiring data when trigger conditions are met. To stop acquiring data, press the **Run/Stop** button. When stopped, the last acquired waveform is displayed.
- When the **Run/Stop** button is red, data acquisition is stopped. Red "Stop" is displayed next to the trademark logo in the status line at the top of the display. To start acquiring data, press **Run/Stop**.
- To capture and display a single acquisition (whether the oscilloscope is running or stopped), press **Single**. The **Single** run control lets you view single- shot events without subsequent waveform data overwriting the display. Use **Single** when you want maximum memory depth for pan and zoom.

When you press **Single**, the display is cleared, the trigger mode is temporarily set to Normal (to keep the oscilloscope from auto- triggering immediately), the trigger circuitry is armed, the **Single** key is illuminated, and the oscilloscope waits until a user defined trigger condition occurs before it displays a waveform.

When the oscilloscope triggers, the single acquisition is displayed and the oscilloscope is stopped (the **Run/Stop** button is illuminated in red).

Press **Single** again to acquire another waveform

Overview of Sampling

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

Sampling Theory

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

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

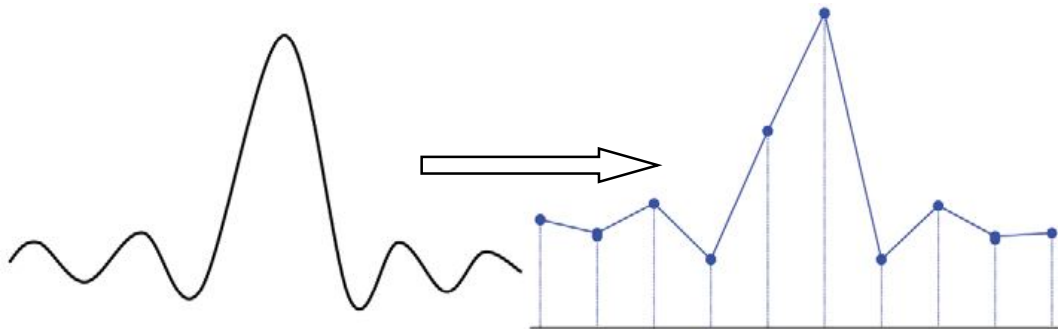
Sample Rate

The maximum sample rate of SDS2000 is 2GSa/s. The actual sample rate of the oscilloscope is determined by the horizontal scale. Turn the **Horizontal Scale Knob** to adjust the sample rate.

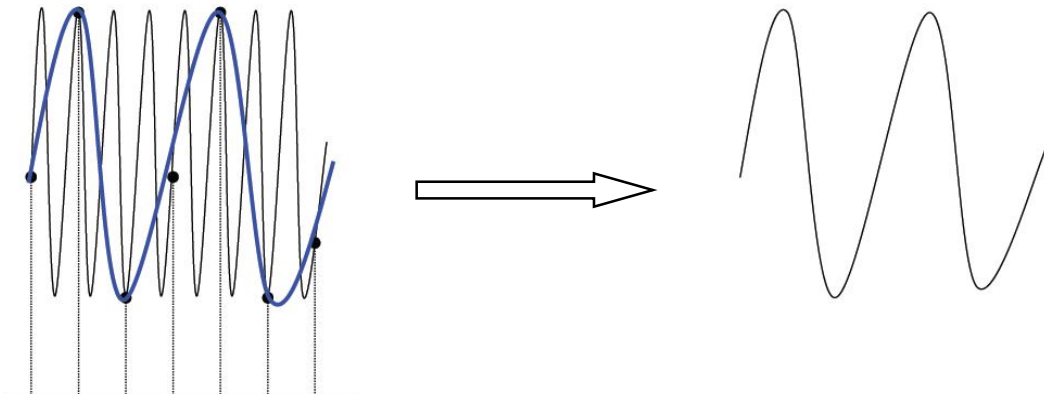
The actual sample rate is displayed in the information area at the upper-right corner of the screen.

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

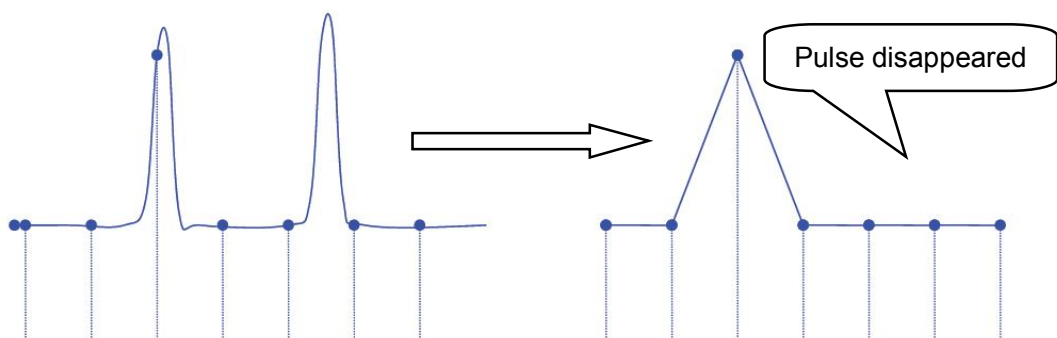
1. **Waveform Distortion:** when the sample rate is too low, some waveform details are lost and the waveform displayed is rather different from the actual signal.



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 fast edge.



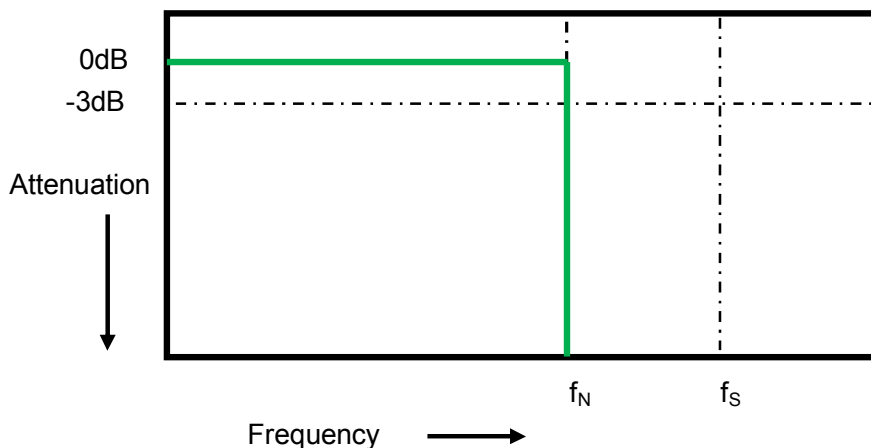
3. **Waveform Leakage:** when the sample rate is too low, the waveform rebuilt from the sample data does not reflect all the actual signal information.



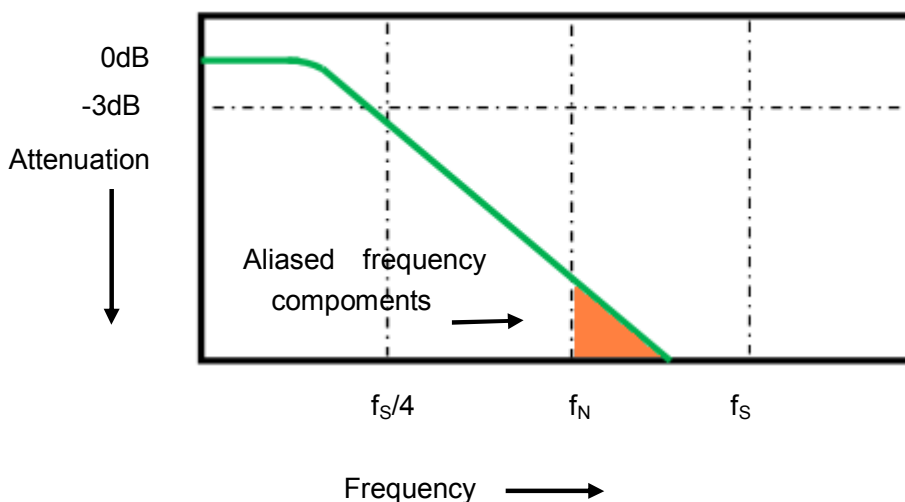
Oscilloscope Bandwidth and Sample Rate

An oscilloscope's bandwidth is typically described as the lowest frequency at which input signal sine waves are attenuated by 3 dB (- 30% amplitude error).

At the oscilloscope bandwidth, sampling theory says the required sample rate is $f_s = 2f_{BW}$. However, the theory assumes there are no frequency components above f_{MAX} (f_{BW} in this case) and it requires a system with an ideal brick-wall frequency response.



However, digital signals have frequency components above the fundamental frequency (square waves are made up of sine waves at the fundamental frequency and an infinite number of odd harmonics), and typically, for 500 MHz bandwidths and below, oscilloscopes have a Gaussian frequency response.



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

So, in practice, an oscilloscope's sample rate should be four or more times its bandwidth: $f_s = 4f_{BW}$. This way, there is less aliasing, and aliased frequency components have a greater amount of attenuation.

To Specify 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. SDS2000 provides up to 28 Mpts memory depth.

Press the **Acquire** button on the front panel; press the **Mem Depth** softkey and then turn the **Universal Knob** to select the desired value and push down the knob to confirm. Press the **Mem Depth** softkey continually can also select the desired value.

The actual memory depth is displayed in the information area at the upper- right corner of the screen. Memory depth that available:7K, 14K, 70K, 140K, 700K, 1.4M, 7K, 14M.

Since the oscilloscope has two acquisition memories, when only one channel is on, the maximal memory depth is up to 28Mpts.

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

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

To Select Sampling Mode

SDS2000 only supports real-time sample. In this mode, the oscilloscope samples and displays waveform within a trigger event. The maximum real-time sample rate of SDS2000 is 2GSa/s.

Press the **RUN/STOP** button 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.

To Specify Waveform Interpolation Method

Under real-time sampling, the oscilloscope acquires the discrete sample values of the waveform being displayed. In general, a waveform of dots display type is very difficult to observe. In order to increase the visibility of the signal, the digital oscilloscope usually uses the interpolation method to display a waveform.

Interpolation method is a processing method to “connect all the sampling points”, and using some points to calculate the whole appearance of the waveform. For real-time sampling interpolation method is used, even if the oscilloscope in a single captures only a small number of sampling points. The oscilloscope can use interpolation method for filling out the gaps between points, to reconstruct an accurate waveform.

Press the **Acquire** button on the front panel to enter the ACQUIRE Function menu; then press the **Sinx/x** softkey to select **Sinx** or **x**.

- **x**: In the adjacent sample points are directly connected on a straight line. This method is only confined to rebuild on the edge of signals, such as square wave.
- **Sinx**: Connecting the sampling points with curves has stronger versatility. Sinx interpolation method uses mathematical processing to calculation results in the actual sample interval. This method bending signal waveform, and make it produce more realistic regular shape than pure square wave and pulse. When the sampling rate is 3 to 5 times the bandwidth of the system. Recommended sinx interpolation method.



Figure 11 Display Type Set to Dots



Figure 12 x Interpolation



Figure 13 Sinx Interpolation

To Select Acquisition Mode

The acquisition mode is used to control how to generate waveform points from sample points. SDS2000 provides the following acquisition mode: Normal, Peak Detect, Average and High Resolution.

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

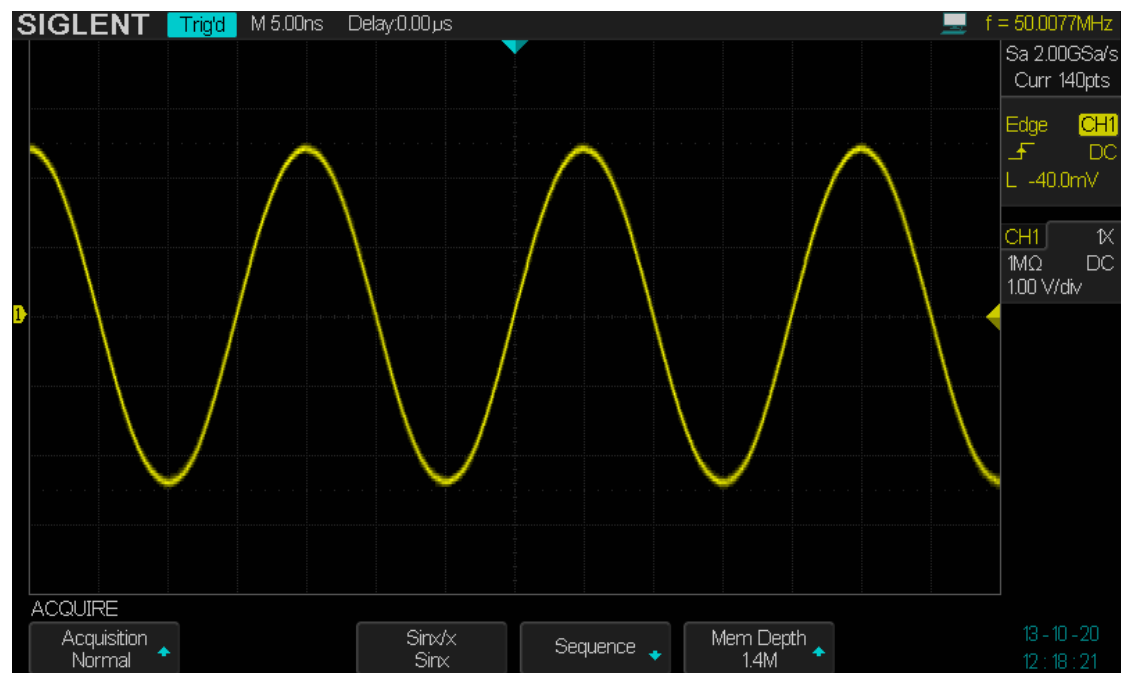


Figure 14 Acquisition System

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. It is the default acquisition mode.

Peak Detect

In this mode, the oscilloscope acquires the maximum and minimum values of the signal within the sample interval to get the envelope of the signal 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.

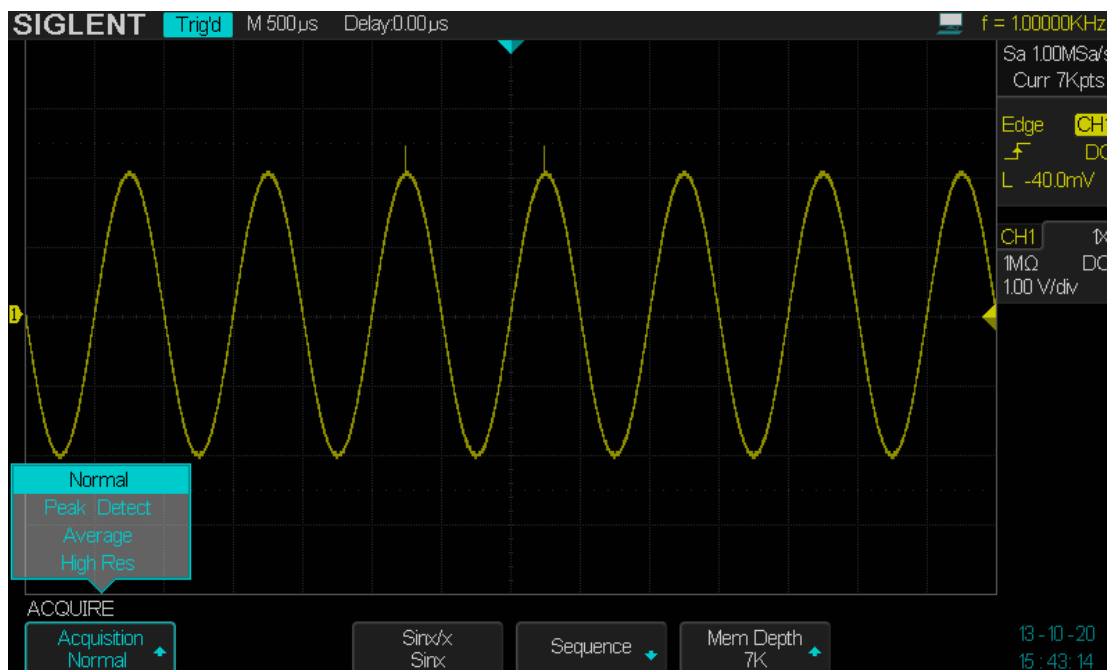


Figure 15 Sine With Glitch, Normal Mode

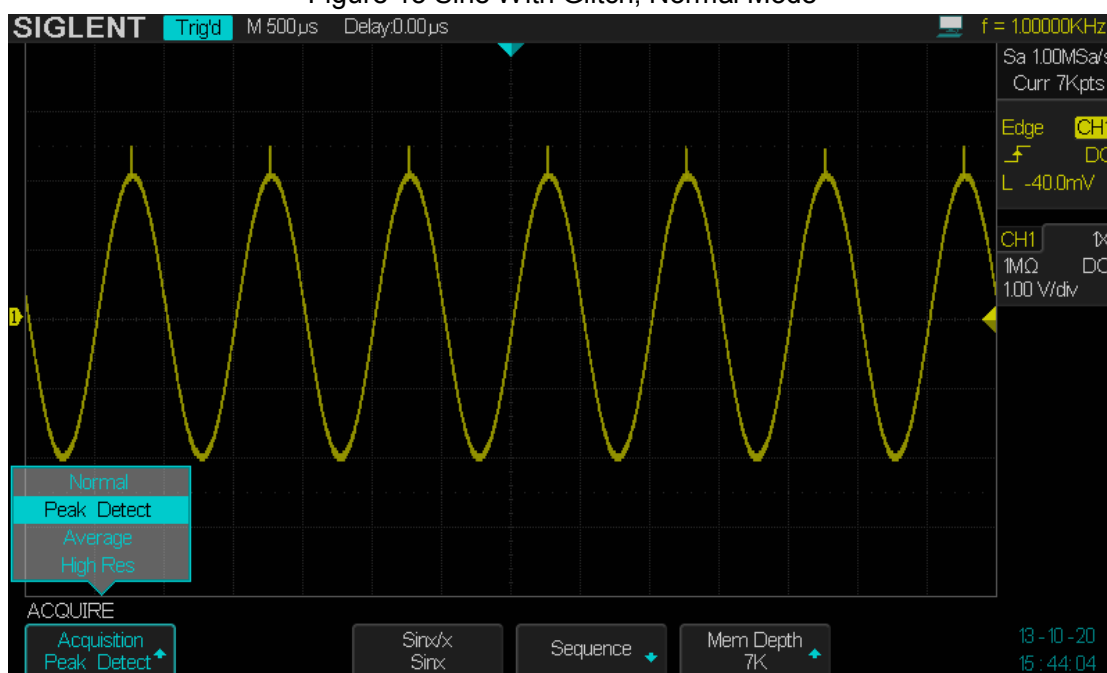


Figure 16 Sine With Glitch, Peak Detect Mode

Average

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

The available range of the number of averages is from 4 to 1024 and the default is 4.

When Average mode is selected, press **Averages** and turn the universal knob or press the softkey continually to set the desired average time.



Figure 17 Sine With Random Noise, Normal Mode

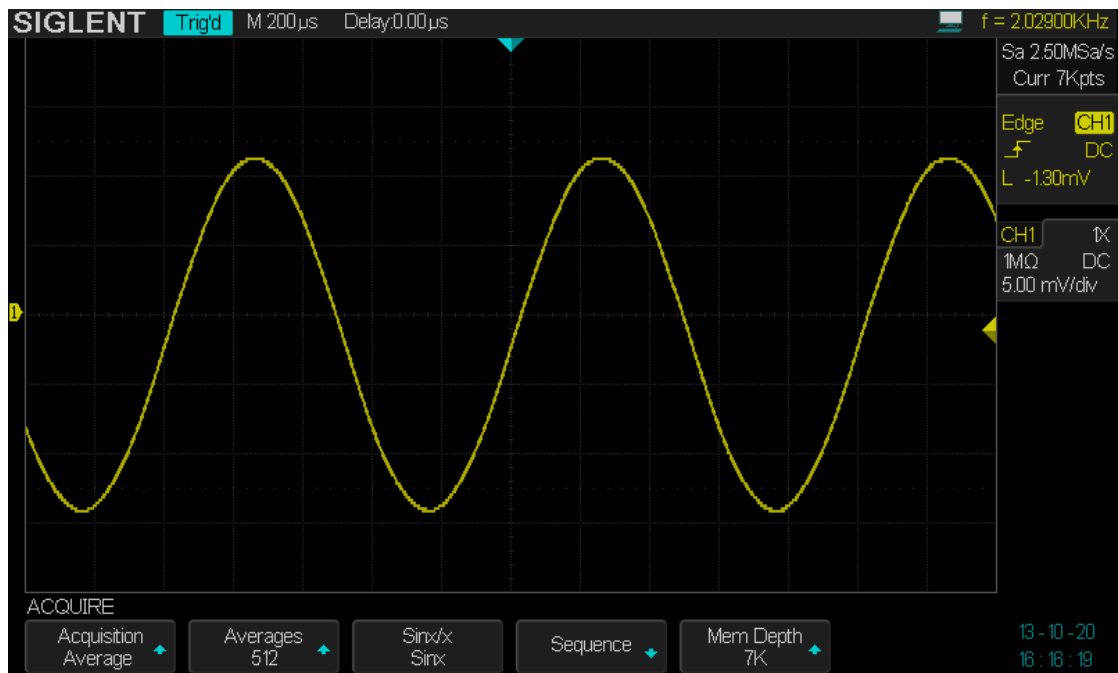


Figure 18 Sine With Random Noise, Average Mode

High Resolution

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

High Resolution mode can be used on both single-shot and repetitive signals and it does not slow waveform update. This mode limits the oscilloscope's real-time bandwidth because it effectively acts like a low-pass filter.

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

To Use Sequence Mode

Sequence is also a kind of acquisition mode, which does not display waveform during sampling process. It improves the waveform capture rate, and the maximal capture rate is more than 300,000 wfs/s. So it can capture the small probability event effectively.

The oscilloscope runs and fills a memory segment for each trigger event. The oscilloscope is busy acquiring multiple segments. The oscilloscope continues to trigger until memory is filled, and then display the waveforms on the screen. Because of the limit of display, the oscilloscope can display at most 20 frames waveform of per channel at the same time.

Press the Run/Stop button can view all the frames in details.

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

Do the following steps to use the sequence mode.

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



Figure 19 SEQUENCE Function Menu

3. Press the **Acq.Mode** softkey to select **On** to turn on the sequence mode. The default setup is **Off**.
4. Press the **Display Mode** softkey to select the desired display mode.
 - **Overlay**: display all the frames by overlaying.
 - **Waterfall**: display all the frames like a waterfall.

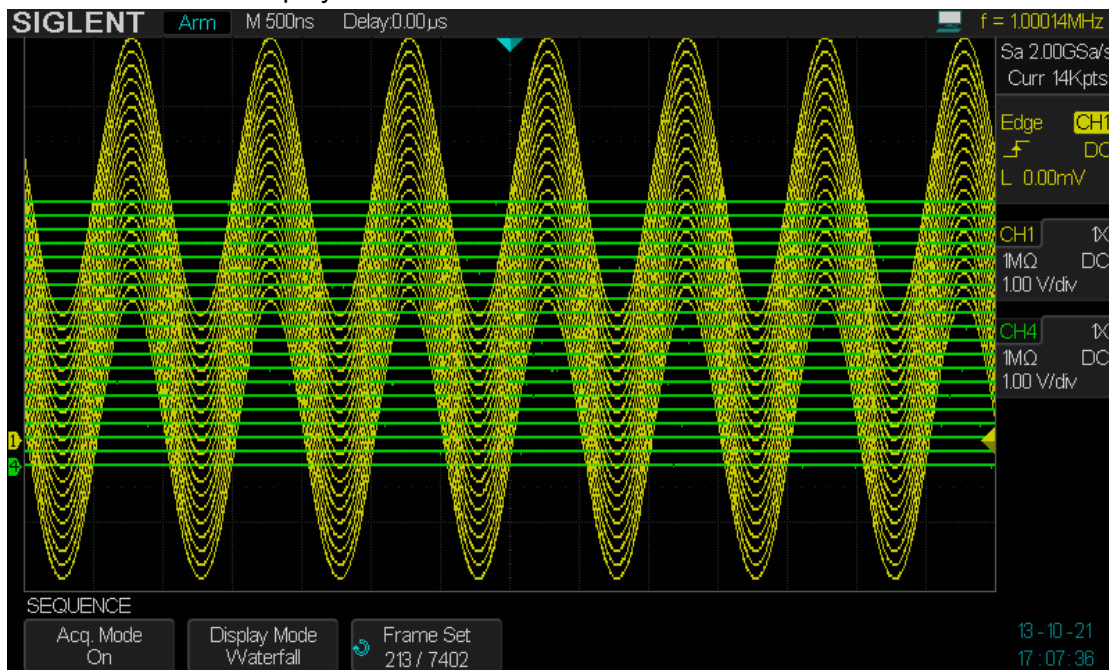


Figure 20 Waterfall Display Mode

5. Press the **Frame Set** softkey; and then turn the **Universal Knob** to select the desired value.

- The **Frame Set** display format is A/B, A is the frame number you set by turning the **Universal Knob** and B is the maximal number that you can set under the time base.
6. Press the **Run/Stop** button on the front panel to stop acquisition and it will display the replay options shown as the picture below. The **Start**, **End** and **Operation** softkeys only appear under stop state.

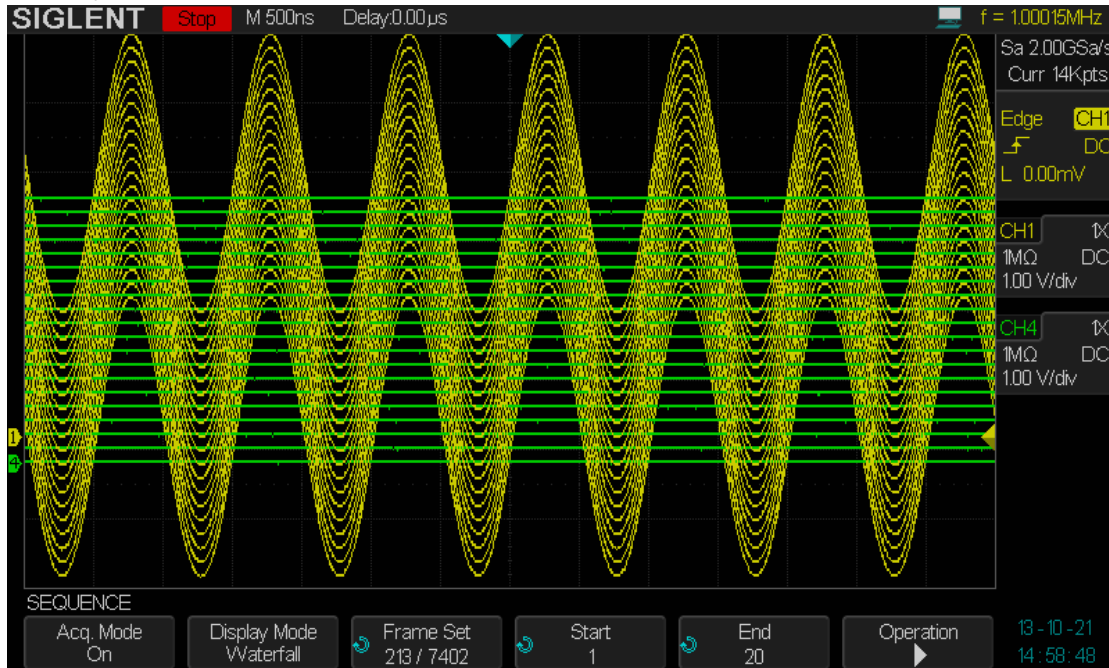




Figure 21 Stop under Sequence Mode

7. Press the **Start** softkey; then turn the **Universal Knob** to select the desired value. It is the start frame to be replay and its default value is 1.
8. Press the **End** softkey; then turn the **Universal Knob** to select the desired value. It is the last frame to display together with the **Start** frame and its default value is 20 when the **Frame Set** value is not less than 20. The End value shouldn't be less than the Start value
 - The difference between the **End** and the Start shouldn't be larger than 19.
 - When the difference between the **End** and the **Start** value is 19, increase the **End** value will also increase the **Start** value. They will keep the difference 19 until to the last frame.
 - When the **End** value is equal to the **Start** value, decrease the **End** value will also decrease the **Start** value. They will keep the same value until to 1.
9. Press the **Operation** softkey to start replay the waveform according the **Start** and **End** frame set above.
 -  : It is stop now, press the softkey to start.
 -  : It is running now, press the softkey to pause. After pause, press again to continue.

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

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

3. Press the **Acq.Mode** softkey to select **On** to turn on the sequence mode.
4. Press the **Display Mode** softkey to select the desired display mode.
5. Press the **Frame Set** softkey; and then turn the **Universal Knob** to select the desired value.
6. Press the **Run/Stop** button on the front panel to stop acquisition
7. Press the **Utility** button on the front to enter the UTILITY function menu.
8. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
9. Press the **History** softkey to enter the HISTORY function menu.

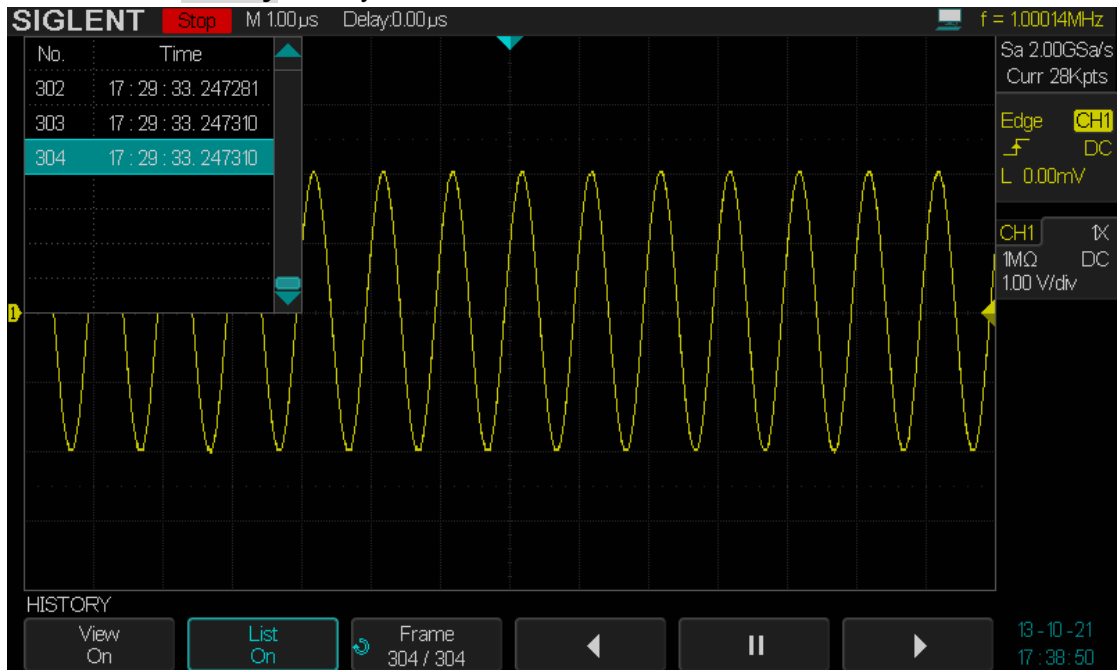





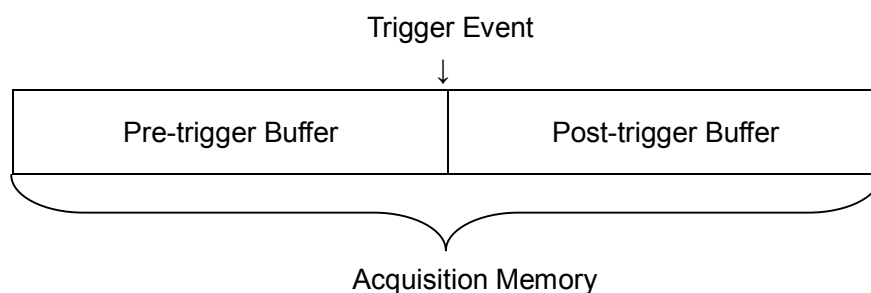
Figure 22 HISTORY Function Menu

10. Press the **View** softkey to select On to turn on the history function.
11. Press the **List** softkey to turn on or off the list display. The list records the acquisition time of every frame and shows the frame number that displaying on the screen.
12. Press the **Frame** softkey; and then turn the **Universal Knob** to select the frame to display.
13. Press the  softkey to replay the waveform from the current frame to 1.
14. Press the  softkey to stop replay.
15. Press the  softkey to replay the waveform from the current frame to the last frame.

To Trigger the Oscilloscope

For trigger, you set certain trigger condition according to the requirement and when a waveform in the waveform stream meets this condition, the oscilloscope captures this waveform as well as the neighboring part and displays them on the screen. For digital oscilloscope, it displays waveform continuously no matter whether it is stably triggered, but only stable trigger can ensure stable display. The trigger circuit ensures that every time base sweep or acquisition starts from the input signal and the user-defined trigger condition, namely every sweep is synchronous to the acquisition and the waveforms acquired overlap to display stable waveform.

The following is the schematic diagram of the acquisition memory. As shown in the figure below, the position of the trigger event is determined by the reference time point and the delay setting.



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.

SDS2000 provides abundant advanced trigger functions which can help you to focus on the desired waveform details. These trigger types are edge, slope, pulse, video, window, interval, dropout, runt, pattern and serial trigger. This chapter will mainly introduce all these trigger functions which mentioned above in details and tell you how to set the trigger conditions to capture desired waveform.

The contents of this chapter:

- ◆ Trigger Source
- ◆ Trigger Mode
- ◆ Trigger Level
- ◆ Trigger Coupling
- ◆ Trigger Holdoff
- ◆ Noise Rejection
- ◆ Trigger Type
 - Edge Trigger
 - Slope Trigger
 - Pulse Trigger
 - Video Trigger
 - Window Trigger
 - Interval trigger
 - DropOut Trigger
 - Runt Trigger
 - Pattern Trigger

Trigger Source

SDS2000 trigger source includes analog channels (**CH1**, **CH2**, **CH3** and **CH4**), **EXT**, **EXT/5** and **AC Line**.

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

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

Analog channel input:

Signals input from analog channels **CH1**, **CH2**, **CH3**, and **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 connect external trigger signal to the EXT TRIG channel when all of the four channels are sampling data. The trigger signal (such as external clock and signal of the circuit to be tested) will be connected to **EXT** and **EXT/5** trigger source via the **[EXT TRIG]** connector. **EXT/5** trigger source attenuates the signal by a factor of 5. It extends the trigger level. You can set the trigger condition within the range of trigger level (-8 div to +8 div).

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 signal (such as illuminating device) and power (power supply device). For example, it is mainly used in related measurement of the power industry to stably trigger the waveform output from the transformer of a transformer substation.

Note: to select stable channel waveform as the trigger source to stabilize the display.

Trigger Mode

SDS2000 trigger mode includes auto, normal and single. Trigger mode affects the way in which the oscilloscope searches for the trigger

After the oscilloscope starts running, 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 (First Input First Out, FIFO).

When a trigger is found, the pre- trigger buffer contains the events that occurred just before the trigger. Then, the oscilloscope fills the post- trigger buffer and displays the acquisition memory.

Press the **Auto**, **Normal** and the **Single** buttons on the front panel to select the desired trigger mode, and the corresponding status light will be lighted.

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

The **Auto** trigger mode is appropriate when:

- Checking DC signals or signals with unknown levels or activity.
- When trigger conditions occur often enough that forced triggers are unnecessary.

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

The **Normal** trigger mode is appropriate when:

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

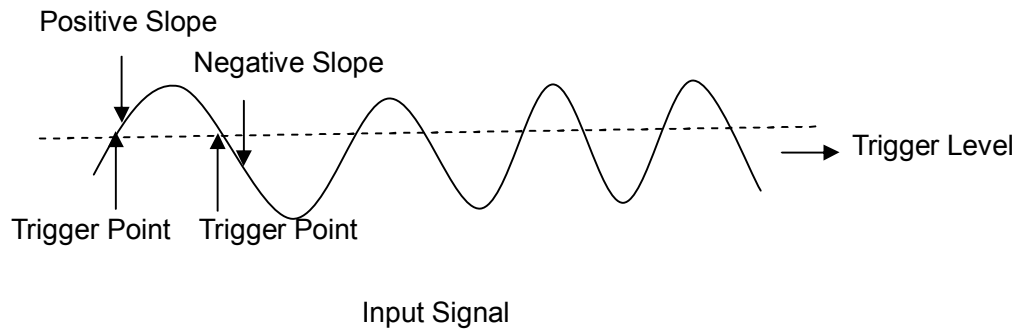
- In the Single trigger mode, the oscilloscope waits for a trigger and displays the waveform when the trigger condition is met and then stops.

The **Single** trigger mode is appropriate when:

- To capture single event or aperiodic signal.
- To capture burst or other unusual signals.

Trigger Level

Trigger level and slope define the trigger point,



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

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

The position of the trigger level for the analog channel is indicated by the trigger level icon ◀ (If the analog channel is on) at the left side of the display. The value of the analog channel trigger level is displayed in the upper- right corner of the display.

Trigger Coupling

Press the **Setup** button on the front panel to enter the TRIGGER function menu, and then press the **Coupling** softkey and turn the **Universal Knob** or press the **Coupling** softkey continually to select the desired coupling mode.

SDS2000 provides 4 kinds of trigger coupling modes:

- **DC**: allow DC and AC components into the trigger path.
- **AC**: block all the DC components and attenuate signals lower than 5.8 Hz. Use AC coupling to get a stable edge trigger when your waveform has a large DC offset.
- **LF Reject**: block the DC components and reject the low frequency components lower than 2.08MHz. Low frequency reject removes any unwanted low frequency components from a trigger waveform, such as power line frequencies, etc., that can interfere with proper triggering. Use **LF Reject** coupling to get a stable edge trigger when your waveform has low frequency noise.
- **HF Reject**: reject the high frequency components higher 1.27MHz)

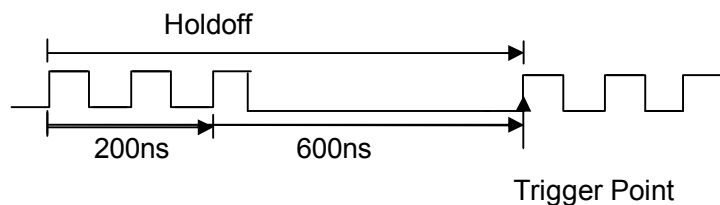
Note: trigger coupling has nothing to do with the channel coupling.

Trigger Holdoff

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

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

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



The correct holdoff setting is typically slightly less than one repetition of the waveform. Set the holdoff to this time to generate a unique trigger point for a repetitive waveform. Only edge trigger and serial trigger have holdoff option. The holdoff time of SDS2000 is adjustable from 100ns to 1.5s.

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

Note: adjust the time scale and horizontal position will not affect the holdoff time.

Noise Rejection

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

Press the **Setup** button on the front panel, and then press the **Noise Reject** softkey continually to set the option to **On** or **Off** to turn on or off the noise rejection function.

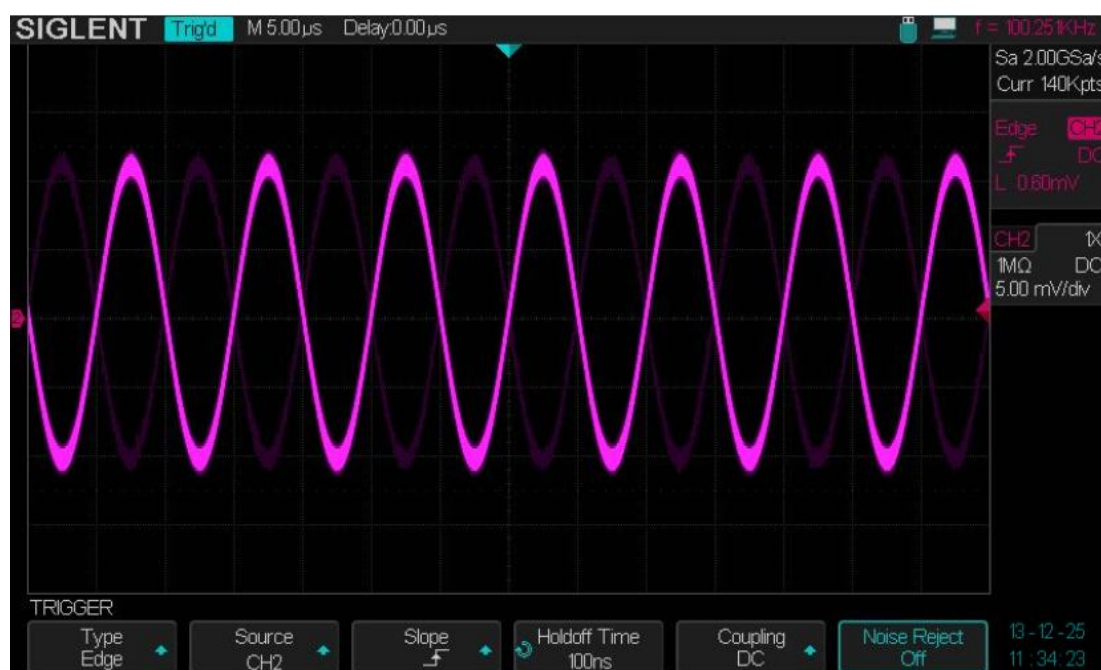


Figure 23 Turn off the Noise Reject

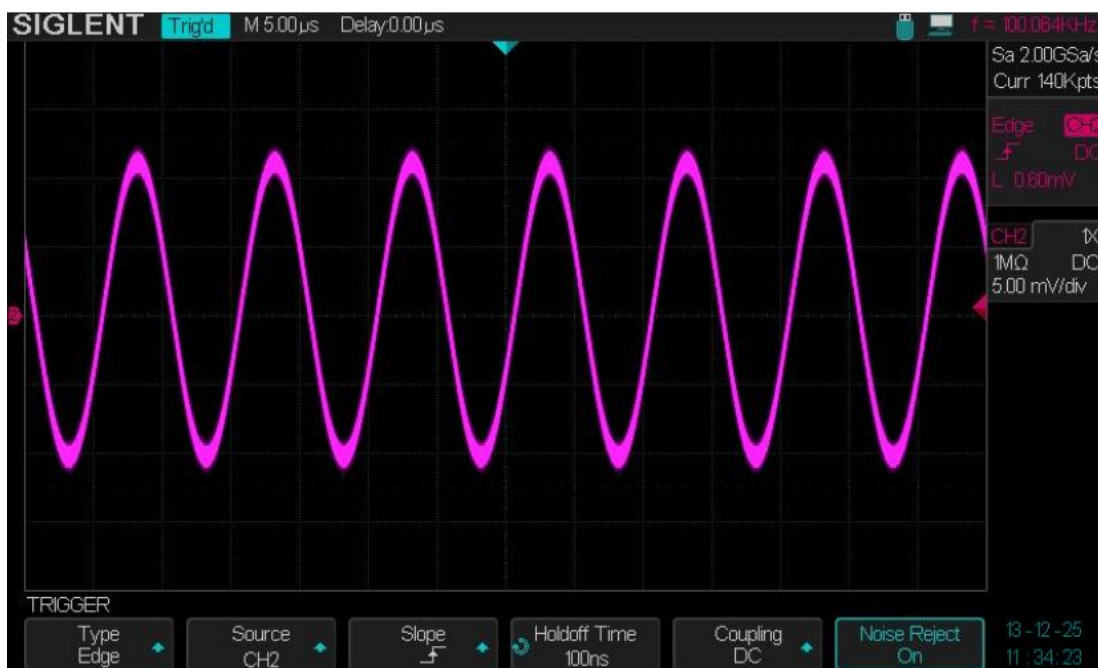


Figure 24 Turn on the Noise Reject

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

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

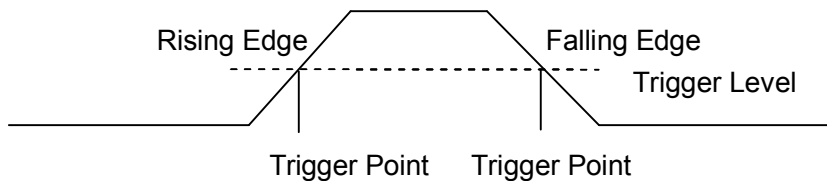
Trigger Type

SDS2000 provides abundant advanced trigger functions, including various serial bus triggers.

- ◆ Edge trigger
- ◆ Slope trigger
- ◆ Pulse trigger
- ◆ Video trigger
- ◆ Window trigger
- ◆ Interval trigger
- ◆ DropOut trigger
- ◆ Runt trigger
- ◆ Pattern trigger

Edge Trigger

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



1. Press the **Setup** button on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** softkey; turn the **Universal Knob** to set select **Edge** and then push the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select **CH1**, **CH2**, **CH3**, **CH4**, **EXT**, **EXT/5** or **AC Line** as the trigger source.
4. Press the **Slope** softkey; turn the **Universal Knob** to select the desired trigger edge (rising, falling or rising & falling), and then press down the knob to confirm. The current trigger slope is displayed at the upper right corner of the screen.
5. Turn the **Trigger Level Knob** to adjust the trigger level to obtain stable trigger.

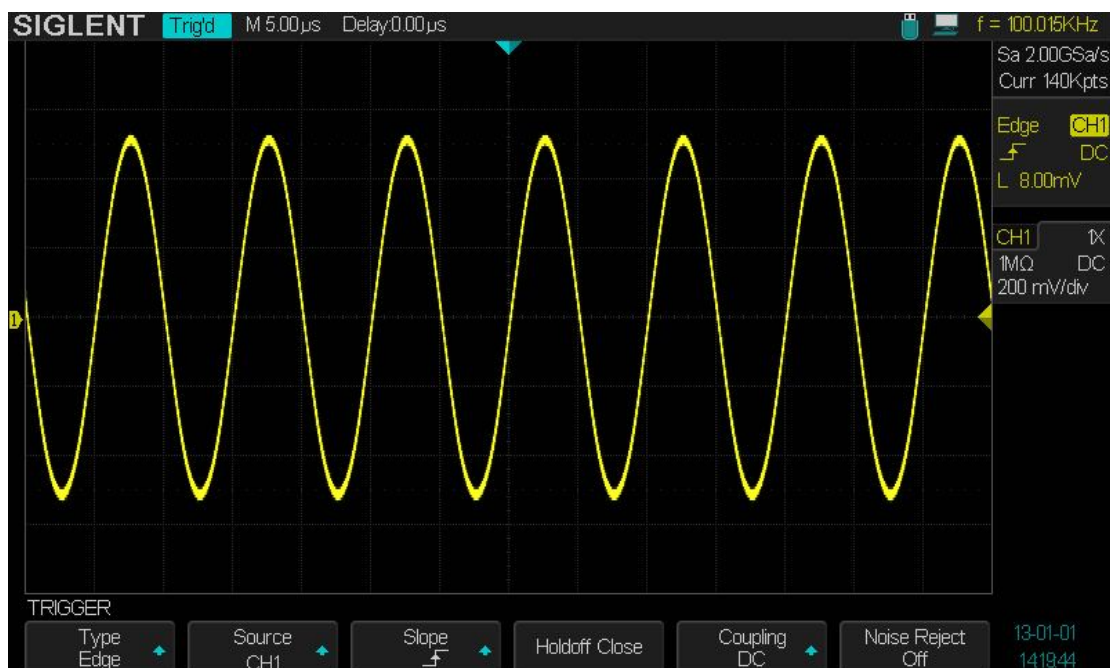


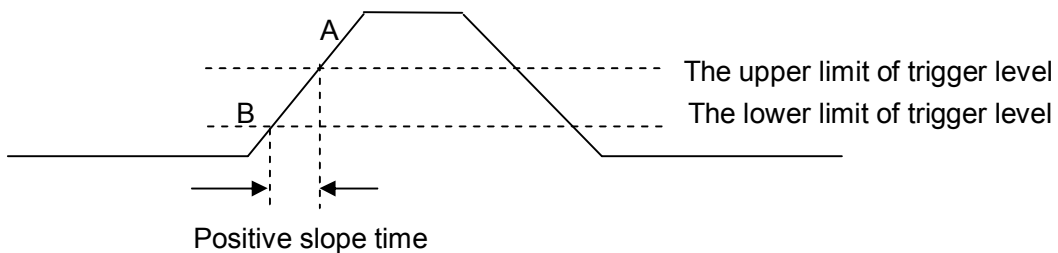
Figure 25 Edge Trigger

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

Slope Trigger

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

In SDS2000, positive slope time is defined as the time difference between the two crossing points of trigger level line A and B with the positive edge as shown in the figure below.



1. Press the **Setup** button on the front panel to enter the TRIGGER function menu.
2. Press the **Type** softkey; turn the **Universal Knob** to set select **Slop** and then push the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select CH1, CH2, CH3 or CH4 as the trigger source.
4. Press the Slop softkey; turn the **Universal Knob** to set select the desired trigger edge (rising or falling), and then push down the knob to confirm. The current trigger slope is displayed at the upper right corner of the screen.
5. Press **Lower Upper** softkey to select the **Lower** or **Upper** trigger level; then turn the **Trigger Level Knob** to adjust the position. The trigger level values are displayed at the upper right corner of the screen.

The lower trigger level cannot be upper than the upper trigger level. In the trigger state message box, L1 means the upper trigger lever while L2 means the lower trigger level.

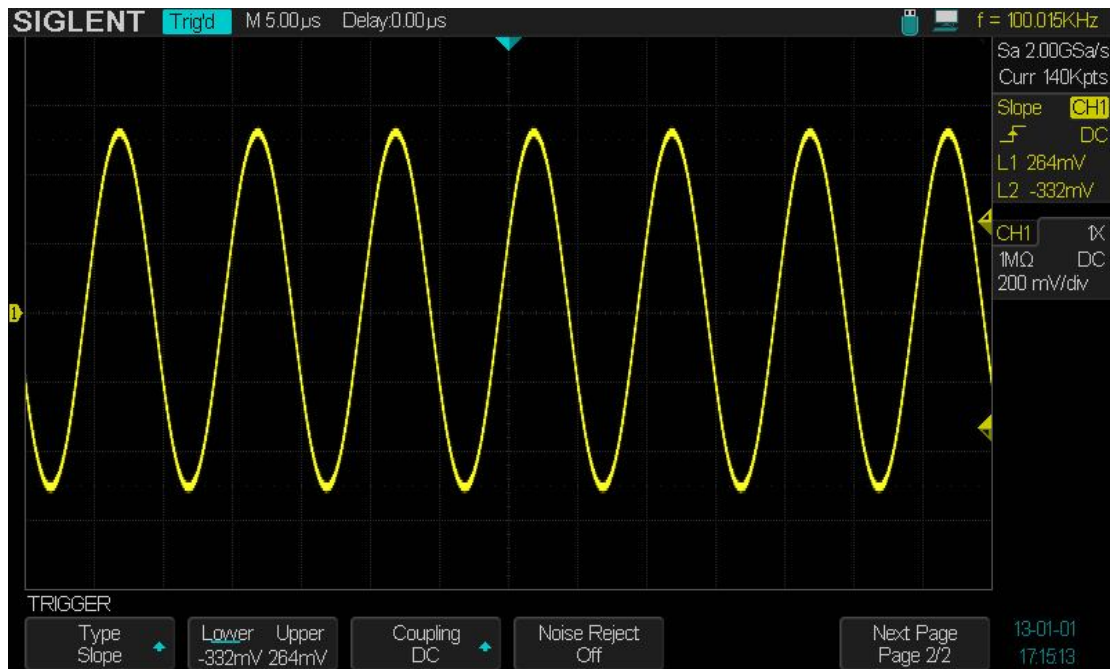
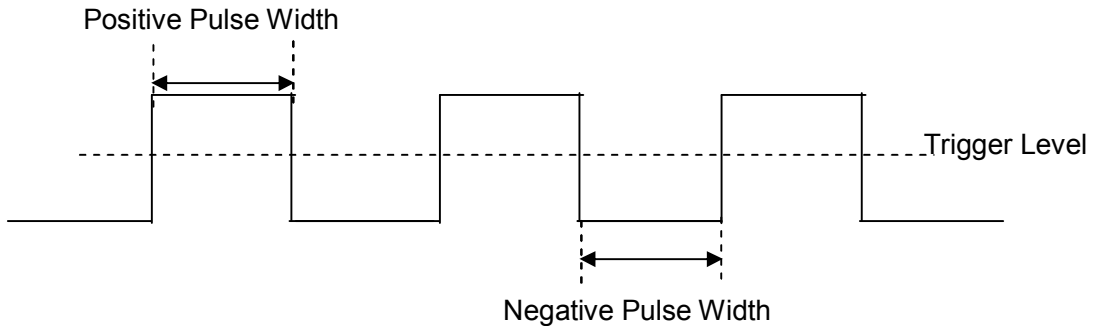


Figure 26 Slope Trigger

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

Pulse Trigger

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



1. Press the **Setup** button on the front panel to enter the TRIGGER function menu.
2. Press the **Type** softkey; turn the **Universal Knob** to select **Pulse** and then push the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select **CH1**, **CH2**, **CH3** or **CH4** as the trigger source.
4. Turn the **Trigger Level Knob** to adjust the trigger level to the desired place.
5. Press the **Polarity** softkey to select **Positive** or **Negative** pulse that to trigger on. The current trigger polarity is displayed at the upper right corner of the screen.
6. Press the **Limit Range** softkey; turn the **Universal Knob** to select the desired condition.

- **<** (less than a time value): trigger when the positive or negative pulse time of the input signal is lower than the specified time value.
For example, for a positive pulse, if you set t (pulse real width) $< 100\text{ns}$, the waveform will trigger.

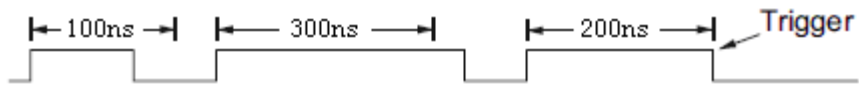


- **>** (greater than a time value): trigger when the positive or negative pulse time of the input signal is greater than the specified time value.
For example, for a positive pulse, if you set t (pulse real width) $> 100\text{ns}$, the waveform will trigger.



- $< >$ (within a range of time value): trigger when the positive or negative pulse time of the input signal is greater than the specified lower limit of time and lower than the specified upper limit of time value.

For example, for a positive pulse, if you set t (pulse real width) $> 100\text{ns}$ and $t < 300\text{ns}$, the waveform will trigger.



- $> <$ (outside a range of time value): trigger when the positive or negative pulse time of the input signal is greater than the specified upper limit of time and lower than the specified lower limit of time value.

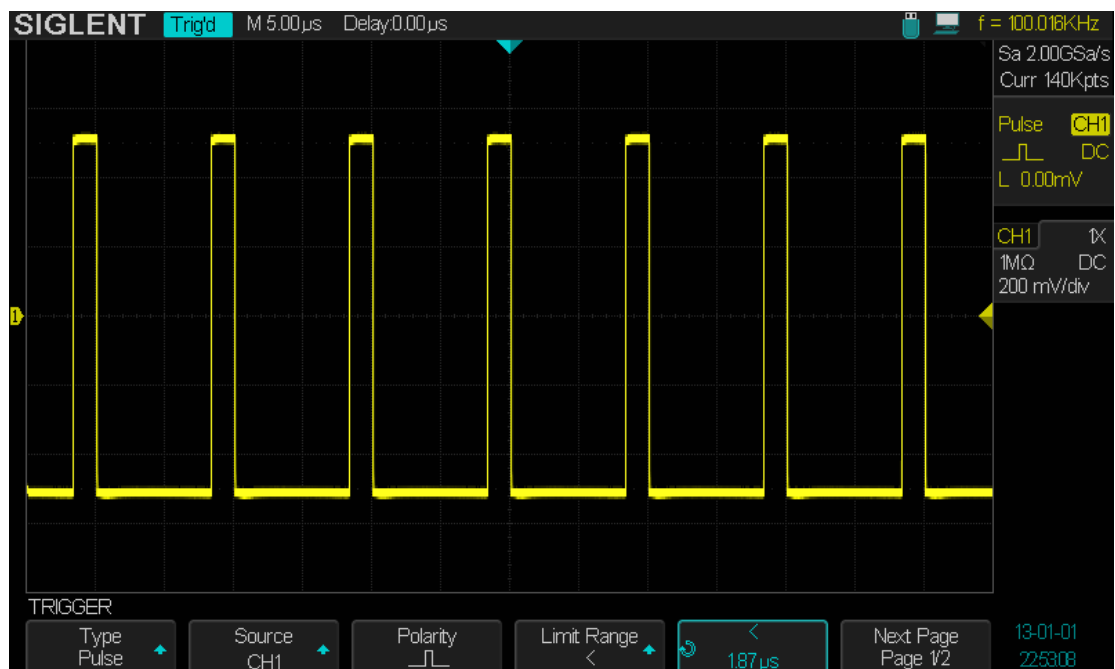


Figure 27 Pulse Trigger

Video Trigger

Video triggering can be used to capture the complicated waveforms of most standard analog video signals. The trigger circuitry detects the vertical and horizontal interval of the waveform and produces triggers based on the video trigger settings you have selected. SDS2000 supports standard video signal field or line of NTSC (National Television Standards Committee), PAL (Phase Alternating Line) HDTV (High Definition Television) and custom video signal trigger.

1. Press the **Setup** button on the front panel to enter the TRIGGER function menu.
2. Press the **Type** softkey; then turn the **Universal Knob** to select **Video** and push down the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select **CH1**, **CH2**, **CH3** or **CH4** as the trigger source. Turn the **Trigger Level Knob** does not change the trigger level because the trigger level is automatically set to the sync pulse.
4. Press the **Standard** softkey to select the desired video standard. SDS2000 supports the following video standards.

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

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

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

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

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

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

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

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

The following are exercises to familiarize you with video triggering.

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

To Trigger on a Specific Line of Video

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

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

1. Press the **Setup** button on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** softkey; then use the **Universal Knob** to select Video and push down the knob to confirm.
3. Press the **Source** softkey; turn the Universal Knob to select CH1 as the trigger source, and press the knob to confirm.
4. Press the **Standard** softkey; turn the Universal Knob to select NTSC, and press the knob to confirm.
5. Press the **Sync** softkey and set the option to **Select**; press the **Line** softkey and then turn the universal to select **0124** and push the knob to confirm; press the **Field** softkey and then turn the **Universal Knob** to select **2** and push the knob to confirm

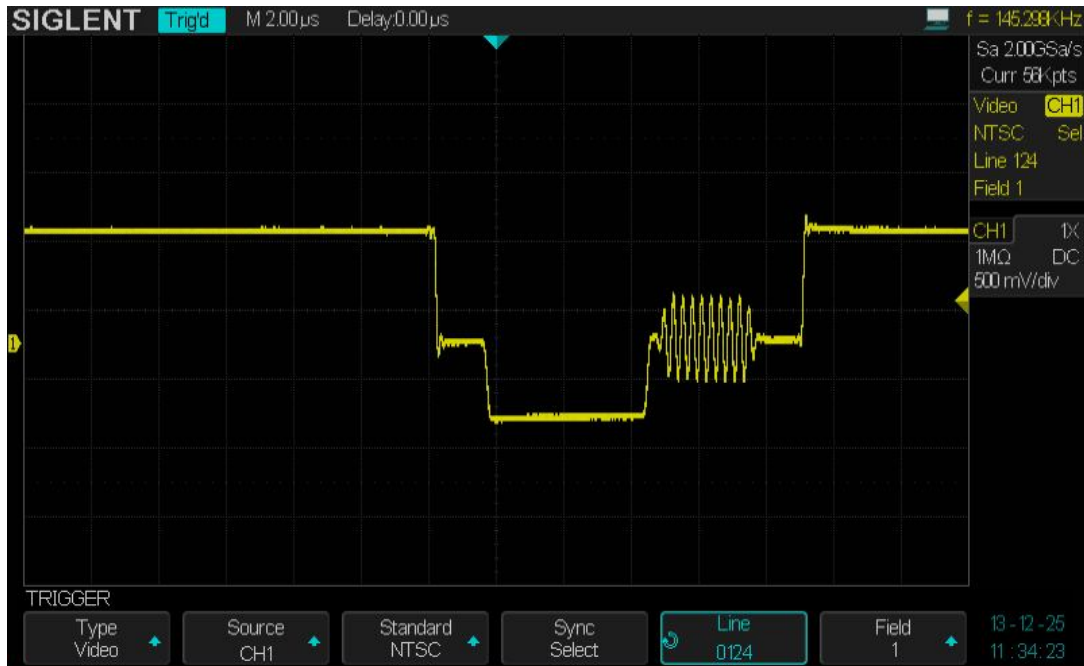


Figure 28 Video Trigger

To Use Custom Video Trigger

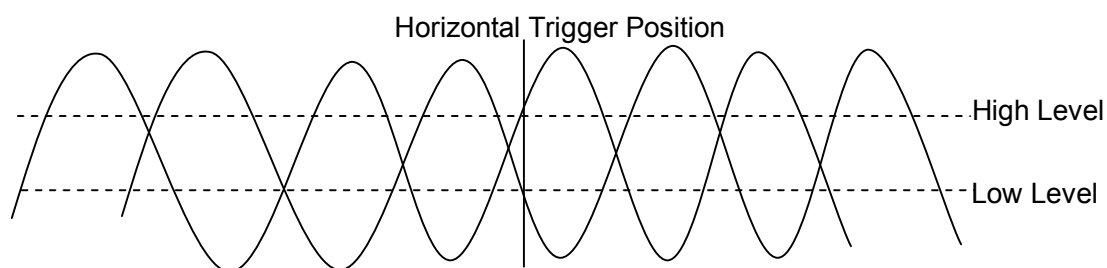
Custom video trigger supports frame rate of 25Hz, 30Hz, 50Hz and 60Hz, and the line range is available from 300 to 2000. The steps below show how to set custom trigger.

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

Window Trigger

Windows trigger provides a high trigger level and a low trigger level. The instrument triggers when the input signal passes through the high trigger level or the low trigger level.

There are two kinds of window types: Absolute and Relative. They have different trigger level adjustment methods. Under Absolute window type, the lower and the upper trigger levels can be adjusted respectively via the Level knob; under Relative window type, adjust the Center value to set the window center; adjust the Delta value to set the window range, the lower and the upper trigger levels always move together.



- If the lower and the upper trigger levels are both within the waveform amplitude range, the oscilloscope will trigger on both rising and falling edge.
- If the upper trigger level is within the waveform amplitude range while the lower trigger level is out of the waveform amplitude range, the oscilloscope will trigger on rising edge only.
- If the lower trigger level is within the waveform amplitude range while the upper trigger level is out of the waveform amplitude range, the oscilloscope will trigger on falling edge only.

To set window trigger via Absolute window type:

1. Press the **Setup** button on the front panel to enter the TRIGGER function menu.
2. Press the **Type** softkey; then use the **Universal Knob** to select **Window** and push down the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select **CH1**, **CH2**, **CH3** or **CH4** as the trigger source.
4. Press the Window **Type** softkey to select Absolute.
5. Press the **Lower Upper** softkey to select **Lower** or **Upper** trigger level; then turn the **Trigger Level Knob** to adjust the position. The trigger level values are displayed at the upper right corner of the screen.

The Lower trigger level cannot be upper than the upper trigger level. In the trigger state message box, **L1** means the upper trigger level while **L2** means the lower trigger level.

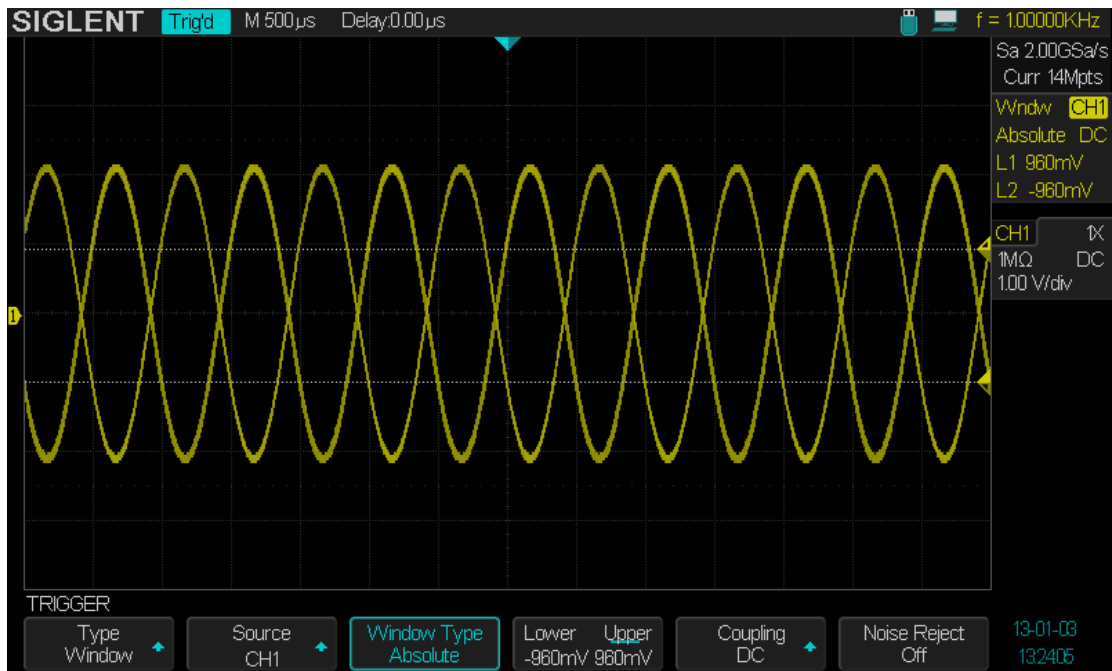


Figure 29 Absolute Window Trigger

To set window trigger via Relative window type:

1. Press the **Setup** button on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** softkey; then use the **Universal Knob** to select **Window** and push down the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select **CH1**, **CH2**, **CH3** or **CH4** as the trigger source.
4. Press the **Window Type** softkey to select **Relative**.
5. Press the **Center Delta** softkey to select **Center** or **Delta** trigger level mode; then turn the **Trigger Level Knob** to adjust the position. The **Center** and **Delta** values are displayed at the upper right corner of the screen.

In the trigger state message box, **C** means **Center**, the center value of the lower and upper trigger levels; **D** means **Delta**, the difference between the lower (or upper) trigger level and the trigger level center.

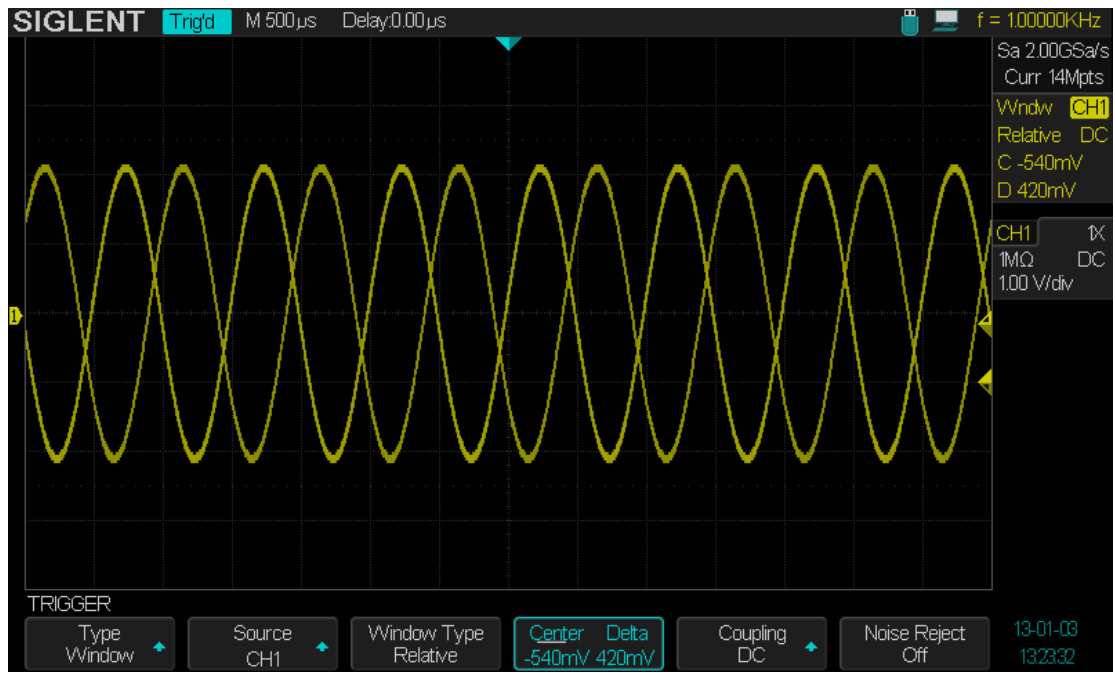
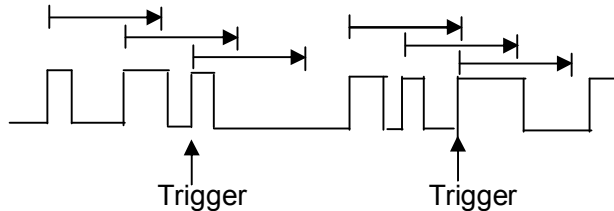


Figure 30 Relative Window Trigger

Interval Trigger

Trigger when the times difference between the neighboring rising or falling edges meets the time limit (<, >, <>, ><).



To set interval trigger:

1. Press the **Setup** button on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** softkey; then use the **Universal Knob** to select **Interval** and push down the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select CH1, CH2, CH3 or CH4 as the trigger source.
4. Press the **Slope** softkey to select rising or falling edge.
5. Press the **Limit Range** softkey; turn the **Universal Knob** to select desired condition.
 - < (less than a time value): trigger when the positive or negative pulse time of the input signal is lower than the specified time value.
 - > (greater than a time value): trigger when the positive or negative pulse time of the input signal is greater than the specified time value.
 - < > (within a range of time value): trigger when the positive or negative pulse time of the input signal is greater than the specified lower limit of time and lower than the specified upper limit of time value.
 - > < (outside a range of time value): trigger when the positive or negative pulse time of the input signal is greater than the specified upper limit of time and lower than the specified lower limit of time value.
6. Press the **Time Setting** softkey (<, >, <>, ><), turn the **Universal Knob** to select the desired value.

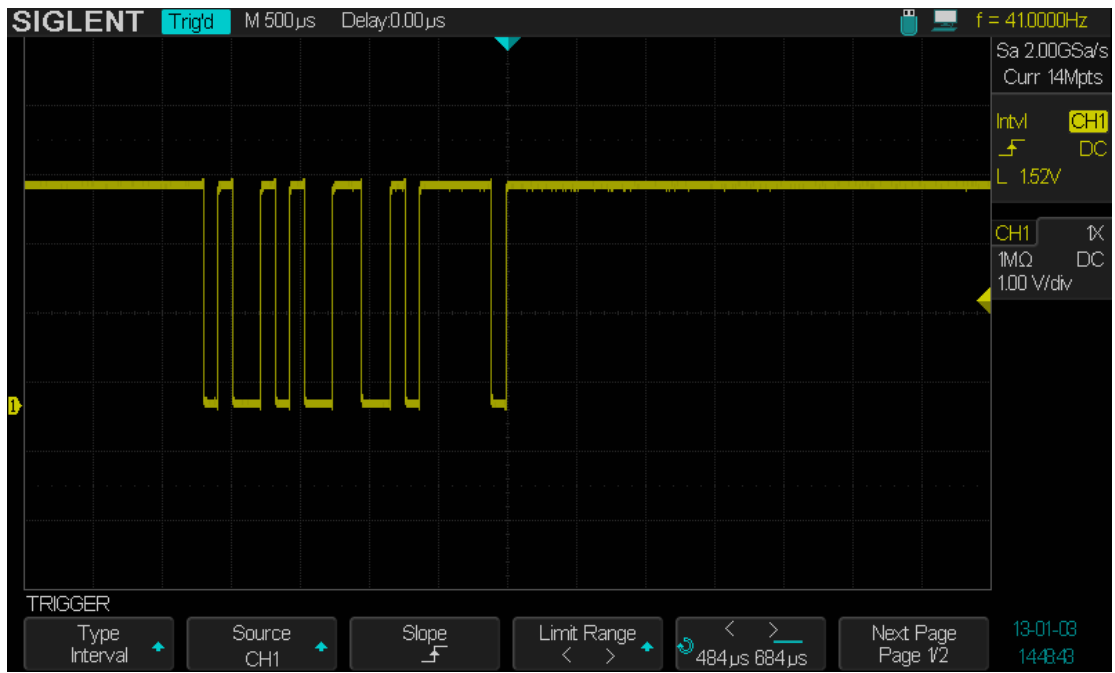


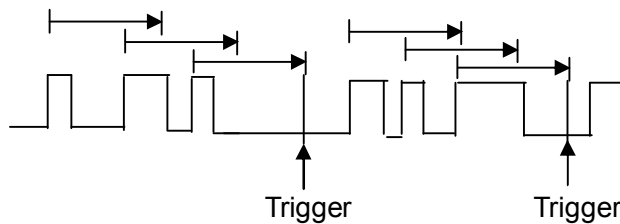
Figure 31 Interval Trigger

DropOut Trigger

DropOut trigger includes two types: edge and state.

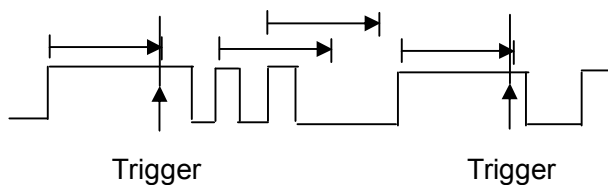
Edge

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



State

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



To set edge DropOut trigger:

1. Press the **Setup** button on the front panel to enter the TRIGGER system function menu.
2. Press the **Type** softkey; then use the **Universal Knob** to select **DropOut** and push down the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select **CH1**, **CH2**, **CH3** or **CH4** as the trigger source. The current trigger source is displayed at the upper right corner of the screen. Select channel with signal input as trigger source to obtain stable trigger.
4. Press the **Slope** softkey to select rising or falling edge.
5. Press the **OverTime Type** softkey to select **Edge**.
6. Press the **Time** softkey; turn the universal to select the desired value.

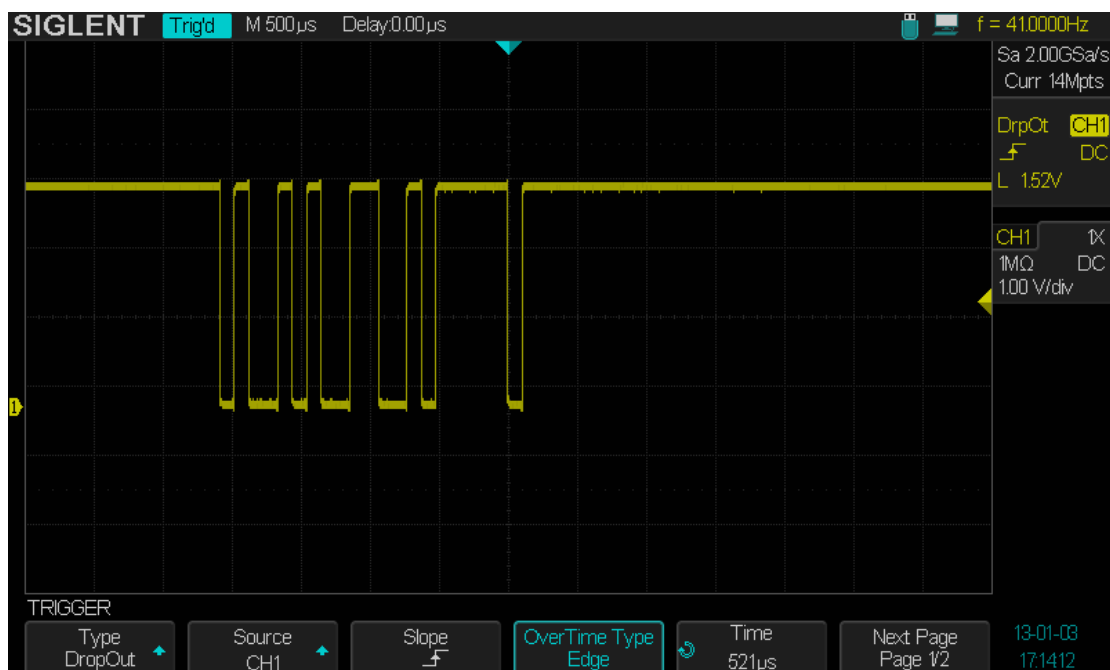


Figure 32 Edge DropOut Trigger

To set state DropOut trigger:

1. Press the **Setup** button to enter the TRIGGER system function menu.
2. Press the **Type** softkey; then turn the **Universal Knob** to select **DropOut** and push down the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select **CH1**, **CH2**, **CH3** or **CH4** as the trigger source.
4. Press the **Slope** softkey to select rising or falling edge.
5. Press the **OverTime** Type softkey to select State.
6. Press the **Time** softkey; turn the **Universal Knob** to select the desired value.

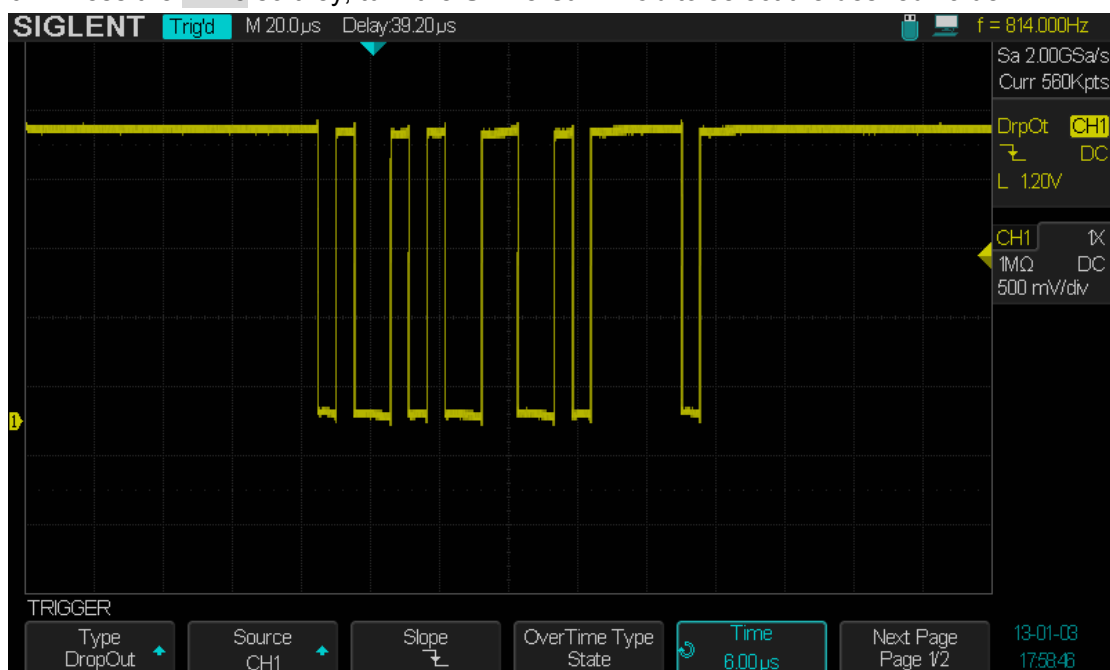
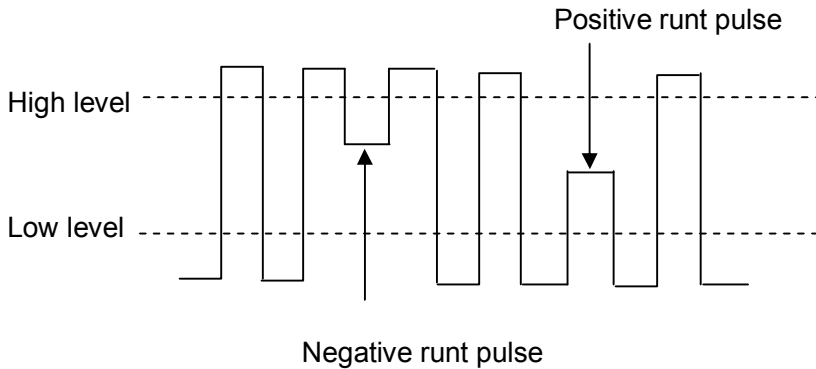


Figure 33 State DropOut Trigger

Runt Trigger

The Runt trigger looks for pulses that cross one threshold but not another as shown in the picture below.



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

To trigger on runt pulse:

1. Press the **Setup** button on the front panel to enter the **TRIGGER** system function menu.
2. Press the **Type** softkey; then turn the **Universal Knob** to select **DropOut** and push down the knob to confirm.
3. Press the **Source** softkey; turn the **Universal Knob** to select **CH1**, **CH2**, **CH3** or **CH4** as the trigger source.
4. Press the **Polarity** softkey to select **Positive** or **Negative** pulse to trigger.
5. Press the **Limit Range** softkey; turn the **Universal Knob** to select the desired condition (<, >, <> or ><).
6. Press the **Time Setting** softkey, and then turn the **Universal Knob** to select the desired value.
7. Press the **Next Page** softkey to enter the second page of the TRIGGER system function menu. Press the **Lower Upper** softkey to select **Lower** or **Upper** trigger level, and then turn the **Universal Knob** to set the position.

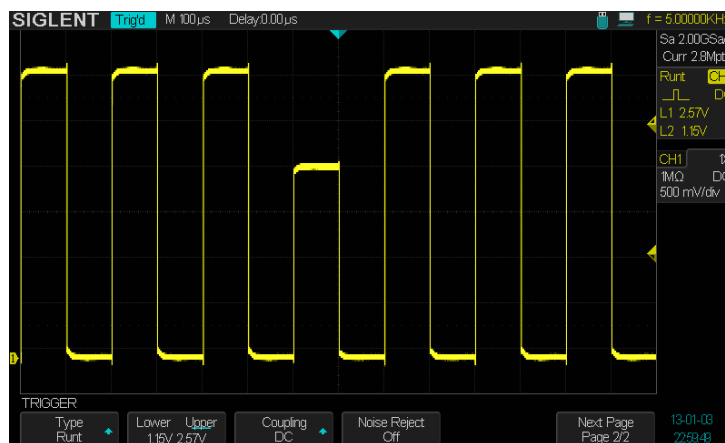
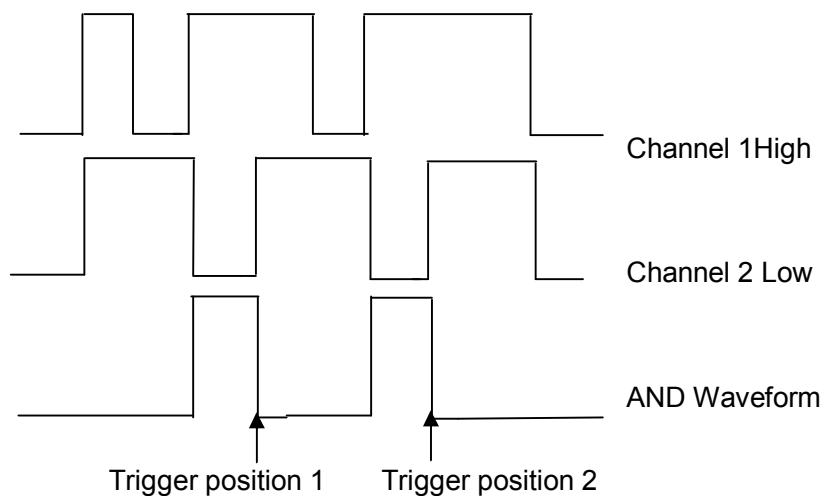


Figure 34 Runt Trigger

Pattern Trigger

The Pattern trigger identifies a trigger condition by looking for a specified pattern. The pattern trigger can be expanded to incorporate delays similar to other triggers. Pattern durations are evaluated using a timer. The timer starts on the last edge that makes the pattern “true”. Potential triggers occur on the first edge that makes the pattern false, provided that the time qualifier criterion has been met. SDS2000 provides 4 patterns: logical AND, OR, NAND and NOR combination of the channels. Each channel can set to low, high or invalid.



Do the following steps to set pattern trigger:

1. Press the **Setup** button on the front panel to enter the TRIGGER function menu.
2. Press the **Type** softkey; turn the **Universal Knob** to select Pattern, and then push down the knob to confirm.
3. Press each channel softkey to select **Invalid**, **High** or **Low**.
 - **Low** sets the pattern to low on the selected channel. A low is a voltage level that is less than the channel's trigger level or threshold level.
 - **High** sets the pattern to high on the selected channel. A high is a voltage level that is greater than the channel's trigger level or threshold level.
 - **Invalid** sets the pattern to don't care on the selected channel. Any channel set to don't care is ignored and is not used as part of the pattern. However, if all channels in the pattern are set to **Invalid**, the oscilloscope will not trigger. Adjust the trigger level for the selected analog channel by turning the Trigger Level knob. **Invalid** doesn't need to set trigger level.
4. Press the **Next Page** softkey to enter the second page of the pattern trigger menu.
5. Press the **Logic** softkey and then turn the **Universal Knob** to select the desired logic combination **AND**, **OR**, **NAND** or **NOR**.

6. Press the **Limit Range** softkey; then turn the **Universal Knob** to select the desired option.
 - **< (Less Than)** – when the pattern is present for less than a time value.
 - **> (Greater Than)** – when the pattern is present for greater than a time value. The trigger occurs
 - **>< (In Range)** – when the pattern is present for a time within a range of values.
 - **<> (Out of Range)** – when the pattern is present for a time outside of range of values.
7. Press the **Holdoff Close** softkey to turn on the Holdoff function; then turn the **Universal Knob** to select the desired value.

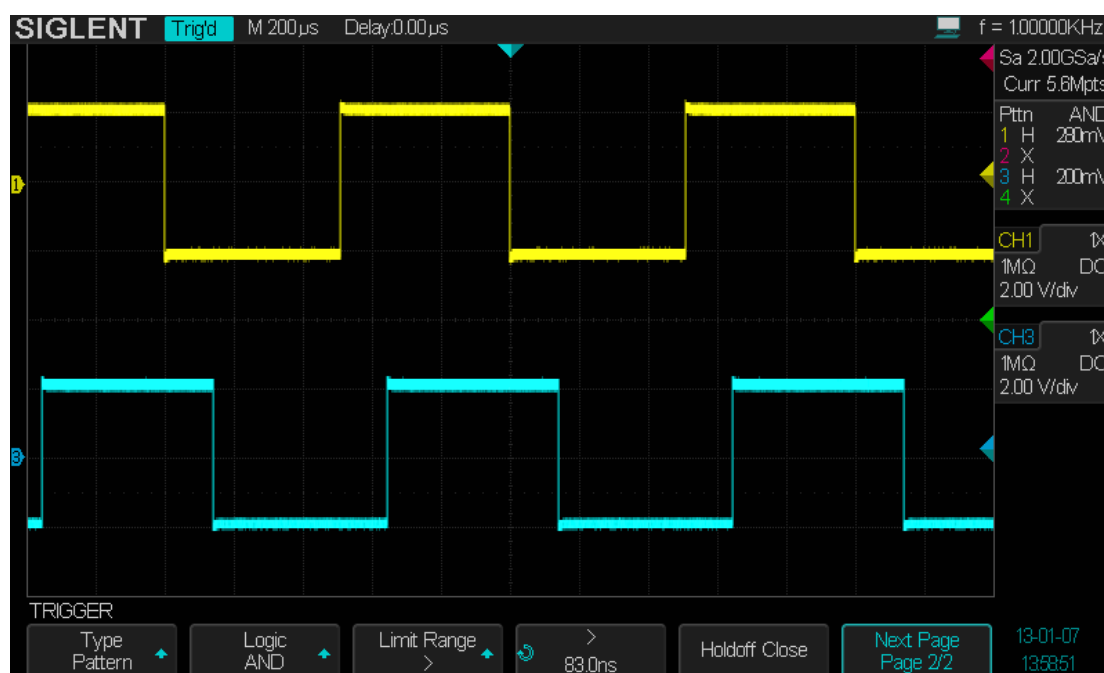


Figure 35 Pattern Trigger

Serial trigger and decode

SDS2000 provides IIC, SPI, UART/RS232, CAN and LIN serial trigger and decode. This chapter introduces how to set every kind of serial trigger and decode in details.

The contents of this chapter:

- ◆ IIC Trigger and Decoder
- ◆ SPI Trigger and Decoder
- ◆ UART/RS232 Trigger and Decoder
- ◆ CAN Trigger and Decoder
- ◆ LIN Trigger and Decoder

IIC Trigger and Serial Decode

Please take the order of “**Setup for IIC Signals**”, “**IIC Triggering**” and “**IIC Decode**” to trigger and decode the signals.

Setup for IIC Signals

IIC (Inter- IC bus) signals setup consists of connecting the oscilloscope to the serial data (SDA) line and the serial clock (SCL) line and then specifying the input signal threshold voltage levels.

To set up the oscilloscope to capture IIC signals, use the **Signal** softkey which appears in the Decode Menu:

1. Press the **Decode** button on the front panel to turn on the DECODE function menu.
2. Press the **Serial** softkey and select the desired slot (Serial 1 or Serial 2).
3. Press the **Decode** option and select IIC with the **Universal Knob**, and then push down the **Universal Knob** to confirm.



Figure 36 IIC SINGAL Menu

4. Press **Signal** softkey to enter the IIC SIGNAL menu.
 5. For both the SCL (serial clock) and SDA (serial data) signals:
 - a. Connect an oscilloscope channel to the signal in the device under test.
 - b. Press the **SCL** or **SDA** softkey; then, turn the Universal Knob to select the channel for the signal.
 - c. Press the corresponding **Threshold** softkey; then, turn the Universal Knob to set the signal threshold voltage level.
- The threshold voltage level is used in decoding, and it will become the trigger level when the trigger type is set to the selected serial decode slot.
 - Data must be stable during the whole high clock cycle or it will be interpreted as a start or stop condition (data transitioning while the clock is high).

IIC Triggering

After the oscilloscope has been set up to capture IIC signals, you can trigger on a stop/start condition, a restart, a missing acknowledge, an EEPROM data read, a read/write frame with a specific device address and data value, or on a data length.

1. Press the button **Setup** on the front panel to enter trigger function menu.
2. Press the **Type** softkey; then, turn the **Universal Knob** to select the serial slot (**Serial 1** or **Serial 2**) on which the IIC signals are being decoded above.

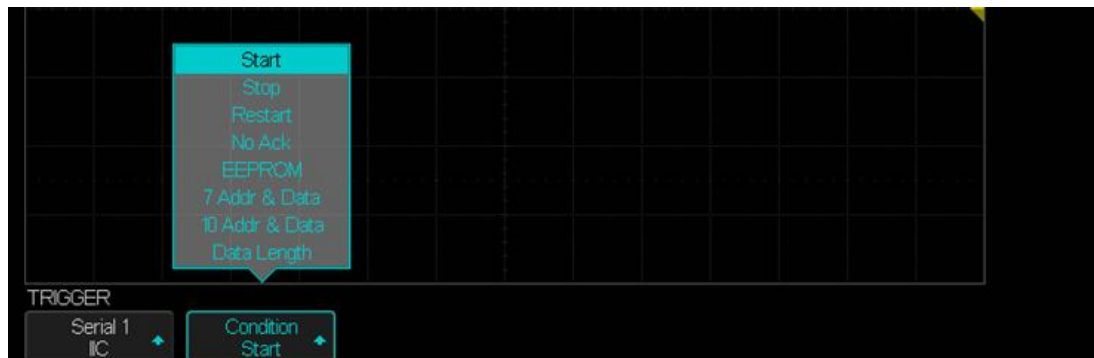
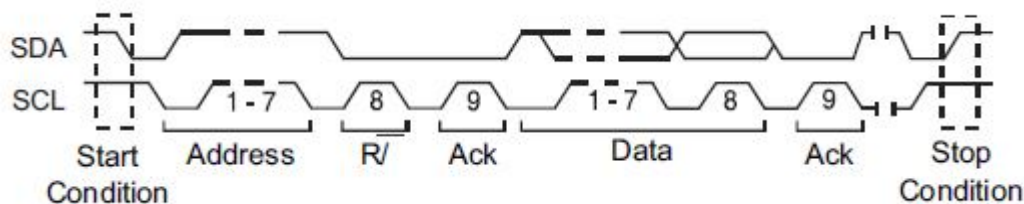
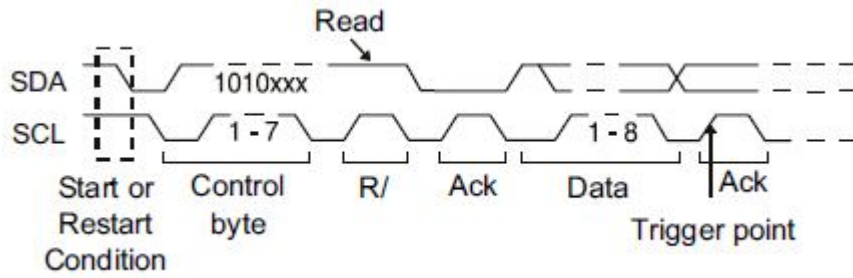


Figure 37 IIC TRIGGER Menu

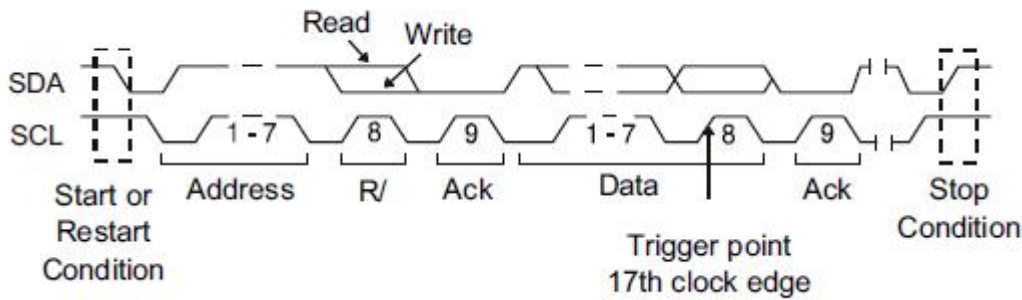
3. Press the **Condition** softkey; then, turn the Universal Knob to select the trigger condition:
 - **Start Condition**— The oscilloscope triggers when SDA data transitions from high to low while the SCL clock is high. For triggering purposes (including frame triggers), a restart is treated as a start condition.
 - **Stop Condition**— The oscilloscope triggers when data (SDA) transitions from low to high while the clock (SCL) is high.



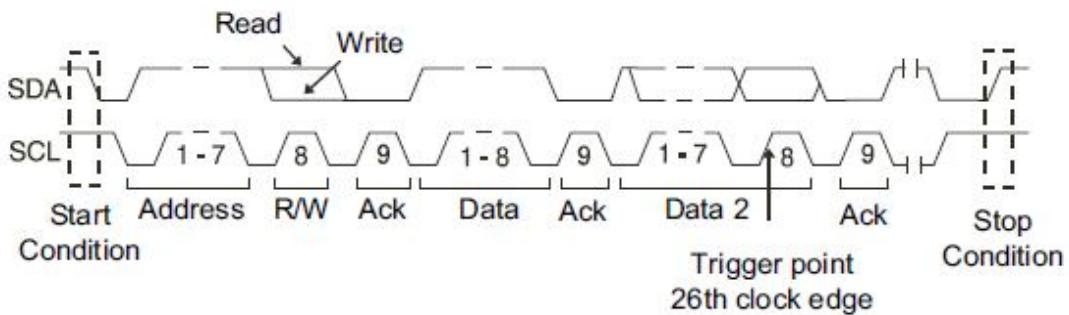
- **Restart**— The oscilloscope triggers when another start condition occurs before a **stop condition**.
- **Missing Acknowledge**— The oscilloscope triggers when SDA data is high during any Ack SCL clock bit.
- **EEPROM Data Read**— The trigger looks for EEPROM control byte value 1010xxx on the SDA line, followed by a Read bit and an Ack bit. It then looks for the data value and qualifier set by the **Limit Range** softkey and the **Data1** softkey. When this event occurs, the oscilloscope will trigger on the clock edge for the Ack bit after the data byte. This data byte does not need to occur directly after the control byte.



- 7-bit Address & Data Condition** —The oscilloscope triggers on a read or write frame in 7-bit addressing mode on the 17th or the 26th clock edge if all bits in the pattern match. For triggering purposes, a restart is treated as a start condition

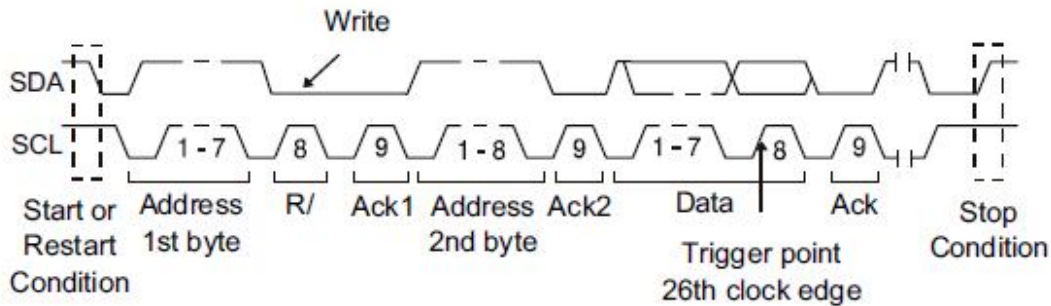


On the 17th Clock Edge



On the 26th Clock Edge

- 10-bit Address & Data Condition** — The oscilloscope triggers on a 10-bit write frame on the 26th (or 34th) clock edge if all bits in the pattern match. The frame is in the format: Frame (Start: Address byte 1: Write: Address byte 2: Ack: Data) The oscilloscope triggers on a read or write frame in 10-bit addressing mode on the 26th clock edge if all bits in the pattern match. **Data 2** shows **oxXX**. For triggering purposes, a restart is treated as a start condition.
 Frame (Start: Address byte 1: Write: Address byte 2: Ack: Data: Ack: Data) The oscilloscope triggers on a read or write frame in 7-bit addressing mode on the 34th clock edge if all bits in the pattern match. **Data2** is available. For triggering purposes, a restart is treated as a start condition



- **Data Length** — The oscilloscope triggers when the SDA data length is equal to the value set in the **Byte Length** softkey.
4. If you have set the oscilloscope to trigger on an EEPROM Data Read condition: Press the **Limit Range** softkey to set the oscilloscope to trigger when data is = (equal to), < (less than), or > (greater than) the data value set in the **Data1** softkey. The oscilloscope will trigger on the clock edge for the Ack bit after the trigger event is found. This data byte does not need to occur directly after the control byte. The oscilloscope will trigger on any data byte that meets the criteria defined by the **Byte Length** and **Data1** softkeys during a current address read or a random read or a sequential read cycle.
 5. If you have set the oscilloscope to trigger on a 7- bit address read or write frame condition or a 10- bit read or write frame condition:
 - a. Press the **Addr** softkey and turn the **Universal Knob** to select the 7- bit or 10- bit device address.
You can select from an address range of 0x00 to 0x7F (7- bit) or 0x3FF (10-bit) hexadecimal. When triggering on a read/write frame, the oscilloscope will trigger after the start, address, read/write, acknowledge, and data events occur. If don't care is selected (0xXX or 0XXX) for the address, the address will be ignored. The trigger will always occur on the 17th clock for 7- bit addressing or 26th clock for 10- bit addressing.
 - b. Press the **Data1** or **Data2** softkey and turn the **Universal Knob** to select the 8- bit data pattern on which to trigger. You can select a data value in the range of 0x00 to 0xFF (hexadecimal). The oscilloscope will trigger after the start, address, read/write, acknowledge, and data events occur.
 - c. If don't care (0xXX) is selected for data, the data will be ignored. The trigger will always occur on the 17th clock for 7- bit addressing or 26th clock for 10- bit addressing. If you have selected a three- byte trigger, press the **Data2** value softkey and turn the **Universal Knob** to select the 8- bit data pattern on which to trigger.
 6. If you have set the oscilloscope to trigger on data length condition:
 - a. Press **Address** to set the SDA address length 7bit or 10 bit.
 - b. Press **Byte Length** softkey and turn the **Universal Knob** to select the byte length.
 - c. The range of the **Byte Length** is 1 to 12.

IIC Serial Decode

To set up IIC serial decode:

1. Press **Decode** to display the DECODE function menu.



Figure 38 IIC Decode Menu

2. Press the **Address** softkey to choose 7 bit or 10 bit address size.
3. Press the **Display** softkey and choose On to display the decode line on the display.
4. Press the **List** softkey to enter the LIST function menu.
5. Press the **Display** softkey and choose On to display the decode list on the display.
6. Press the **Scroll** softkey and the **Lines** softkey to set the position of the cursor and the line number of the list with the **Universal Knob**. The range of the **Lines** is 1 to 7.

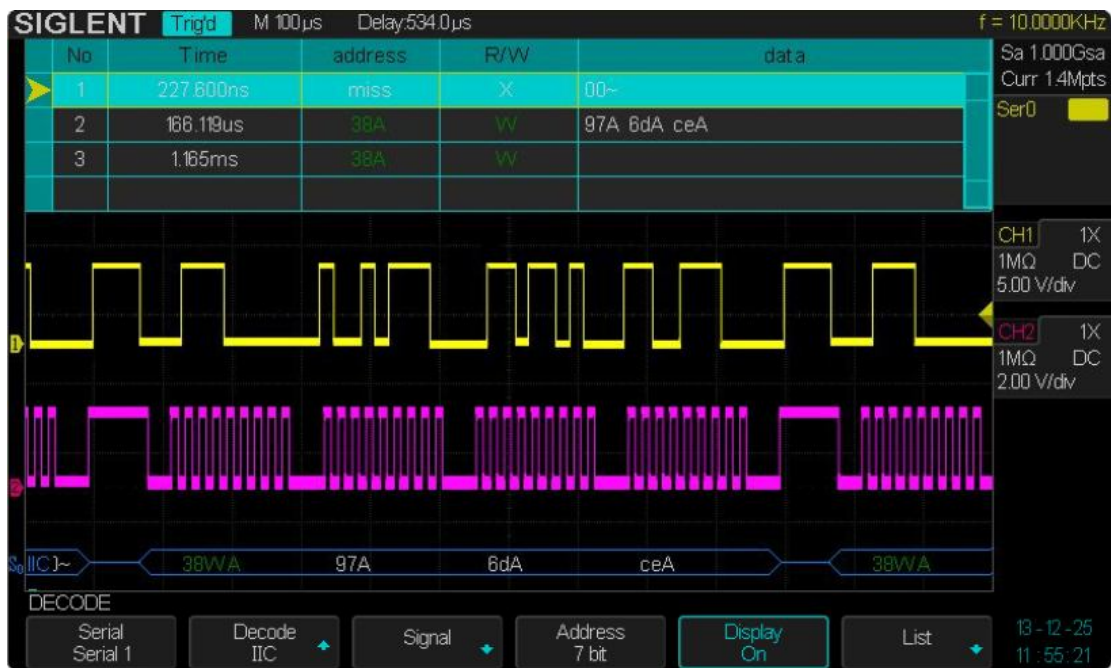


Figure 39 IIC Decode Result

Interpreting IIC Decode

- Angled waveforms show an active bus (inside a packet/frame).
- Mid-level blue lines show an idle bus.
- In the decoded hexadecimal data:
 - Address values appear at the start of a frame.
 - Write addresses appear in dark-green along with the "W" character.
 - Read addresses appear in yellow along with the "R" character.
 - Data values appear in white.
 - "A" indicates Ack (low), "~A" indicates No Ack (high).
 - Decoded text is truncated at the end of the associated frame when there is insufficient space within frame boundaries.
- Blue vertical bars indicate you need to expand the horizontal scale (and run again) to see decode.
- Red dots in the decode line indicate that more data can be displayed. Scroll or expand the horizontal scale to view the data.

Interpreting IIC Lister Data

In addition to the standard Time column, the IIC Lister contains these columns:

- NO— frame number from left to right
- Address — colored blue for writes, yellow for reads.
- R/W—yellow R for read, dark green W for write and black X for miss..
- Data — data bytes.

SPI Triggering and Serial Decode

Please take the order of “**Setup for SPI Signals**”, “**SPI Triggering**” and “**SPI Decode**” to trigger and decode the signals.

Setup for SPI Signals

Serial Peripheral Interface (SPI) signals setup consists of connecting the oscilloscope to a clock, MOSI data, MISO data, and framing signal, then setting the threshold voltage level for each input channel, and finally specifying any other signal parameters.

To set up the oscilloscope to capture SPI signals, use the **Signal** softkey which appears in the DECODE function menu:

1. Press the **Decode** button on the front panel to enter the DECODE function menu.
2. Press the **Serial** softkey and select the desired slot (Serial 1 or Serial 2).
3. Press **Decode** softkey and select **SPI** with the **Universal Knob**, and then press the knob to confirm. Serial 1 and Serial 2 could not be set to SPI at the same time. And the default setup of Serial 2 is SPI.
4. Press the **Signal** softkey to enter the SPI SIGNAL menu.



Figure 40 SPI SINGAL Menu

5. Press the **CLK** softkey to enter SPI CLK menu.



Figure 41 SPI Clock Menu

In the SPI Clock Menu:

- a. Press the **CLK** softkey; then, turn the **Universal Knob** to select the channel connected to the SPI serial clock line.
 - b. Press the **Threshold** softkey; then, turn the **Universal Knob** to select the clock signal threshold voltage level. The threshold voltage level is used in decoding, and it will become the trigger level when the trigger type is set to the selected serial decode slot.
 - c. Press the **Edge Select** softkey to select rising edge or falling edge for the selected clock source. This determines which clock edge the oscilloscope will use to latch the serial data.
6. Press the **UP** softkey to return to the SPI SINGAL menu.
 7. Press the **MISO** softkey to enter the MISO menu.



Figure 42 MOSI Menu

In the MISO menu:

- a. Press the **MISO** softkey; then, turn the **Universal Knob** to select the channel that is connected to a second SPI serial data line. (If the channel you selected is off, switch it on.)
 - b. Press the **Threshold** softkey; then, turn the **Universal Knob** to select the MISO signal threshold voltage level. The threshold voltage level is used in decoding, and it will become the trigger level when the trigger type is set to the selected serial decode slot
8. Press the **UP** softkey to return to the SPI SINGAL menu.
 9. Press the **MOSI** softkey to enter the MOSI menu.



Figure 43 MOSI Menu

In the MOSI menu:

- a. Press the **MOSI** softkey; then, turn the **Universal Knob** to select the channel that is connected to a SPI serial data line. (If the channel you selected is off, switch it on.)
 - b. Press the **Threshold** softkey; then, turn the **Universal Knob** to select the MOSI signal threshold voltage level. The threshold voltage level is used in decoding, and it will become the trigger level when the trigger type is set to the selected serial decode slot.
10. Press the **UP** softkey to return to the SPI SINGAL menu.
 11. Press the **CS** softkey to open the SPI CS menu.



Figure 44 SPI CS Menu

In the SPI CS menu:

- a. Press the **Cs Type** softkey to select a framing signal that the oscilloscope will use for determining which clock edge is the first clock edge in the serial stream. You can set the oscilloscope to trigger during a high chip select (**CS**), a low chip select (**~CS**), or after a **Timeout** period during which the clock signal has been idle.
 - If the framing signal is set to **CS** (or **~CS**), the first clock edge as defined, rising or falling, seen after the **CS** (or **~CS**) signal transitions from low to high (or high to low) is the first clock in the serial stream.
 - Press the **CS** or **~CS** softkey; then, turn the **Universal Knob** to select the channel that is connected to the SPI frame line. The label (**~CS** or **CS**) for the source channel is automatically set. The data pattern and the clock transition must occur during the time when the framing signal is valid. The framing signal must be valid for the entire data pattern.
 - If the framing signal is set to **Timeout**, the oscilloscope generates its own internal framing signal after it sees inactivity on the serial clock line.
 - **CLK Timeout** — Select **Clock Timeout** in the **Cs Type** softkey, then select the **Limit** softkey and turn the **Universal Knob** to set the minimum time that

the Clock signal must be idle (not transitioning) before the oscilloscope will search for the Data pattern on which to trigger. The **Limit** value can be set anywhere from 100 ns to 1 s.

- b. Press the **Threshold** softkey; then, turn the **Universal Knob** to select the chip select signal threshold voltage level. The threshold voltage level is used in decoding, and it will become the trigger level when the trigger type is set to the selected serial decode slot.
12. Press the **Idle Level** softkey to set idle level to **High** or **Low** according to the input signal.
 13. Press the **Bit Order** softkey to set the bit order to **LSB** or **MSB** according to the input signal.

SPI Triggering

After the oscilloscope has been set up to capture SPI signals, you can then trigger on a data pattern that occurs at the start of a frame. The serial data string can be specified to be from 4 to 96 bits long.

1. Press **Setup** button on the front panel to enter the TRIGGER function menu.
2. Press the **Type** softkey; then, turn the **Universal Knob** to select the serial slot (**Serial 1** or **Serial 2**) on which the SPI signals are being decoded above.
3. Press the **Trigger Setting** softkey to enter the SPI TRIG SET menu.



Figure 45 SPI TRIG SET Menu

4. Press the **Trigger Type** softkey to select the trigger condition.
 - **MISO DATA (Master-In, Slave-Out)** — for triggering on the MISO data signal.
 - **MOSI DATA (Master-Out, Slave-In)** — for triggering on the MOSI data signal.
5. Press the **Data Length** softkey, and turn the **Universal Knob** to set the number of bits in the serial data string. The number of bits in the string can be set anywhere from 4 bits to 96 bits. The data values for the serial string are displayed in the MISO/ MOSI data string in the waveform area.
6. For each bit in the MISO/ MOSI data string:
 - a. Press the **Bit Roll** softkey; then, turn the **Universal Knob** to select the bit location. As you rotate the Entry knob, the bit is highlighted in the Data string shown in the waveform area.
 - b. Press the **Bit Value** softkey to set the bit selected in the **Bit Roll** softkey to **0** (low), **1** (high), or **X** (don't care).
7. Press the **All Same** softkey will set all bits in the data string to **0** (low), **1** (high), or **X** (don't care) at once.
8. Press the **Bit Order** softkey to set the bit order to **LSB** (Least significant bit first) or **MSB** (Most significant bit first) which used when displaying data in the serial decode waveform and in the Lister.



Figure 46 SPI Trigger

SPI Serial Decode

To set up SPI serial decode:

1. Press **Decode** to display the DECODE function menu.



Figure 47 SPI Decode function menu

2. Press the **Data Length** softkey, then, turn the **Universal Knob** to set the bit number of the SPI decoded data.
3. Press the **Display** softkey and choose On to display the decode line on the display.
4. Press the **List** softkey to enter the LIST function menu.
5. Press the **Display** softkey and choose **On** to display the decode list on the display.
6. Press the **Scroll** softkey and the **Lines** softkey to set the position of the cursor and the line number of the list with the **Universal Knob**. The range of the **Lines** is 1 to 7.

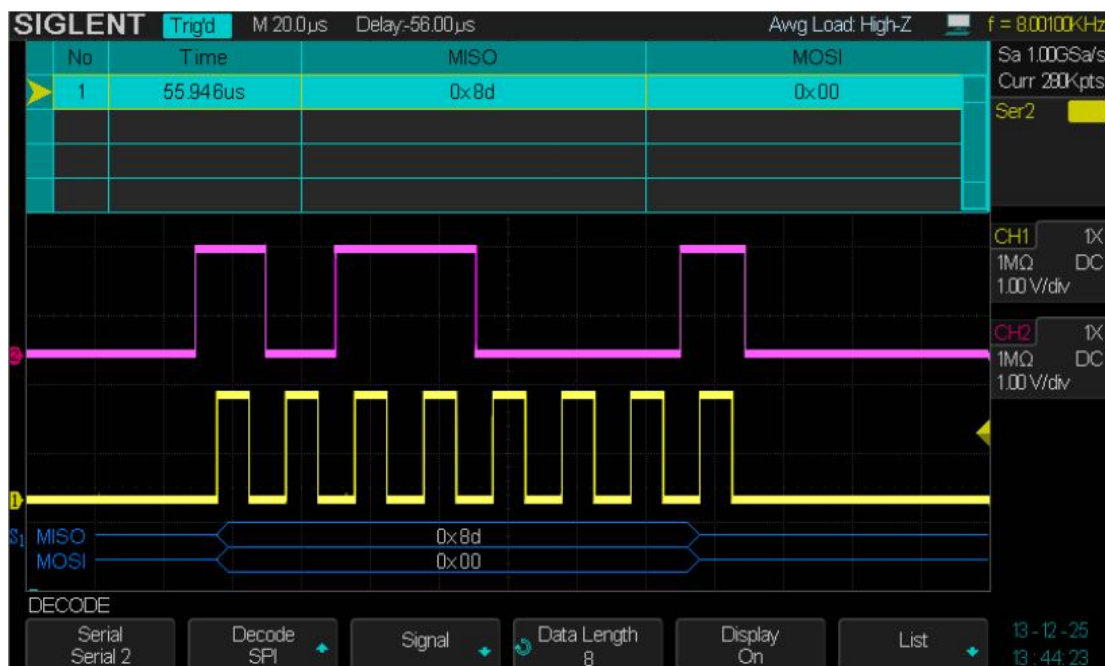


Figure 48 SPI Decode

Interpreting SPI Decode

- Angled waveforms show an active bus (inside a packet/frame).
- Mid-level blue lines show an idle bus.
- The number of clocks in a frame appears in light-blue above the frame, to the right.
- Decoded hexadecimal data values appear in white.
- Decoded text is truncated at the end of the associated frame when there is insufficient space within frame boundaries.
- Pink vertical bars indicate you need to expand the horizontal scale (and run again) to see decode.
- Red dots in the decode line indicate that there is data that is not being displayed. Scroll or expand the horizontal scale to view the information.
- Aliased bus values (undersampled or indeterminate) are drawn in pink.
- Unknown bus values (undefined or error conditions) are drawn in red.

Interpreting SPI Lister Data

In addition to the standard Time column, the SPI Lister contains these columns:

- NO— frame number from left to right.
- MISO — Data for MISO decode.
- MOSI — Data for MOSI decode.

UART/RS232 Triggering and Serial Decode

Please take the order of “**Setup for UART/RS232 Signals**”, “**UART/RS232 Triggering**” and “**UART/RS232 Decode**” to trigger and decode the signals.

Setup for UART/RS232 Signals

To set up the oscilloscope to capture UART/RS232 signals:

1. Press **Decode** on the front panel to turn on the DECODE function menu.
2. Press **Serial** and select the desired slot (Serial 1 or Serial 2).
3. Press **Decode** option and select **UART/RS232** with the **Universal Knob**, and then push down **Universal Knob** to confirm.
4. Press the **Signal** softkey to enter the UART/RS232 SIGNAL menu.



Figure 49 UART/RS232 SIGNAL Menu

5. For both the Rx and Tx signals:
 - a. Connect an oscilloscope channel to the signal in the device under test.
 - b. Press the **Rx** or **Tx** softkey to select the channel for the signal.
 - c. Press the corresponding **Threshold** softkey; then, turn the **Universal Knob** to select the signal threshold voltage level. The threshold voltage level is used in decoding, and it will become the trigger level when the trigger type is set to the selected serial decode slot.
6. Press the **UP** softkey to return to the DECODE menu.
7. Press the **Configure** softkey to open the UART/RS232 BUS CONFIG menu. Set the following parameters.



Figure 50 BUS CONFIG Menu

- **Baud** — Press the **Baud** softkey, then press the **Universal Knob** and select a baud rate to match the signal in your device under test. If the desired baud rate is not listed, select **Custom** on the **Baud** softkey, and then press the **Custom** softkey and turn the **Universal Knob** to set the desired baud rate. You can set the UART baud rate to 115200b/s at most.
- **Parity Check**— Choose odd, even, or none, based on your device under test.
- **Stop Bit**—Set the number of stop bits in the UART/RS232 words to match your device under test.
- **Data Length**— Set the number of bits in the UART/RS232 words to match your device under test (selectable from 5- 8 bits).

UART/RS232 Triggering

To trigger on a UART (Universal Asynchronous Receiver/Transmitter) signal connect the oscilloscope to the Rx and Tx lines and set up a trigger condition. RS232 (Recommended Standard 232) is one example of a UART protocol.

1. Press **Setup** button on the front panel to enter the TRIGGER function menu.
2. Press the **Type** softkey; then, turn the **Universal Knob** to select the serial slot (**Serial 1** or **Serial 2**) on which the UART/RS232 signals are being decoded above.
3. Press the **Trigger** Setting softkey to enter the UART TRIG SET menu.



Figure 51 UART TRIG SET Menu

4. Press the **Source Type** softkey to select TXDATA or RX DATA.
5. Press the **Trigger** softkey and set up the desired trigger condition:
 - **Start** — The oscilloscope triggers when a start bit occurs.
 - **Stop** — Triggers when a stop bit occurs on Rx. vice under test uses 1, 1.5, •
 - **Data** — Triggers on a data byte that you specify. For use when the device under test data words are from 5 to 8 bits in length
 - a. Press the **Compare Type** softkey, and choose an equality qualifier. You can choose equal to, less than, or greater than a specific data value.
 - b. Use the **Value** softkey to choose the data value for your trigger comparison. This works in conjunction with the **Compare Type** softkey. The range of the Value is 0x00 to 0xff.

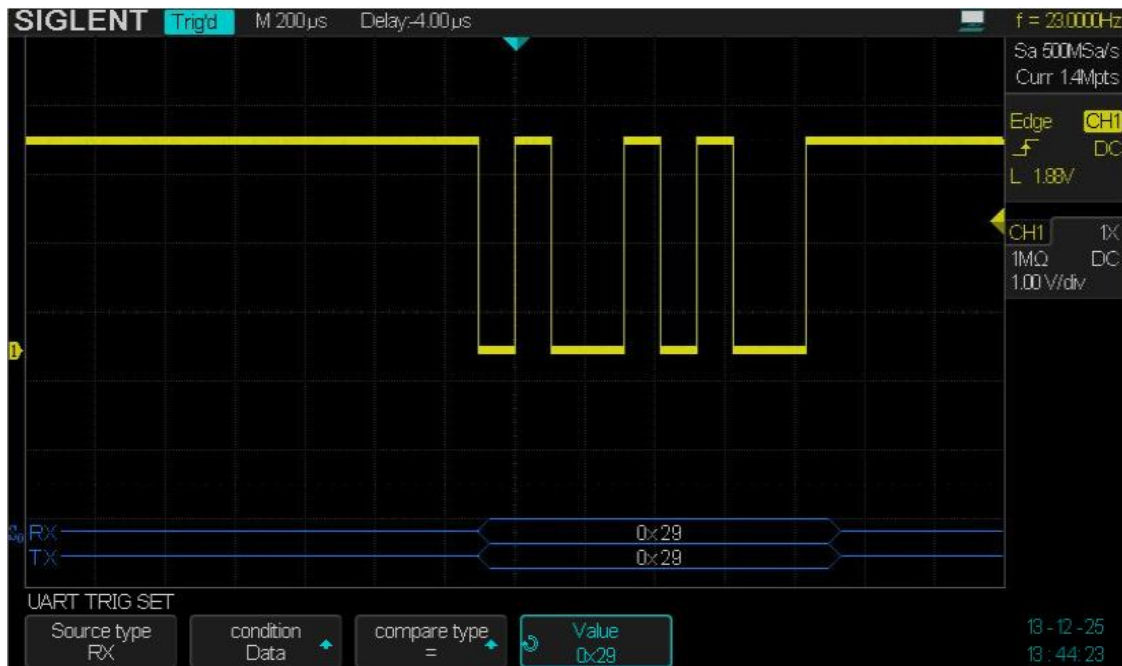


Figure 52 UART/RS232

UART/RS232 Serial Decode

To set up UART/RS232 serial decode:

1. Press **Decode** button on the front panel to display the DECODE function menu.



Figure 53 UART/RE232 DECODE Function Menu

2. Press the **Display** softkey and choose On to display the decode line on the display.
3. Press the **List** softkey to enter the LIST function menu.
4. Press the **Display** softkey and choose On to display the decode list on the display.
5. Press the **Scroll** softkey and the **Lines** softkey to set the position of the cursor and the line number of the list with the **Universal Knob**. The range of the **Lines** is 1 to 7.



Figure 54 UART/RE232 Decode

Interpreting UART/RS232 Decode

- Angled waveforms show an active bus (inside a packet/frame).
- Mid-level blue lines show an idle bus.
- Mid-level red lines show that the idle level is wrong.
- The decoded data is displayed in white.
- Decoded text is truncated at the end of the associated frame when there is insufficient space within frame boundaries.
- Blue vertical bars indicate you need to expand the horizontal scale (and run again) to see decode.
- When the horizontal scale setting does not permit the display of all available decoded data, red dots will appear in the decoded bus to mark the location of hidden data. Expand the horizontal scale to allow the data to display.
- An unknown (undefined) bus is shown in red.

Interpreting UART/RS232 Lister Data

In addition to the standard Time column, the UART/RS232 Lister contains these columns:

- Rx — Receive data.
- Tx — Transmit data.
- Rx err— Parity error or unknown error when receiving data.
- Tx err— Parity error or unknown error when transmit data.

CAN Trigger and Serial Decode

Please take the order of “**Setup for CAN Signals**”, “**CAN Triggering**” and “**CAN Decode**” to trigger and decode the signals.

Setup for CAN Signals

Setup consists of connecting the oscilloscope to a CAN signal, using the SIGNAL menu to specify the signal source, threshold voltage level, baud rate, and sample point.

To set up the oscilloscope to capture CAN signals.

1. Press **Decode** on the front panel to enter the DECODE function menu.
2. Press **Serial** and select the desired slot (Serial 1 or Serial 2).
3. Press **Decode** softkey and select CAN with the **Universal Knob**, and then push down the **Universal Knob** to confirm.
4. Press the **Signal** softkey to enter the CAN SIGNAL menu.



Figure 55 CAN SIGNAL Menu

5. Press **CAN-H** or **CAN-L**; then, select the channel for the CAN signal.
6. Press the **Threshold** softkey; then, turn the **Universal Knob** to select the CAN signal threshold voltage level. The threshold voltage level is used in decoding, and it will become the trigger level when the trigger type is set to the selected serial decode slot.
7. Press the **Back** key to return to the DECODE menu.
8. Press the **Configure** softkey to enter the BUS CONFIG menu.
9. Press the **Baud** softkey; then, turn the **Universal Knob** to select the baud rate that matches your CAN bus signal. The CAN baud rate can be set to predefined baud rates from 5 kb/s up to 1 Mb/s or a custom baud rate from 1b/s to 1 Mb/s. If the desired baud rate is not listed, select **Custom** on the **Baud** softkey, and then press the **Custom** softkey and turn the **Universal Knob** to set the desired baud rate.
10. Press the **Decode Source** softkey to set the decode source that matches the single®
 - **CAN_H** — The actual CAN_H differential bus.
 - **CAN_L** — The actual CAN_L differential bus signal.
 - **CAN_H-CAN_L** — The CAN differential bus signals connected to an analog source channel using a differential probe. Connect the probe's positive lead to the dominant- high CAN signal (CAN_H) and connect the negative lead to the dominant- low CAN signal (CAN_L).

CAN Triggering

After setting up the oscilloscope to capture a CAN signal:

1. Press **Setup** button the front panel to enter the TRIGGER function menu.
2. Press the **Type** softkey; then, turn the **Universal Knob** to select the serial slot (**Serial 1** or **Serial 2**) on which the CAN signals are being decoded above.
3. Press the **Trigger Setting** softkey to enter the CAN TRIG SET menu.
4. Press the **Trigger** softkey; then, turn the **Universal Knob** to select the trigger condition:
 - **Start**— The oscilloscope triggers at the start of a frame.
 - **Remote** — The oscilloscope triggers on remote frames with the specified ID.
 - a. Press the **ID Bits** softkey to select the ID number 11bits or 29 bits.
 - b. Press the **Curr ID Byte** softkey and turn the universal to select the byte to be set.
 - c. Press the **ID** softkey and then turn the universal to set the ID
 - **ID** — The oscilloscope will trigger on remote or data frames matching the specified ID
 - a. Press the **ID Bits** softkey to select the ID number 11bits or 29 bits.
 - b. Press the **Curr ID Byte** softkey and turn the universal to select the byte to be set.
 - c. Press the **ID** softkey and then turn the universal to set the ID
 - **ID+DATA**—The oscilloscope will trigger on data frames matching the specified ID and data.
 - a. Press the **ID Bits** softkey to select the ID number 11bits or 29 bits.
 - b. Press the **Curr ID Byte** softkey and turn the **Universal Knob** to select the byte to be set.
 - c. Press the **ID** softkey and then turn the **Universal Knob** to set the ID
 - d. Press the **Data1** softkey and then turn the **Universal Knob** to set the value of the first byte.
 - e. Press the **Data1** softkey and then turn the **Universal Knob** to set the value of the first byte.
 - **Error**—The oscilloscope will trigger when any form error or active error is encountered

The picture below triggers on ID, the ID is 14b2d4ff and the Band rate is 100kb/s:

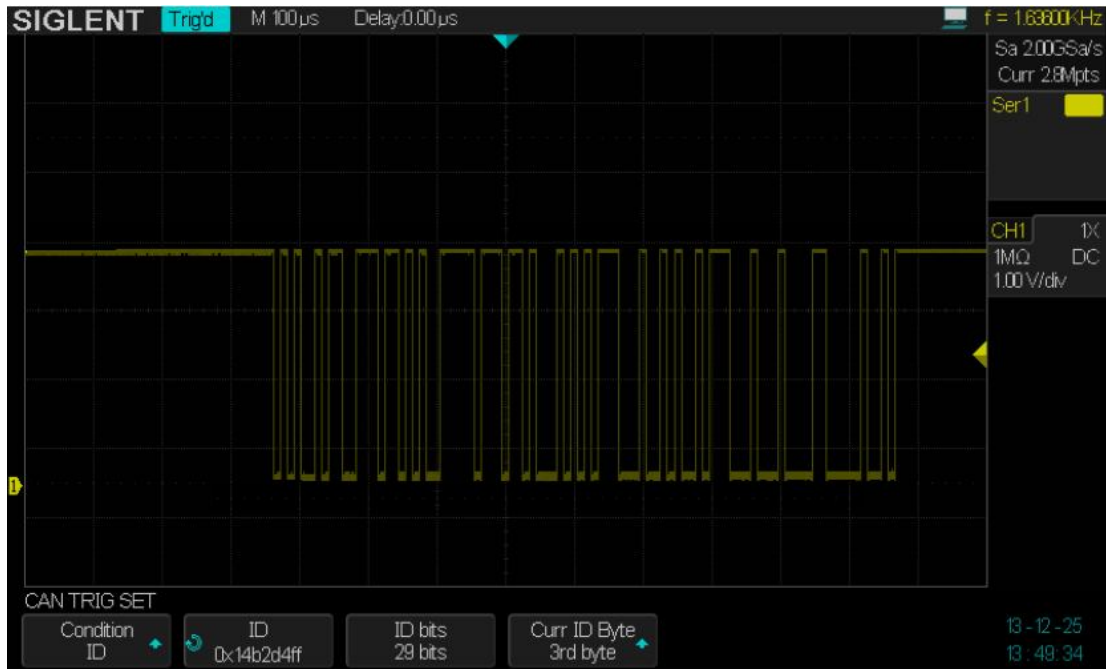


Figure 56 CAN Trigger

CAN Serial Decode

To set up CAN serial decode:

1. Press **Decode** to display the DECODE function menu.
2. Press the **Display** softkey and choose On to display the decode line on the display.
3. Press the **List** softkey to enter the LIST function menu.
4. Press the **Display** softkey and choose On to display the decode list on the display.
5. Press the **Scroll** softkey and the **Lines** softkey to set the position of the cursor and the line number of the list with the **Universal Knob**. The range of the **Lines** is 1 to 7.

The picture below triggers on ID, the ID is 14b2d4ff and the Band rate is 100kb/s:

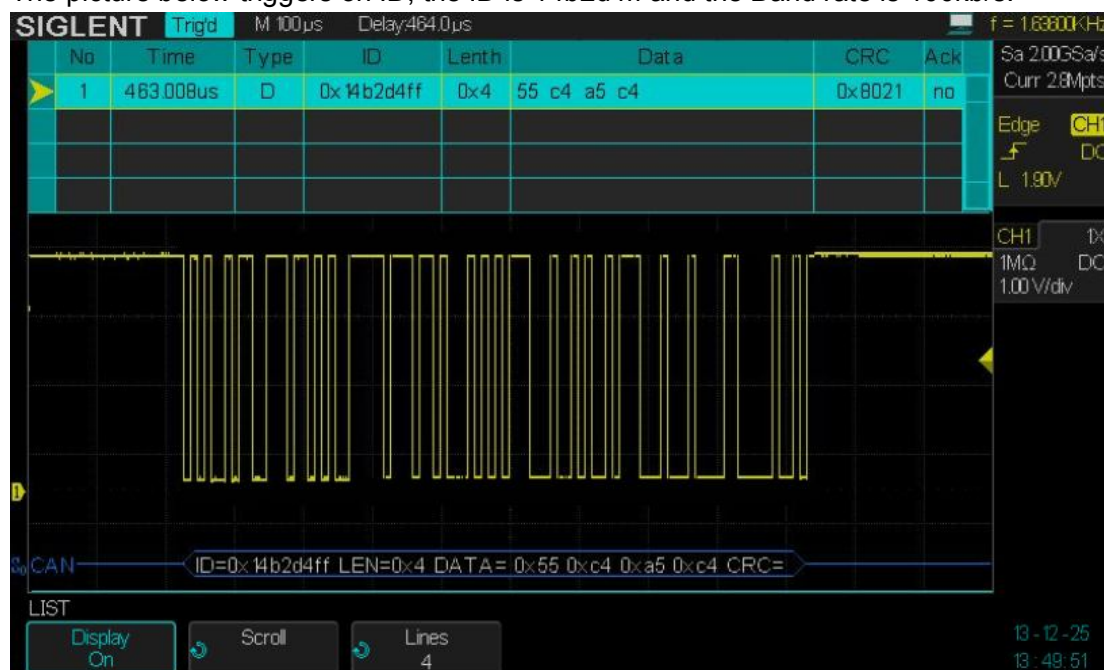


Figure 57 CAN Decode

Interpreting CAN Decode.

- Data bytes appear in hex digits in white
- Cyclic redundancy check (CRC) appears in hex digits in blue when valid, or in red to indicate that the oscilloscope's hardware decode calculated a CRC that is different from the incoming CRC data stream.
- Angled waveforms show an active bus (inside a packet/frame).
- Mid-level blue lines show an idle bus.
- Decoded text is truncated at the end of the associated frame when there is insufficient space within frame boundaries.
- Pink vertical bars indicate you need to expand the horizontal scale (and run again) to see decode.
- Red dots in the decode line indicate that there is data that is not being displayed. Scroll or expand the horizontal scale to view the information.
- Aliased bus values (undersampled or indeterminate) are drawn in pink.

Interpreting CAN Lister Data

In addition to the standard Time column, the CAN Lister contains these columns:

- NO— frame number from left to right.
- Type —R indicates remote frame and D indicates data frame.
- ID — frame ID.
- Length — Data length code.
- Data— Data for CAN decode.
- CRC — Cyclic redundancy check.
- Ack— Indicates that it has Ack.

LIN Triggering and Serial Decode

Please take the order of “**Setup for LIN Signals**”, “**LIN Triggering**” and “**LIN Decode**” to trigger and decode the signals.

Setup for LIN Signals

LIN (Local Interconnect Network) signal setup consists of connecting the oscilloscope to a serial LIN signal, specifying the signal source, threshold voltage level, baud rate, sample point, and other LIN signal parameters.

To set up the oscilloscope to capture LIN signals:

1. Press **Decode** on the front panel to turn on the DECODE function menu.
2. Press **Serial** and select the desired slot (Serial 1 or Serial 2).
3. Press **Decode** softkey and select **LIN** with the **Universal Knob**, and then push down the **Universal Knob** to confirm.
4. Press the **Signal** softkey to enter the LIN SIGNAL menu.



Figure 58 LIN SIGNAL Menu

5. Press **Sources** softkey; then, select the channel for the LIN signal.
6. Press the **Threshold** softkey; then, turn the **Universal Knob** to select the LIN signal threshold voltage level. The threshold voltage level is used in decoding, and it will become the trigger level when the trigger type is set to the selected serial decode slot.
7. Press the **UP** key to return to the DECODE menu.
8. Press the **Configure** softkey to enter the BUS CONFIG menu.
9. Press the **Baud** softkey; then, turn the **Universal Knob** to select the baud rate that matches your LIN bus signal. The LIN baud rate can be set to predefined baud rates from 600b/s to 19200b/s or a custom baud rate from 1b/s to 20 Kb/s. If the desired baud rate is not listed, select **Custom** on the **Baud** softkey, and then press the **Custom** softkey and turn the **Universal Knob** to set the desired baud rate.

LIN Triggering

LIN triggering can trigger on the rising edge at the Sync Break exit of the LIN single-wire bus signal (that marks the beginning of the message frame), the Frame ID, or the Frame ID and Data.

A LIN signal message frame is shown below:

1. Press **Setup** to enter trigger function menu.
2. Press the **Type** softkey; then, turn the **Universal Knob** to select the serial slot (**Serial 1** or **Serial 2**) on which the LIN signals are being decoded above.
3. Press the Trigger Setting softkey to enter the LIN TRIG SET menu.
4. Press the **Trigger Setting** softkey; then, turn the **Universal Knob** to select the trigger condition:
 - **Start** — The oscilloscope triggers when a start bit occurs
 - **ID (Frame ID)** — The oscilloscope triggers when a frame with an ID equal to the selected value is detected. Use the **Universal Knob** to select the value for the Frame ID.
 - **ID + Data** (Frame ID and Data) — The oscilloscope triggers when a frame with an ID and data equal to the selected values is detected. Use the **Universal Knob** to select the value for the ID, Data1 and Data2.
 - **Data Error** —The oscilloscope triggers when detects data error.

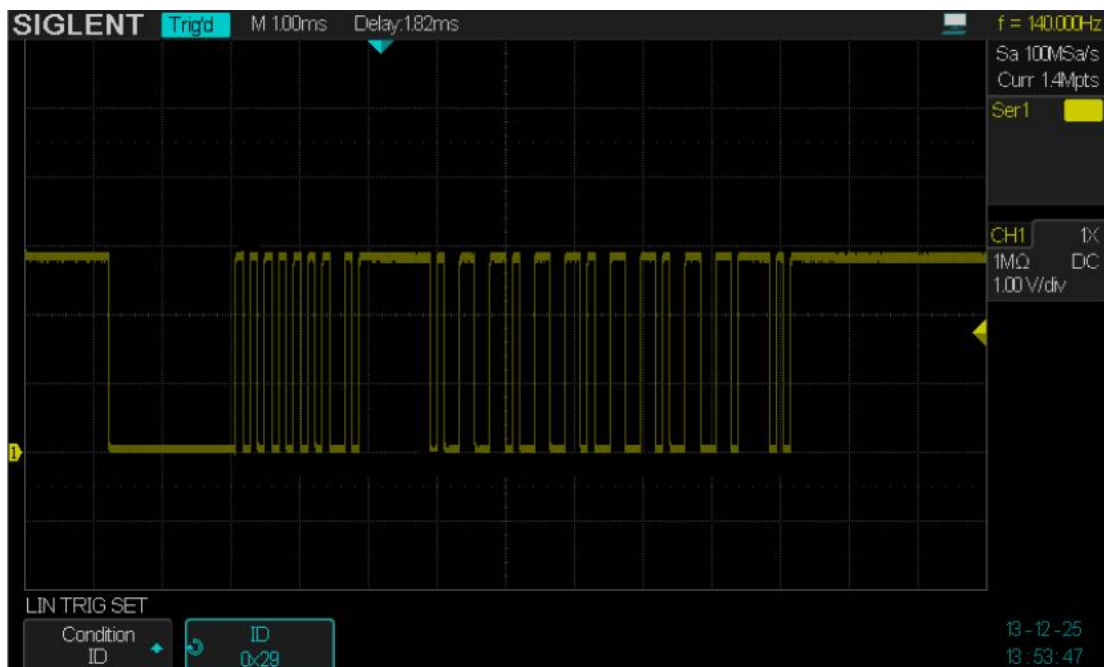


Figure 59 LIN Trigger

LIN Serial Decode

To set up LIN serial decode:

1. Press **Decode** button on the front panel to display the DECODE function menu.

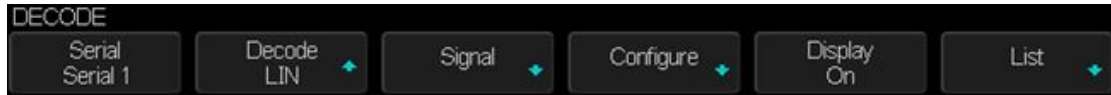


Figure 60 LIN DECODE Function Menu

2. Press the **Display** softkey and choose **On** to display the decode line on the display.
3. Press the **List** softkey to enter the LIST function menu.
4. Press the **Display** softkey and choose **On** to display the decode list on the display.
5. Press the **Scroll** softkey and the **Lines** softkey to set the position of the cursor and the line number of the list with the **Universal Knob**. The range of the **Lines** is 1 to 7.

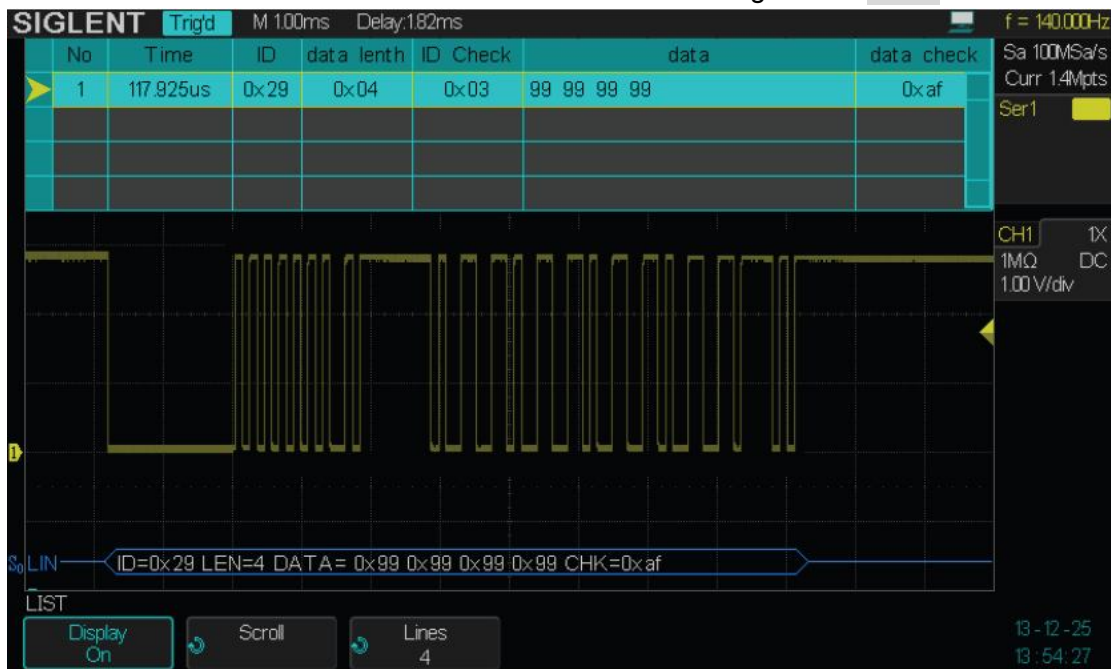


Figure 61 LIN Decode

Interpreting LIN Decode

- Angled waveforms show an active bus (inside a packet/frame).
- Mid-level blue lines show an idle bus.
- The hexadecimal ID and parity bits (if enabled) appear in yellow. If a parity error is detected the hexadecimal ID and parity bits (if enabled) appear in red.
- Decoded hexadecimal data values appear in white.
- Decoded text is truncated at the end of the associated frame when there is insufficient space within frame boundaries.
- Pink vertical bars indicate you need to expand the horizontal scale (and run again) to see decode.
- Red dots in the decode line indicate that there is data that is not being displayed. Scroll or expand the horizontal scale to view the information.
- Unknown bus values (undefined or error conditions) are drawn in red.

Interpreting LIN Lister Data

In addition to the standard Time column, the LIN Lister contains these columns:

- ID — Frame ID.
- Data — Data bytes.
- Data length —Length of data.
- ID Check— ID parity error.
- Data Check— Data error.

Digital Channels

This chapter describes how to use the digital channels of a Mixed-Single Oscilloscope (MSO).

The digital channels are enabled on SDS2000 serials that have installed the MSO license.

This chapter mainly contains the following items:

- ◆ To Connect the Digital Probes to the Device under Test
- ◆ Acquiring Waveform Using the Digital Channels
- ◆ To Change the Display Type of the Digital Channels
- ◆ To Switch a Single Channel On or Off
- ◆ To Switch All Digital Channels On or Off
- ◆ To Change the Logic Threshold for Digital Channels
- ◆ To Reposition a Digital Channel
- ◆ To Display Digital Channels as a Bus

To Connect the Digital Probes to the Device under Test

1. If necessary, turn off the power supply to the device under test.
 - Turning off the power to the device under test would only damage that might occur if you accidentally short lines together while connecting probes. You can leave the oscilloscope powered on because no voltage appears at the probes.
2. Connect one of the digital probe cable ports to the digital channels connector on the front panel of the SDS2000, and the other one to the digital channels connector of the digital probe.
 - The digital probe cable is keyed so you can connect it only one way. You do not need to power-off the oscilloscope.
 - The two ports of the digital probe cable are the same that you can connect them using the way you like.
3. Connect a fly line to one of the digital probe ground pin, connect a grabber to the fly line, and then connect the grabber to ground in the device under test. The ground lead improves signal fidelity to the oscilloscope, ensuring accurate measurements,
4. Connect a fly line to one of the digital probe digital channel pins; connect a grabber to the fly line, and then connect the grabber to a node in the circuit you want to test.
5. Repeat step 4 until you have connected all points of interest.

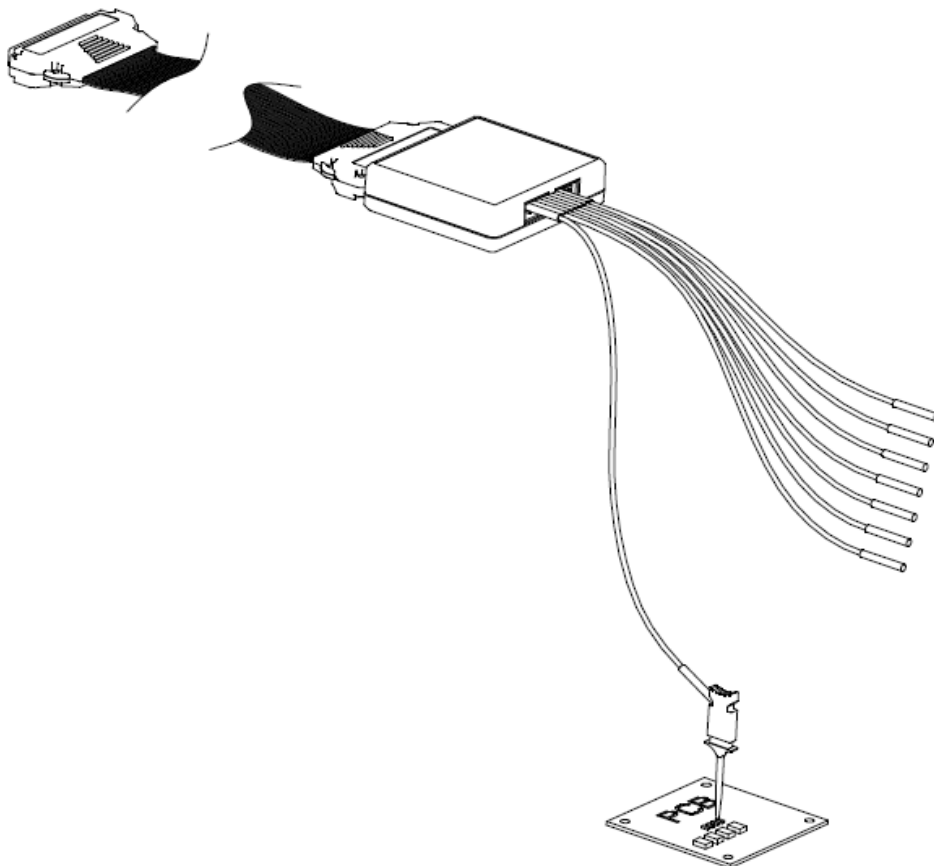


Figure 62 Connect the Digital Probes

Acquiring Waveform Using the Digital Channels

Press the **Digital** button on the front panel to open the digital channels and start acquiring digital channel waveforms.

For digital channels, each time the oscilloscope takes a sample it compares the input voltage to the logic threshold. If the voltage is above the threshold, the oscilloscope stores a 1 in the sample memory; otherwise, it stores a 0.

To Change the Display Type of the Digital Channels

1. Press the **Digital** button on the front panel to open the DIGITAL function menu.



Figure 63 Digital Function Menu

2. Press the **Channel High** softkey to select **Low** or **High** display type.

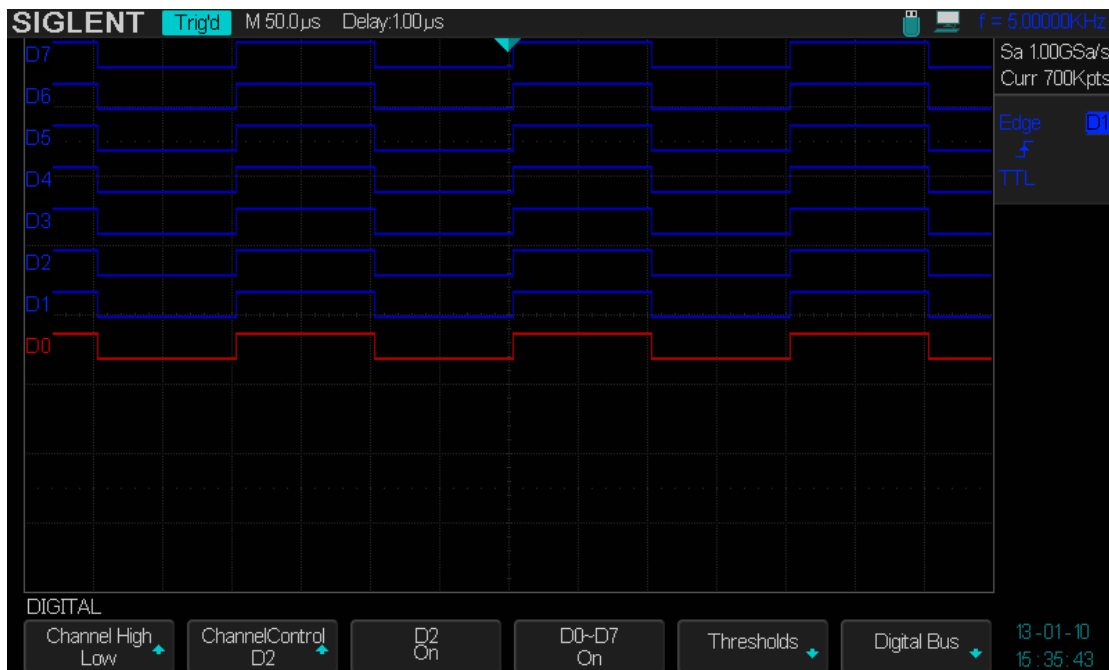


Figure 64 Low Display Type

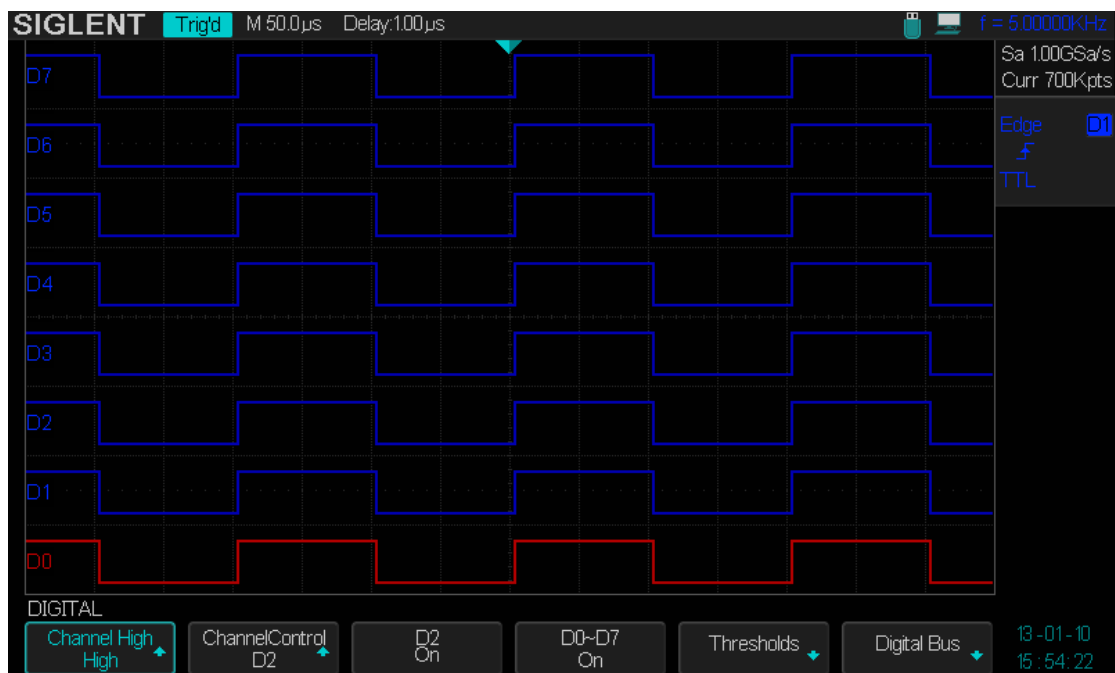


Figure 65 High Display Type

The display type control lets you spread out or compress the digital traces vertically on the display for more convenient viewing.

To Switch a Single Channel On or Off

1. Press the **Digital** button on the front panel to open the DIGITAL function menu.
2. Press the **ChannelControl** softkey, then turn the Universal Knob to select the desired channel such as **Dx** and push down the knob to confirm.
3. Press the **Dx** softkey to select **On** or **Off** to switch the selected channel on or off.

To Switch All Digital Channels On or Off

1. Press the **Digital** button on the front panel to open the DIGITAL function menu.
2. Press the **D0~D7** softkey to select **On** or **Off** to switch all digital channels on or off.
 - Also, under the DIGITAL function menu, press the **Digital** button on the front panel will switch all digital channels off.

To Change the Logic Threshold for Digital Channels

1. Press the **Digital** button on the front panel to open the DIGITAL function menu.
2. Press the **Threshold** softkey to enter the THRESHOLD menu.
3. Press the **D0~D7** softkey, then turn the **Universal Knob** to select a logic family preset or select Custom to define you own threshold.

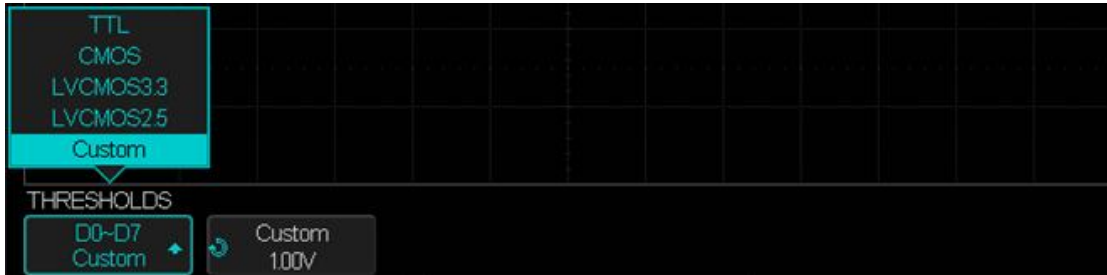


Figure 66 THRESHOLDS Function Menu

Logic Family	Threshold Voltage
TTL	1.5V
CMOS	1.65V
LVCMOS3.3	1.65V
LVCMOS2.5	1.25V
Custom	Variable from -3V to +3V

- The threshold you set applies to all channels.
- Values greater than the set threshold are high (1) and values less than the set threshold are low (0).
- If the **D0~D7** softkey is set to **Custom**, press the **Custom** softkey, then turn the **Universal Knob** to select the desired value and push down the knob to confirm.

To Reposition a Digital Channel

1. Press the **Digital** button on the front panel to open the DIGITAL function menu and display the digital channel waveforms.
2. Turn the **Ref/Math Waveform Vertical Position Knob** to select the channel to reposition. The selected waveform is highlighted in red.
3. Turn the **Ref/Math Waveform Vertical Variable Knob** to move the selected channel to current position.

To Display Digital Channels as a Bus

Digital channels may be grouped and displayed as a bus, with each value displayed at the bottom of the display in hex or binary. You can create up to two buses.

To configure and display each bus, do the following steps:

1. Press the **Digital** button on the front panel to open the DIGITAL function menu and display the digital channel waveforms.
2. Press the **Digital Bus** softkey to enter the DIGITALBUS function menu.



Figure 67 DIGITALBUS Function Menu

3. Press the **Bus Select** softkey to select **Bus1** or **Bus 2**.
4. Press the **Display** softkey to select **On** to display the bus at the bottom of the display.
 - Under the DIGITAL function menu, press the **ChannelControl** softkey, turn the **Universal Knob** to select such as **DX**, and then press the **Dx** softkey to select **On** or **Off** to set the channel to be included or excluded in the bus.
 - Under the DIGITAL function menu, press the **D0~D7** softkey to select **On** or **Off** to include or exclude all digital channels.
 - Under the DIGITAL function menu, an status indicator is displayed in the status line at the bottom of the display. Light blue shows the digital channel is included in the bus while black shows it is excluded in the bus.
 - Under the DIGITAL function menu, press the **System Display** softkey to select **Binary** or **Hex** to display the bus values.
 - The Sampling rate of digital channels is displayed at the bottom of the display.

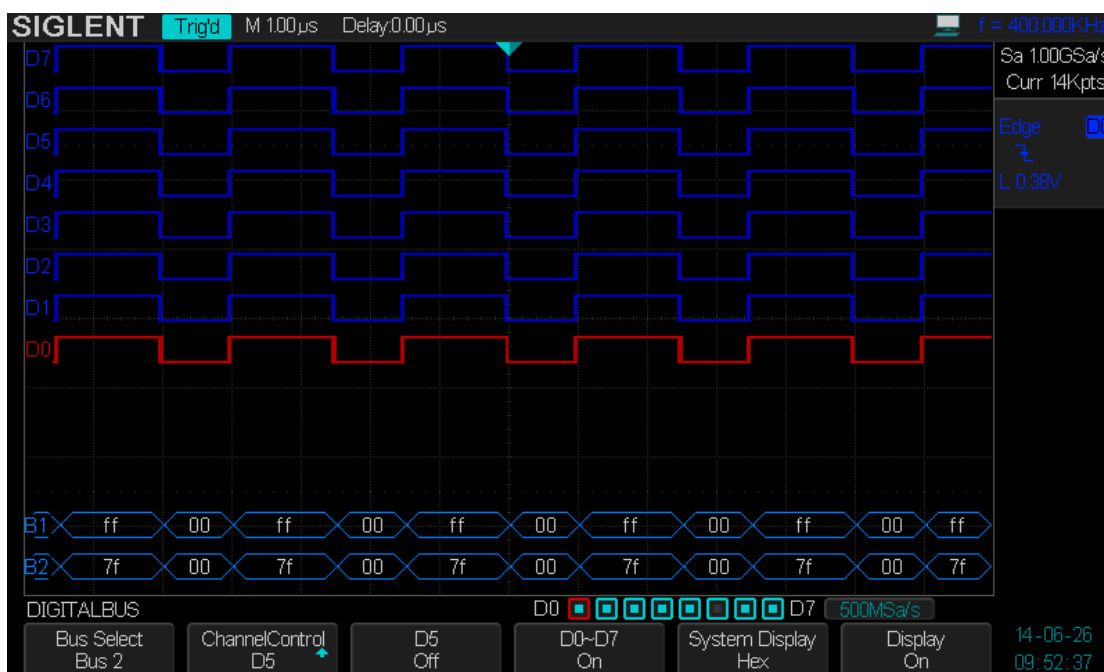


Figure 68 Digital Bus

To Save Reference Waveform

SDS2000 can save analog channel or math waveforms to one of four reference waveform locations in the oscilloscope. Then, a reference waveform can be displayed and compared against other waveforms. Four reference waveforms can be displayed at a time.

Use the appointed Variable knob and the Position knob which on the right side of the front panel to adjust the vertical scale and the vertical position of REF waveform.

The contents of this chapter:

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

To Save REF Waveform to Internal Memory

Do the following steps to save the REF waveform to internal memory:

1. Press the **REF** button on the front to enter the REF WAVE function menu. Note that when the time horizontal format is in X-Y mode, REF function cannot be enabled.
2. Press the **Source** softkey; then, turn the **Universal Knob** to select the source of reference channel. The source includes analog channel and math waveforms.
3. Press the **Location** softkey; then, turn the **Universal Knob** to select the position to save the REF waveform. The source includes analog channel and math waveforms.
4. Press the **Save** softkey to save the channel or math waveform to the appointed location. The vertical scale information and the vertical offset of the waveform will be saved at the same time. It will pop out the message “**Store Data Success**” when the waveform has been saved successfully.

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

To Display REF Waveform

To the following steps to display REF waveform:

1. Press the **REF** button on the front to enter the REF WAVE function menu.
2. Press the **Location** softkey; then, turn the **Universal Knob** to select the REF waveform that you want to display.
3. Press the **Display** softkey to select **On** to display the REF waveform on the screen. Only saved location can be displayed. SDS2000 can display all four reference waveforms at a time.

To Adjust REF Waveform Display

1. Please refer to the “**To Display REF Waveform**” above to display the desired reference waveform.
2. Turn the **Math/Ref Waveform Vertical Scale Knob** to adjust the vertical scale of the reference waveform. Turn the Variable knob clockwise to decrease the vertical scale and the amplitude of the waveform will increase; Turn the **Math/Ref Waveform Vertical Scale Knob** anticlockwise to increase the vertical scale and the amplitude of the waveform will decrease. The vertical scale information displays at the middle of the screen.
3. Turn the **Math/Ref Waveform Vertical Position Knob** to adjust the vertical position of the reference waveform. Turn the **Math/Ref Waveform Vertical Position Knob** clockwise to increase the vertical offset and the reference waveform will move up; Turn the Position knob anticlockwise to decrease the vertical offset and the reference waveform will move down. The vertical position information displays at the middle of the screen.

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

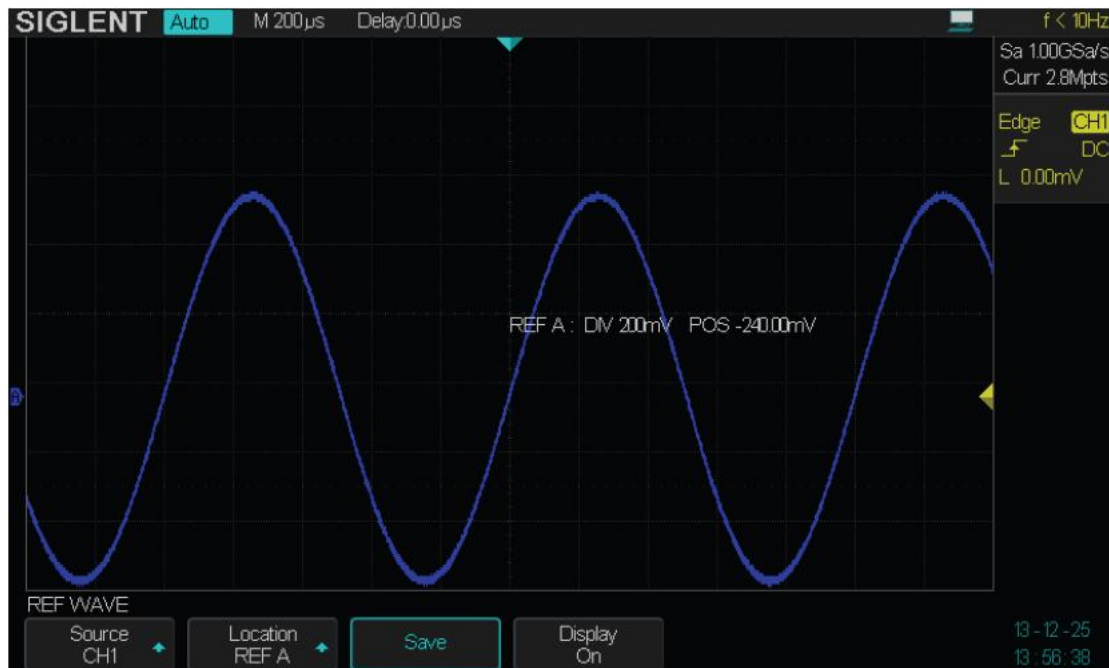


Figure 69 Reference Waveform

Note: The vertical Variable knob and the vertical Position knob are only works for the reference waveform and the math waveform. Please make a difference with the channel vertical system.

To Clear REF Waveform Display

SDS2000 does not have the “Clear” option under the REF WAVE function menu. To clear the appointed reference waveform, you can save a new reference waveform to the same location to cover it.

To Make Math Operation

SDS2000 supports many math operations between analog channels and reference waveforms, including addition (+), subtraction (-), multiplication (*), division (/), differential (d/dt), integral ($\int dt$), square root ($\sqrt{\quad}$). The resulting math waveform is displayed in white and labeled with "M". You can use cursors to measure it.

The contents of this chapter:

- ◆ Units for Math Waveforms
- ◆ Math Operators
- ◆ To Adjust the Math Waveform Scale and Offset

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

Units for Math Waveforms

Use the channel function menu to set the unit of each channel to “V” or “A”. SDS2000 math operation includes units as below:

Math Operation	Unit
Addition (+)or subtraction (-)	V, A
multiplication (*)	V ² , A ² or W (Volt-Amp)
division (/)	None or S
FFT	dBV _{rm} , V _{rm} , dBArms, Arms
differential (d/dt)	V/S or A/S (V/second or A/second)
integral (∫dt)	VS or AS (V/second or A/second)
square root (√)	V ^{1/2} or A ^{1/2}

Math Operators

SDS2000 supports math count operation (Addition, subtraction, multiplication, division), FFT (Fourier transform) operation and math function operation (differential, integral, square root).

Addition or Subtraction

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

1. Press the **Math** button on the front panel to enter the MATH function menu.
2. Press the **Source A** and **Source B** softkey respectively, and then turn the **Universal Knob** to select the source to do math operation. Analog channels (CH1, CH2, CH3, and CH4) and reference waveforms (REFA, REFB, REFC and REFD) can be used as Source A or source B.
3. Press the **Operation** softkey and then turn the universal to select **+** or **-** to make addition or subtraction operation. The resulting math waveform is displayed in white and labeled with "M".



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

Multiplication and Division

Math operators perform arithmetic operations multiplication or division operation on any two analog input channels. When you select multiplication or division, the **Source A** and **Source B** values are multiplied or divided point by point and the result is displayed.

1. Press the **Math** button on the front panel to enter the MATH function menu.
2. Press the **Source A** and **Source B** softkey respectively, and then turn the **Universal Knob** to select the source to do math operation. Analog channels (CH1, CH2, CH3, and CH4) and reference waveforms (REFA, REFB, REFC and REFD) can be used as Source A or source B.
3. Press the **Operation** softkey and then turn the universal to select * or / to make multiplication or division operation. The resulting math waveform is displayed in white and labeled with "M".



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

FFT Operation

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

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

To display a FFT waveform:

1. Press the **Math** button on the front panel to open the MATH function menu.
2. Press the **Operation** softkey and then turn the **Universal Knob** to select **FFT**. The resulting math waveform is displayed in white and labeled with “M”.
3. Press the **Source** softkey, and then turn the **Universal Knob** to select the source to do FFT operation. Analog channels (CH1, CH2, CH3, and CH4) and reference waveforms (REFA, REFB, REFC and REFD) can be used as the source.
4. Press the **Window** softkey, and then turn the **Universal Knob** to select an appropriate window.

Spectral leakage can be considerably decreased when a window function is used. SDS2000 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. Please read the table below carefully to make a appropriate option according to the input signal.

Window	Characteristics	Measurement
Rectangle	The best frequency resolution; the poorest amplitude resolution; similar to the situation when no window is multiplied.	Transient or short pulse, the signal levels before and after the multiplication are the basically the same; Sine waveform with the same amplitude and rather similar frequencies; Wide band random noise with relatively slowly changing waveform spectrum.
Hanning	Better frequency resolution; poorer amplitude resolution.	Sine, periodic and narrow band random noise.
Hamming	A litter 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.

5. Press the **FFT Zoom** softkey, and then turn the **Universal Knob** or turn the horizontal scale knob to select the desired magnifying multiple (1X, 2X, 5X, 10X). Set the **FFT Zoom** to an appropriate magnifying multiple to observe more details of the FFT waveform.
6. Press the **Scale** softkey to select the unit of vertical axis. The unit of the vertical axis can be dB or Vrms which 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, dBVrms is recommended.
7. Press the **Display** softkey to select **Split** 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.

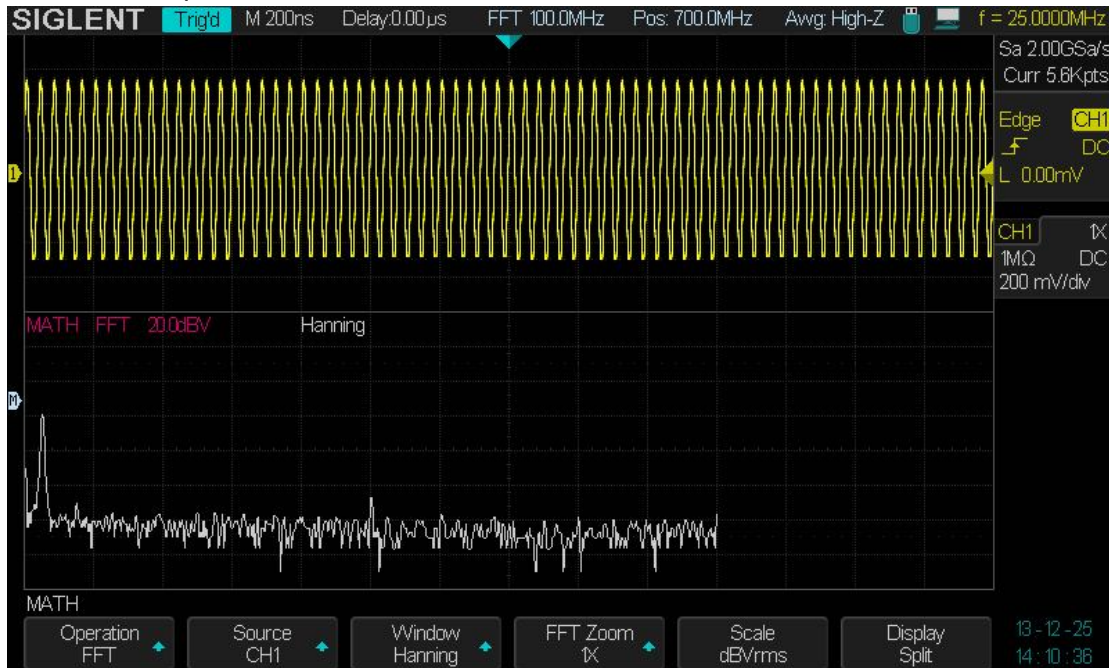


Figure 70 FFT Waveform In Split Mode

Note:

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

To measure FFT waveform:

To make cursor measurements, press the **Cursors** button, and then press the **Mode** softkey to select **On** to turn the cursors, Use the X1 and X2 cursors to measure frequency values and the difference between two frequency values (ΔX). Use the Y1 and Y2 cursors to measure amplitude in dB and difference in amplitude (ΔY).

You can find the frequency value at the first occurrence of the waveform maximum by using the X at Max Y measurement.

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

Math Function Operation

SDS2000 supports math function operation including differential (d/dt), integral ($\int dt$) and square root ($\sqrt{\quad}$).

Differentiate

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

$$di = \frac{y(i + \Delta t) - y(i - \Delta t)}{2 \Delta t}$$

Where:

- d = differential waveform.
- y = channel 1, 2, 3, or 4 data points.
- i = data point index
- Δt = point- to- point time difference.

The **dx** option under d/dt math function operation menu shows the point- to- point time difference, and it ranges from 0.02div to 0.40div. “div” indicates the number of the pixel points that each division has. SDS2000 has 50 pixel points per division. Take 0.2div as an example: $0.2 * 50 = 10$. It means to calculate the ten point’s discrete time derivative of the selected source, and the Δt is the ten point’s point- to- point time difference.

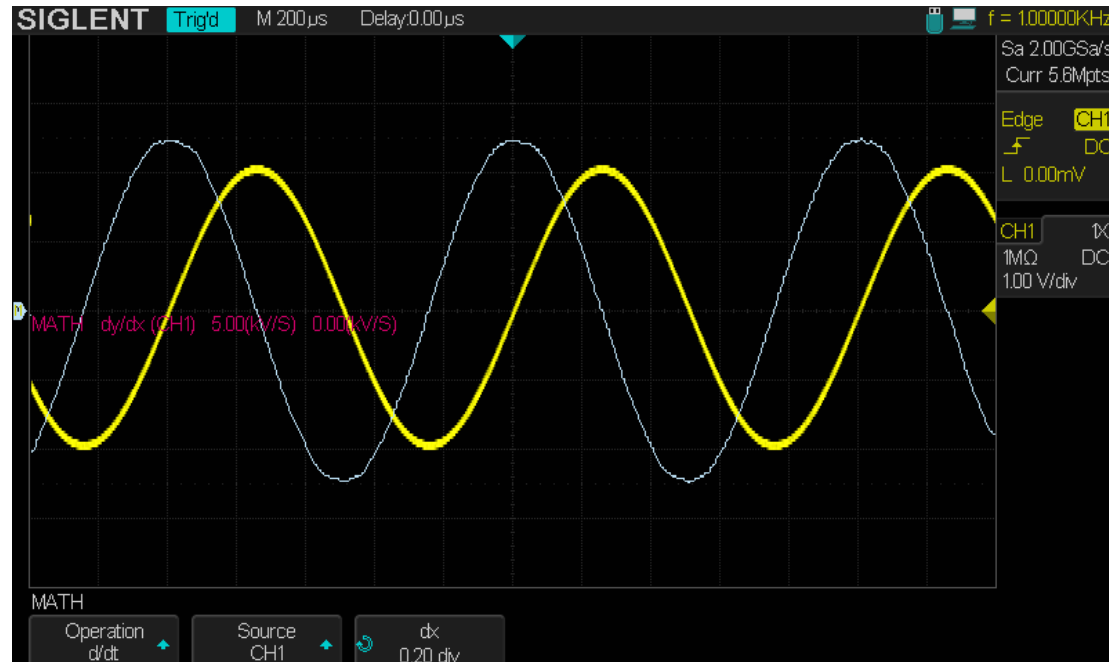


Figure 71 Differential Function Operation

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

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

Integrate

dt (integrate) calculates the integral of the selected source. You can use integrate to calculate the energy of a pulse in volt- seconds or measure the area under a waveform. dt plots the integral of the source using the "Trapezoidal Rule". The equation is:

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

Where:

- I = integrated waveform;
- Δt = point- to- point time difference;
- y = channel 1, 2, 3, or 4, or g(t) (internal arithmetic operation);
- c₀ = arbitrary constant;
- i = data point index;

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

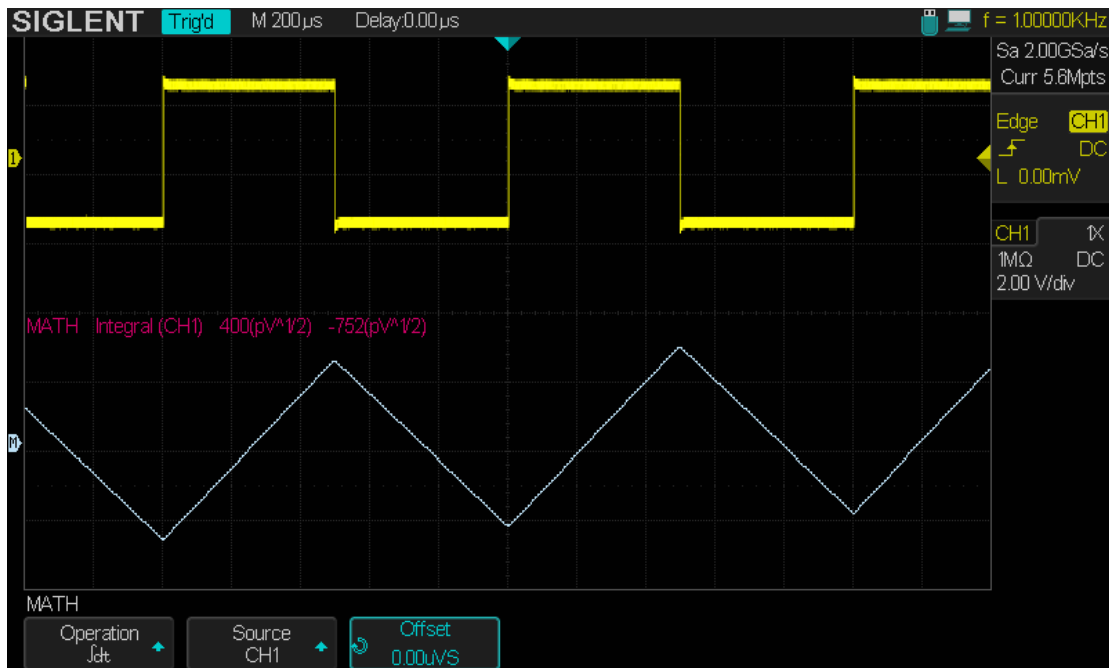


Figure 72 Integral without Offset

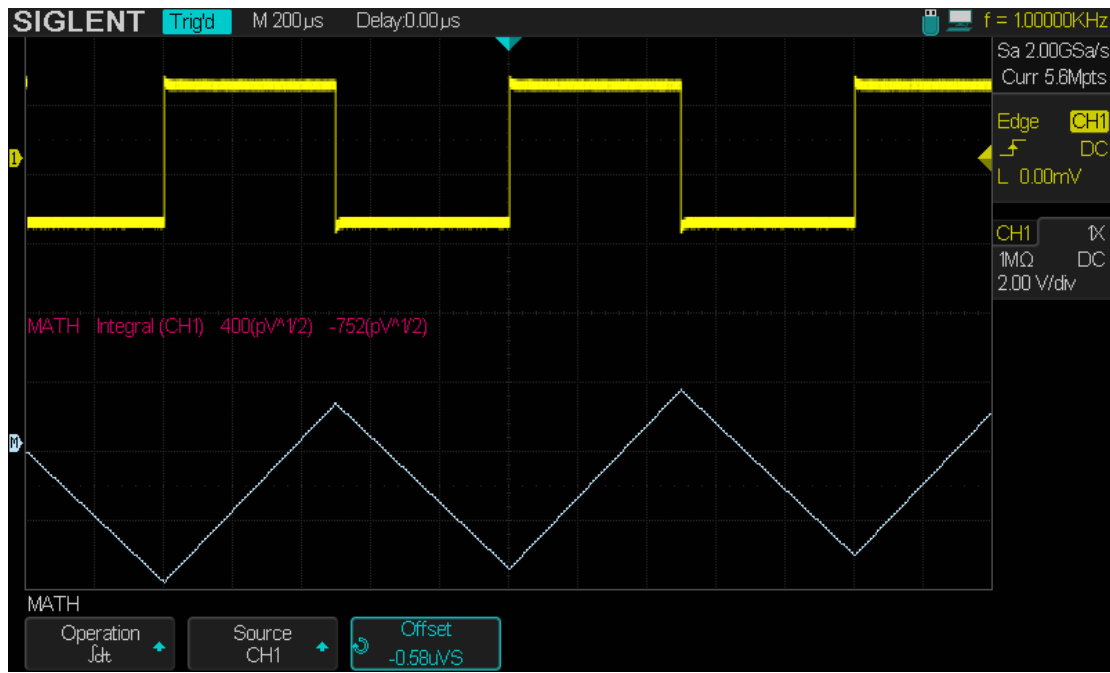


Figure 73 Integral with Offset

Square Root

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

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

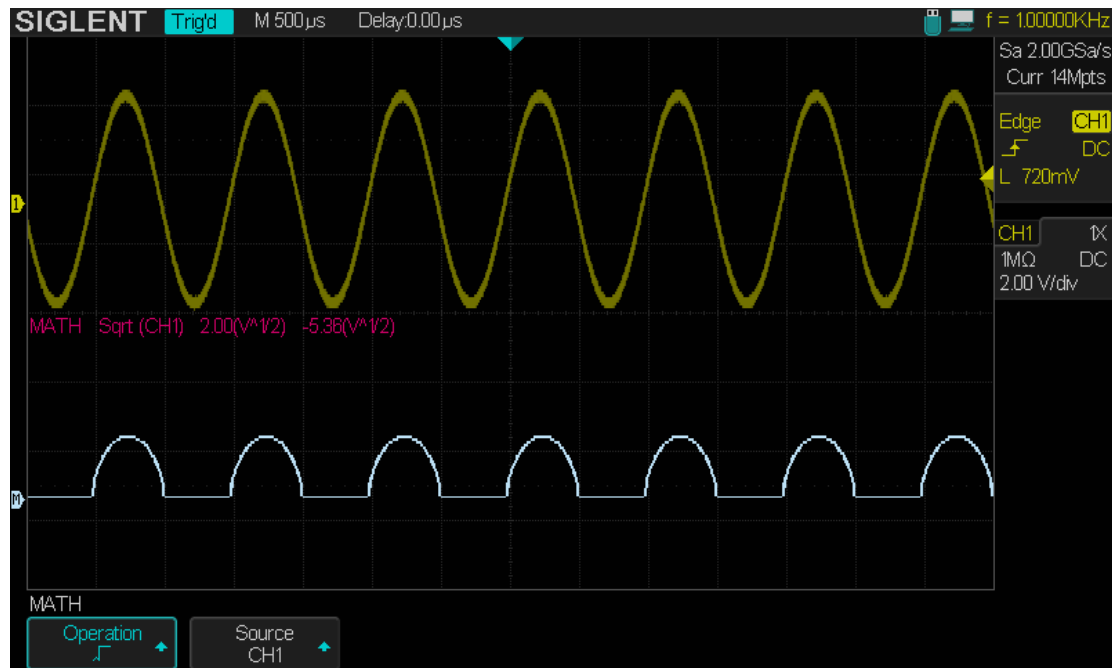


Figure 74 Square Root

To Adjust the Math Waveform Scale and Offset

To Adjust the Math Waveform Horizontal Scale and Offset

To check and analysis the part of the math operation waveform or to find the cared event, you can do some horizontal and vertical adjustment.

To adjust the math waveform horizontal scale and offset

1. Turn the **Horizontal Scale Knob** on the front panel to zoom in or zoom out the math waveform horizontally. The math waveform will zoom in or zoom out along with the source waveform (analog channel or reference waveform).
2. Turn the **Horizontal Position Knob** to turn left or turn right the math waveform horizontally. The math waveform will turn left or turn right along with the source waveform (analog channel or reference waveform).

To Adjust the Math Waveform Vertical Scale and Offset

Please pay attention that the vertical system on the right side of the front only works for the math waveform or the reference waveform,

1. Turn the **Ref/Math Waveform Vertical Scale Knob** on the front panel to zoom in or zoom out the math waveform vertically.
2. Turn the **Ref/Math Waveform Vertical Position Knob** to turn up or turn down the math waveform vertically. The math waveform will turn up or turn down.

To Make Cursors Measurements

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

X Cursors

X cursors are vertical dashed lines that adjust horizontally and can be used to measure time (when the source is FFT waveform, X cursors measure frequency)

X1 cursor is the left (default position) vertical dotted line; it can be moved to any place of the screen.

X2 cursor is the right (default position) vertical dotted line; it can be moved to any place of the screen.

Use the **Universal Knob** to set the **X1** and **X2** cursor values and the values are displayed in the cursors box in the upper-left corner of the screen along with the difference between X1 and X2 (ΔT) and $1/\Delta T$.

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

Y Cursors

Y cursors are horizontal dotted lines that adjust vertically and can be used to measure voltage (V) or current (A). When the cursors source is the math function, the unit will match the math function.

Y1 cursor is the top (default position) horizontal dotted line; it can be moved to any vertical place of the screen.

Y2 cursor is the down (default position) horizontal dotted line; it can be moved to any vertical place of the screen.

Use the **Universal Knob** to set the Y1 and Y2 cursor values and the values are displayed in the cursors box in the top left corner of the screen along with the difference between Y1 and Y2 (ΔY).

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

To Make Cursor Measurements

1. Press the **Cursors** button on the front panel to enter the CURSOR function menu.
2. Press the **Mode** softkey and set the option to **On**.
3. Press the **Source** softkey, and then use the **Universal Knob** to select the desired source. Only analog channels (CH1, CH2, CH3 and CH4), math waveforms and reference waveforms (REFA, REFB, REFC, REFD) that are displayed are available for cursors.
4. To make cursor measurements:
 - To measure the horizontal time, use the **Universal Knob** to move the X1 and X2 cursors to desired place. If necessary, set the cursor type to **X1-X2**, move X1 and X2 cursors together.
 - To measure vertical voltage or current, use the **Universal Knob** to move the Y1 and Y2 cursors to desired place. If necessary, set the cursor type to "Y1-Y2", move Y1 and Y2 cursors together.
 - To adjust the transparency of the cursors message box, press the **Display** button and go to the second page of the DISPLAY function menu, press the **Transparence** (20% to 80%) softkey and then turn the **Universal Knob** to adjust the transparence to the desired value.

Cursor examples:

1. Use cursors to measure pulse width:

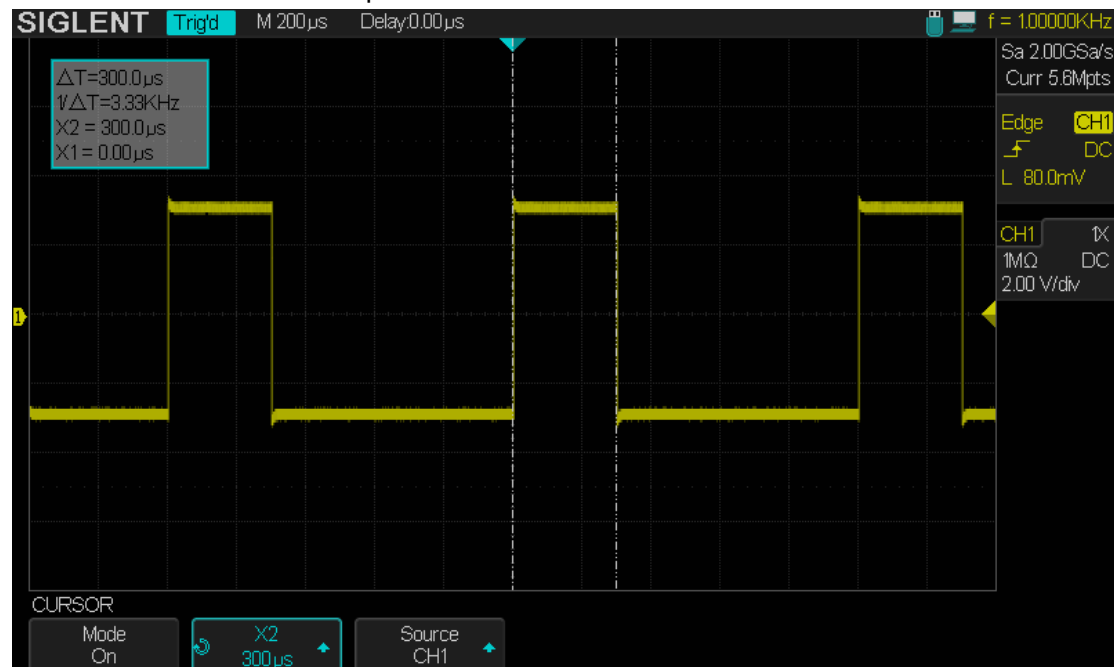


Figure 75 Measure Pulse Width

2. Use "X1-X2" to check the change of the pulse width:

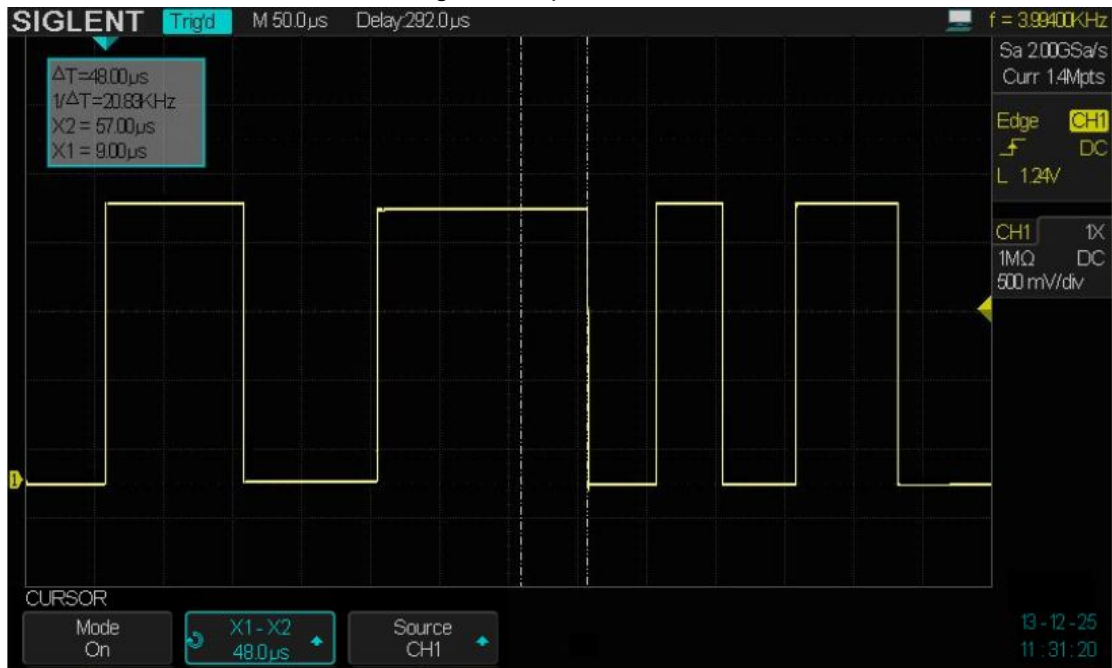


Figure 76 Check the Change of the Pulse Width

To Make Measurements

SDS2000 provides measurements of 32 waveform parameters and the statistics. It contains voltage, time and delay parameters.

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

The contents of this chapter:

- ◆ Type of measurements
 - Voltage Measurements
 - Time Measurements
 - Delay Measurements
- ◆ To make automatic measurement
- ◆ To make statistics measurements
- ◆ To make all parameters measurement

Type of Measurement

Voltage Measurements

Voltage measurements include 14 kinds of voltage parameter measurements.

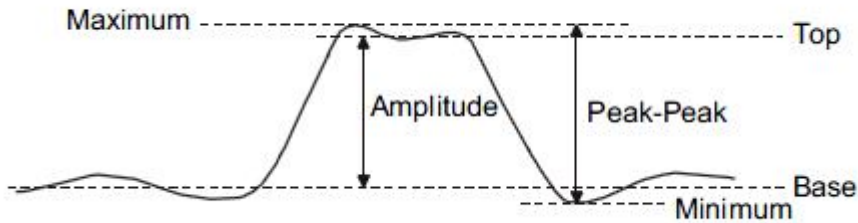


Figure 77 Voltage Measurements

1. **Vpp**: The peak- to- peak value is the difference between Maximum and Minimum values.
2. **Vmax**: Maximum is the highest value in the waveform display.
3. **Vmin**: Minimum is the lowest value in the waveform display.
4. **Vamp**: The Amplitude of a waveform is the difference between its Top and Base values.
5. **Vtop**: The Top of a waveform is the mode (most common value) of the upper part of the waveform, or if the mode is not well defined, the top is the same as Maximum.
6. **Vbase**: The Base of a waveform is the mode (most common value) of the lower part of the waveform, or if the mode is not well defined, the base is the same as Minimum. The Y cursor shows the value being measured.
7. **Vmea**: The arithmetic mean over the first cycle of the waveform.
8. **Mean**: The arithmetic over the entire waveform.
9. **Vrms**: The true Root Mean Square voltage over the entire waveform.
10. **Crms**: The true Root Mean Square voltage of the first cycle in the waveform.
11. **Overshoot**: Overshoot is distortion that follows a major edge transition expressed as a percentage of Amplitude. ROV means rising edge overshoot and FOV means falling edge overshoot.

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

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

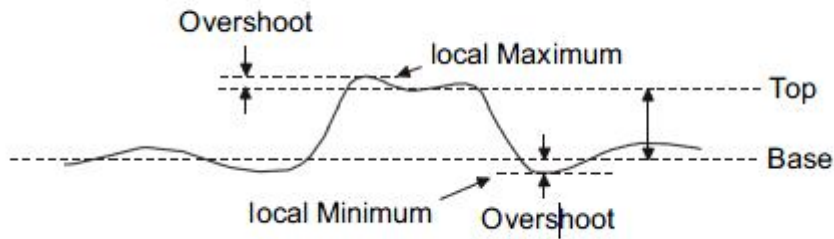


Figure 78 Overshoot

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

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

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

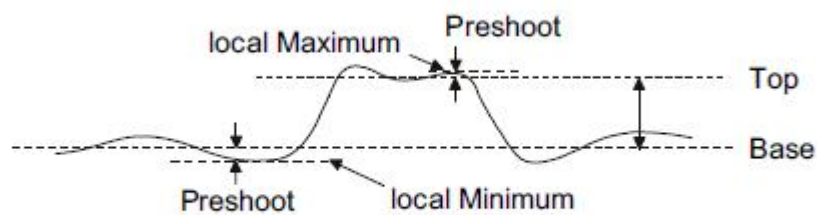


Figure 79 Preshoot

Time Measurements

Time measurements include 9 kinds of time parameter measurements.

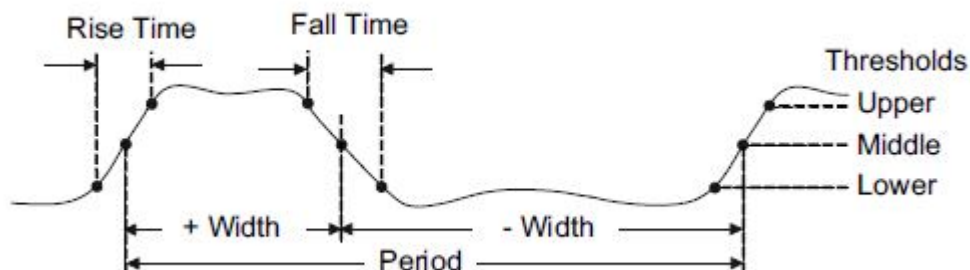


Figure 80 Time Measurements

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. **+ Width:** the time difference between the 50% threshold of a rising edge to the 50% threshold of the next falling edge of the pulse.
4. **- Width:** the time difference between the 50% threshold of a falling edge to the 50% threshold of the next rising edge of the pulse.
5. **Rise Time:** the time for the signal amplitude to rise from 10% to 90%.
6. **Fall Time:** the time for the signal amplitude to fall from 90% to 10%.
7. **BWid:** the duration of a burst over the entire waveform.
8. **+ Duty:** the ratio of the positive pulse width to the period.
9. **- Duty:** the ratio of the negative pulse width to the period.

Delay Measurements

Delay measurements measure the time different between arbitrary two channels, including 9 kinds of delay measurements.

1. **Phase:** The phase difference between two channels.
2. **FRR:** The time between the first rising edge of source 1 and the first rising edge of source 2 of 50 voltage level.
3. **FRF:** The time between the first rising edge of source 1 and the first falling edge of source 2 of 50 voltage level.
4. **FFR:** The time between the first falling edge of source 1 and the first rising edge of source 2 of 50 voltage level.
5. **FFF:** The time between the first falling edge of source 1 and the first falling edge of source 2 of 50 voltage level.
6. **LRR:** The time between the last rising edge of source 1 and the last rising edge of source 2 of 50 voltage level. **LRR:** The time between the last rising edge of source 1 and the last falling edge of source 2 of 50 voltage level.
7. **LRF:** The time between the last rising edge of source 1 and the last falling edge of source 2 of 50 voltage level.
8. **LFR:** The time between the last falling edge of source 1 and the last rising edge of source 2 of 50 voltage level.
9. **LFF:** The time between the last falling edge of source 1 and the last falling edge of source 2 of 50 voltage level.

To Make Automatic Measurement

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

1. Press the **Measure** button on the front panel to enter the MEASURE function menu.
2. Press the **Source** softkey, and then use the **Universal Knob** to select the desired channel. Only analog channels (CH1, CH2, CH3, and CH4) that are displayed are available for measurements.
3. To select and display measurement parameters. Press the **Type** softkey, and then turn the **Universal Knob** to select the desired measurement parameter.
4. Press the **Add** softkey to display the measurement parameter at the bottom of the screen.
5. To display several measurement parameters, press the **Type** softkey and then press the **Add** softkey for several times.

The measurement display area can display 5 measurement parameters at most, and the measurements will arrange according to the selecting order. If press the **Add** softkey to add a sixth measurement parameter, it will delete the first measurement.



Figure 81 Before Adding the Duty Measurement



Figure 82 After Adding Duty Measurement

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

To Clear Measurement Parameters

Press the **Clear** softkey to clear all the measurement parameters that are displaying on the screen. Single appointed measurement cannot be deleted.

To Make Statistics Measurements

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

Before open the statistic function, please perform the “To Make Automatic Measurement” steps to add the desired measurements parameter.

1. Press the **Statistics** softkey to enter the STATISTIC function menu.
2. Press the **Display** softkey to set the option to **On** to display the statistics.

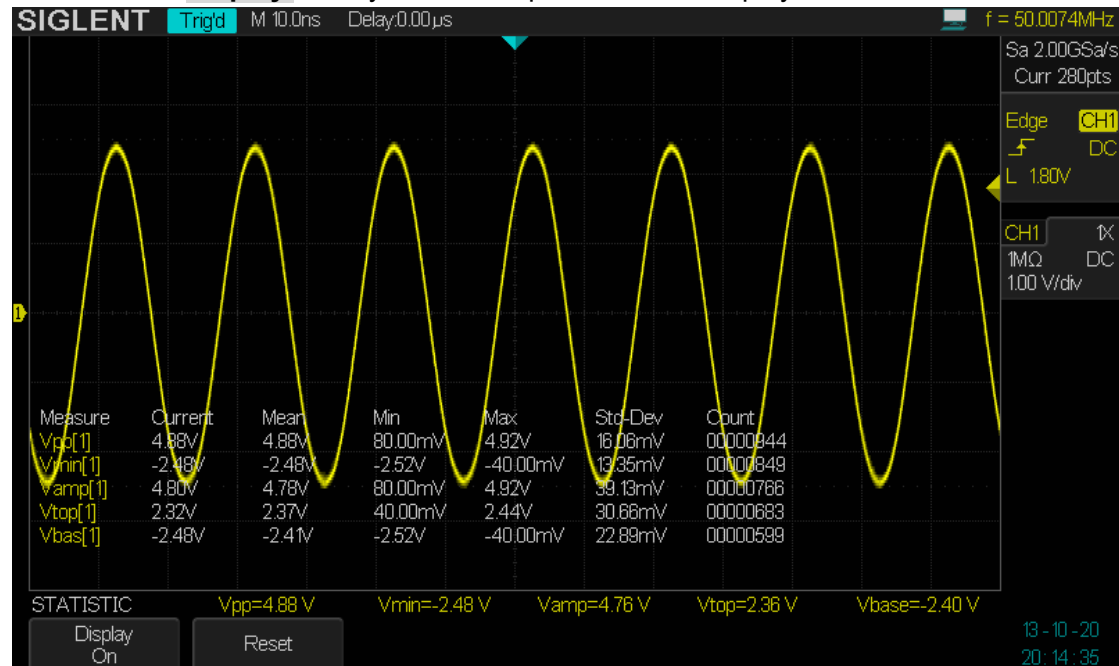


Figure 83 Statistics Measurement

3. To clear the history data and make statistical again, press the **Reset** softkey.
4. To turn off the statistic function, press the **Display** softkey to select **Off**.

To Make All Parameters Measurement

All measurement could measure all the voltage, time and delay parameters of the current measurement source and display the results on the screen.

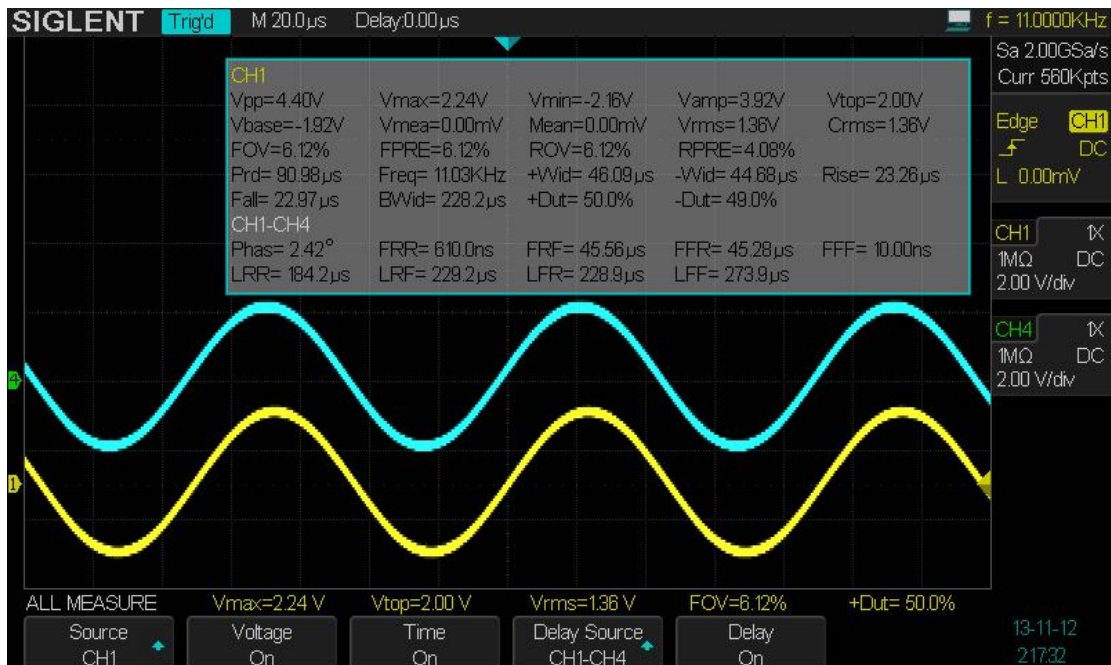


Figure 84 All Parameters Measurement

Do the following steps to make all parameters measurement.

1. Press the **Measure** button on the front panel to enter the MEASURE function menu.
2. Press the **All Measure** softkey to enter the ALL MEASURE menu.
3. Press the **Source** softkey to select the desired channel (CH1, CH2, CH3 and CH4). Only channels that are displayed are available
4. Press the **Voltage** and the **Time** softkeys and set the option to **On** to display all the voltage and time measurement parameters.
5. Press the **Delay Source** to select the desired channel group (CH1-CH2, CH1-CH3, CH1-CH4, CH2-CH3, CH2-CH4 and CH3-CH4). Only channels that are displayed are available.
6. Press the **Delay** softkey to set the option to **On** to display all the delay measurement parameters.

Display Setting

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

The contents of this chapter:

- ◆ To Set Display Type
- ◆ To Set Color Display
- ◆ To Set and Clear Persistence
- ◆ To Clear the Display
- ◆ To Select Grid Type
- ◆ To Adjust the Waveform Intensity
- ◆ To Adjust the Grid Brightness
- ◆ The Adjust the Transparenance

To Set Display Type

Press the **Display** button on the front panel, and then press the **Type** softkey to select **Vectors** or **Dots** display type.

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

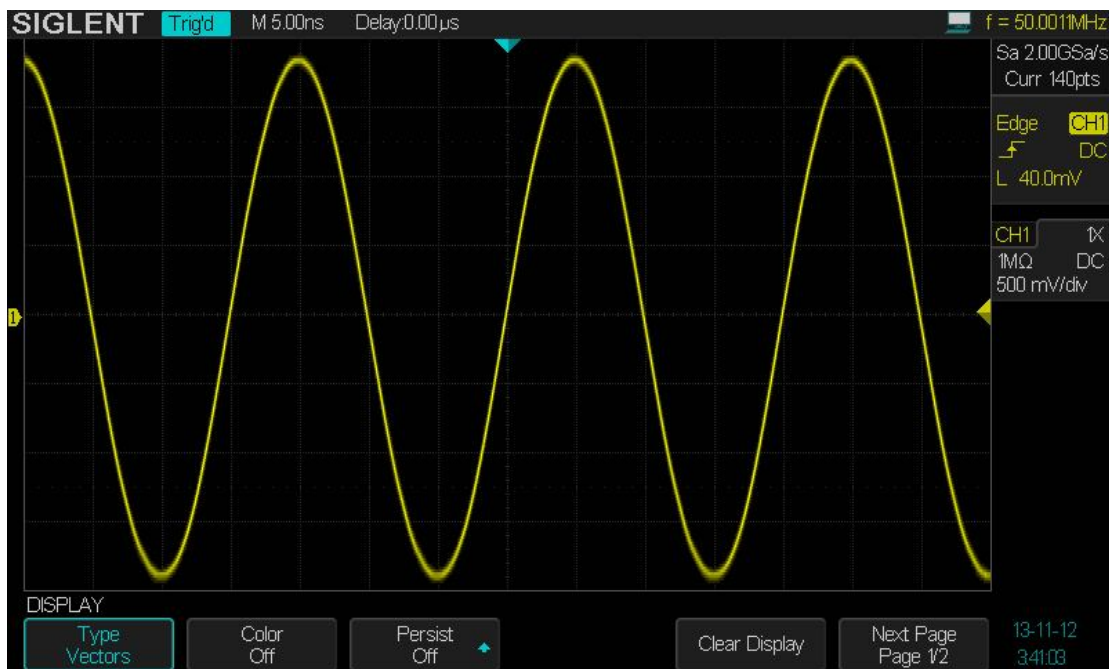


Figure 85 Vectors Display

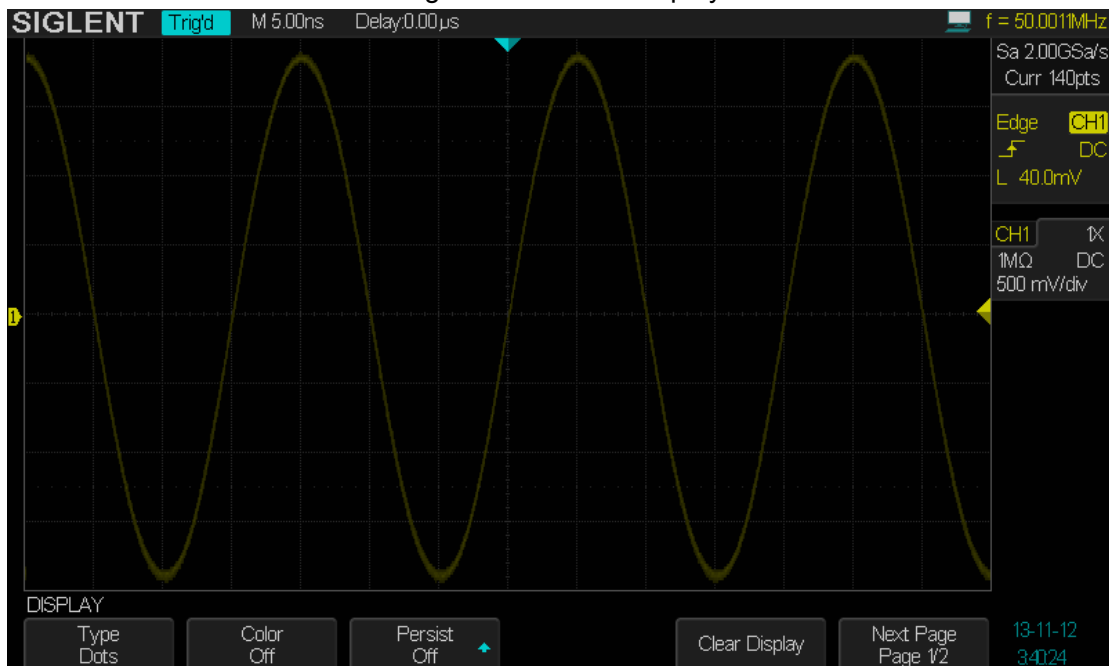


Figure 86 Dots Display

To Set Color Display

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

The picture below shows the change of color from cold to warm. Press the Display button on the front panel, and then press the **Color** softkey and set the option to **On** to turn on the color temperature function. You can compare the waveform's color with the picture below to just the probability that the waveform appears.

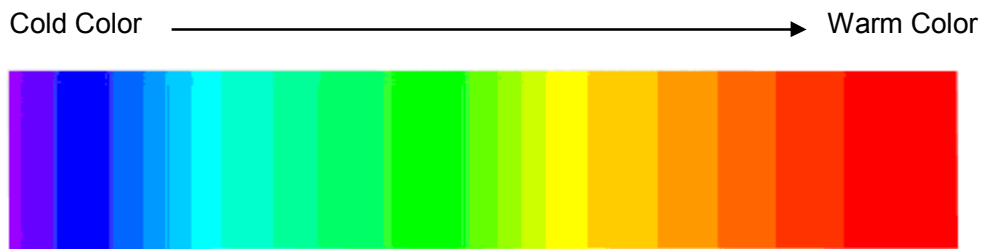


Figure 87 Color Temperature

To Set and Clear Persistence

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

Do the following steps to set and clear persistence:

1. Press the **Display** button on the front panel to enter the DISPLAY function menu.
2. Press the **Persist** softkey; then turn the **Universal Knob** to select the desired option.
 - **Off** —turn off persistence.
 - Variable persistence time (1 second, 5 seconds, 10 seconds, 30 seconds) — select different persistence time, the results of previous acquisitions are erased after a certain amount of time
 - **Infinite** —select “Infinite” Results of previous acquisitions are never erased. Use infinite persistence to measure noise and jitter, to see the worst- case extremes of varying waveforms, to look for timing violations, or to capture events that occur infrequently.

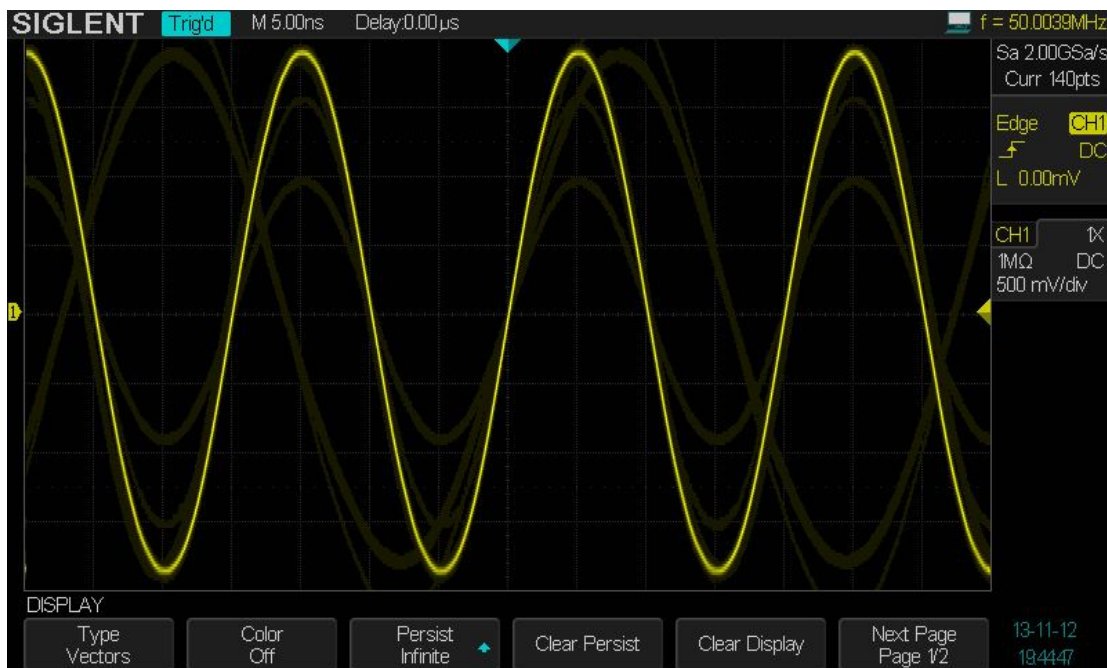


Figure 88 Persist Set to Infinite

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

To clear the display






Press the Display button on the front panel to enter the DISPLAY function menu; press the Clear Display softkey to clear all the waveforms displaying on the screen and acquire and display new waveforms.

To Select Grid Type

To select grid type

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

There are 5 kinds of grid types that are available. Select the grid type according to your really demand.

-  Display 14X8 grid type
-  Display 2X2 grid type
-  Display without grid
-  Display 7X8 grid type
-  Display 14X16 grid type

To Adjust Waveform Intensity

Do the following steps to adjust waveform intensity:

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

Increasing the intensity lets you see the maximum amount of noise and infrequently occurring events. Reducing the intensity can expose more detail in complex signals as shown in the following figures.



Figure 89 Intensity Set to 100%



Figure 90 Intensity Set to 20%

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

To Adjust Grid Brightness

Do the following steps to adjust the grid brightness:

1. Press the **Display** button on the front panel to enter the DISPLAY function menu.
2. Press the **Next Page** softkey to go to the second page of the Display function menu.
3. Press the **Brightness** softkey; and then turn the **Universal Knob** to select the desired value. The default value is 40%, and the range is from 0% to 100%.

To Adjust Transparence

Transparence can be used to adjust the transparence of the message box of cursor, measure, Pass/Fail and all pop-up menus to an appropriate value to observe the date more conveniently.

Under Cursor or Measure or any other menu operation, if want to change the transparence of the message box, do the following steps:

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

Save and Recall

Oscilloscope setups, waveforms, pictures, and CSV files can be saved to internal oscilloscope memory or to a USB storage device. The saved setups, waveforms can be recalled later. SDS2000 provides an USB Host interface on the front panel to connect an USB device for external storage.

The contents of this chapter

- ◆ Save Type
- ◆ Internal Save and Recall
- ◆ External Save and Recall
- ◆ Disk Management

Save Type

SDS2000 supports setups, waveforms, pictures and CSV files storage. The default save type is setups.

1. Setups

It's the default storage type of SDS2000. SDS2000 saves the settings of the oscilloscope in internal or external memory in "*.SET" format. At most 20 setting files (from No.1~ No.20) can be stored in internal memory. The stored settings can be recalled.

2. Waveform

SDS2000 saves the waveform data in external memory in "*.DAV" format. The data of all the channels turned on can be saved in the same file. At recall, the data will be displayed on the screen directly.

3. Picture

SDS2000 saves the screen image in external memory in "*.bmp" format. You can specify the file name and saving directory under the same directory using the same file name. The recall of image is not supported.

4. CSV

SDS2000 saves the waveform data in external memory in "*.CSV" format. The stored files contain the waveform data of the displayed analog channels and the main setting information of the oscilloscope. The recall of CSV file is not supported.

Set the save type to **CSV**, and set the **Para Save** option to **On** or **Off** to turn on or of the parameters storage function.

Internal Save and Recall

Internal save and recall support Setups in **Save/Recall**. In the following part, the save and recall method and procedures are introduced.

➤ **Save the specified oscilloscope setting in internal memory.**

1. Connect the signal to the oscilloscope and obtain stable display.
2. Press **Save/Recall** button on the front panel to enter the SAVE/RECALL function menu.
3. Press the **Type** softkey and then turn the **Universal Knob** to select **Setups**; and then press the knob to confirm.
4. Press the **Save To** softkey to select **Internal** to save the current setup of the oscilloscope to the internal memory.
5. Press the **Setup** softkey button; and then turn the **Universal Knob** to select the location to save. The internal memory can save as many as 20 setup files, from No.1~No.20.
6. Press the **Save** softkey to save the current setup to the appointed location. After a few seconds, it will pop-out the message “**Store Data success!**”

➤ **Load the specified type of file in internal memory.**

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

Press the **Setup** softkey, and then turn the **Universal Knob** to select the location that you want to recall, press the **Recall** softkey to recall the setup, and it will pop-out the message “**Read Data Success!**”

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

External save and recall

Before using external storage and recall, make sure that the USB flash device is connected correctly. External storage supports all the types of files in save, but in recall, **Picture** and **CSV** are not supported.

➤ **Save the specified type of file in the external USB flash device.**

1. Press the **Save/Recall** button on the front panel to enter the SAVE/RECALL function menu.
2. Press the **Type** softkey to select **Waveforms** or **Setup**
3. Press the **Recall** softkey to enter the SAVE/RECALL file system.

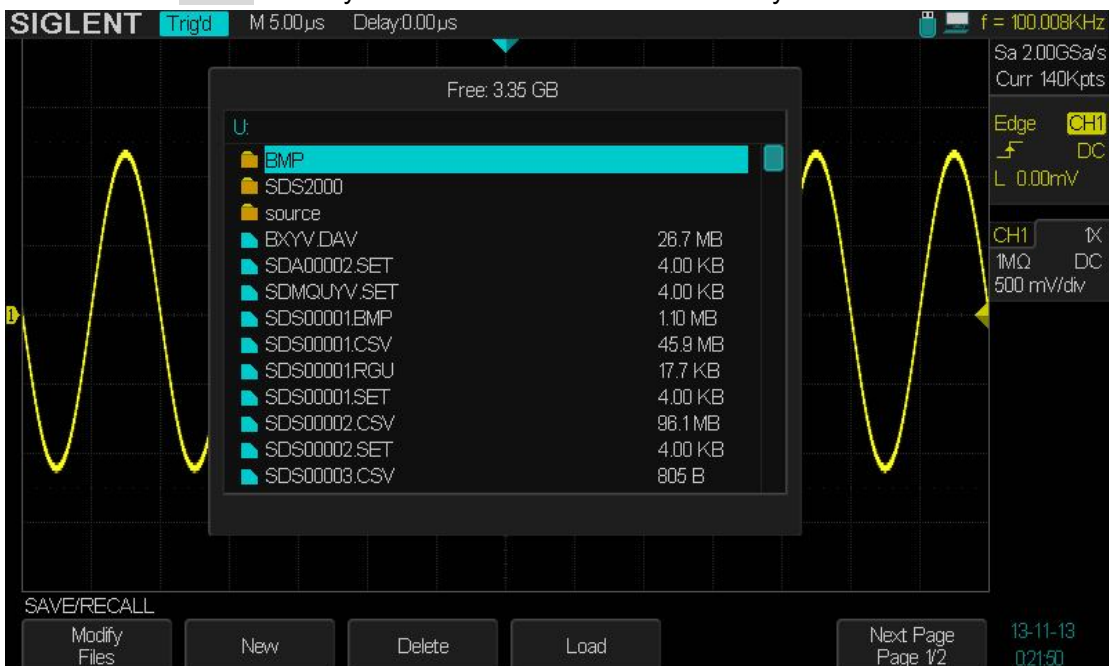


Figure 91 SAVE/RECALL File System

4. Use the **Universal Knob** to select the desired location. File can be stored under the root directory or in a certain folder under the root directory of the USB storage device.

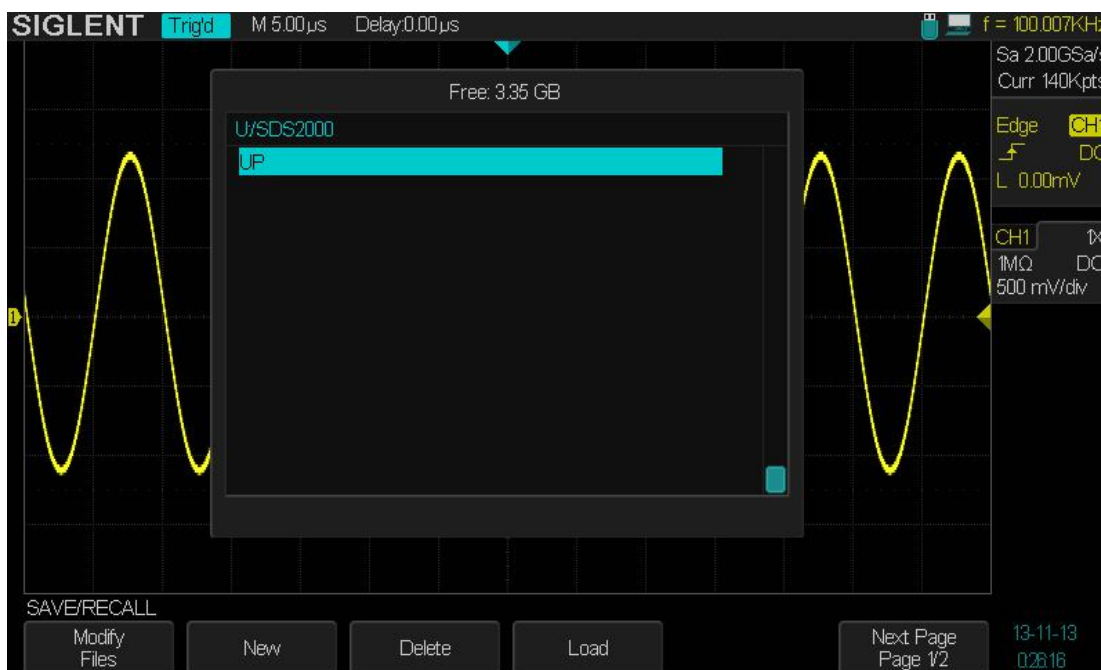


Figure 92 Select Save Location

5. After the save position is selected, press the **New** softkey to turn on the interface as shown in the figure below. Refer to the descriptions in “**To Create a new file or fold**” to create a new file name.

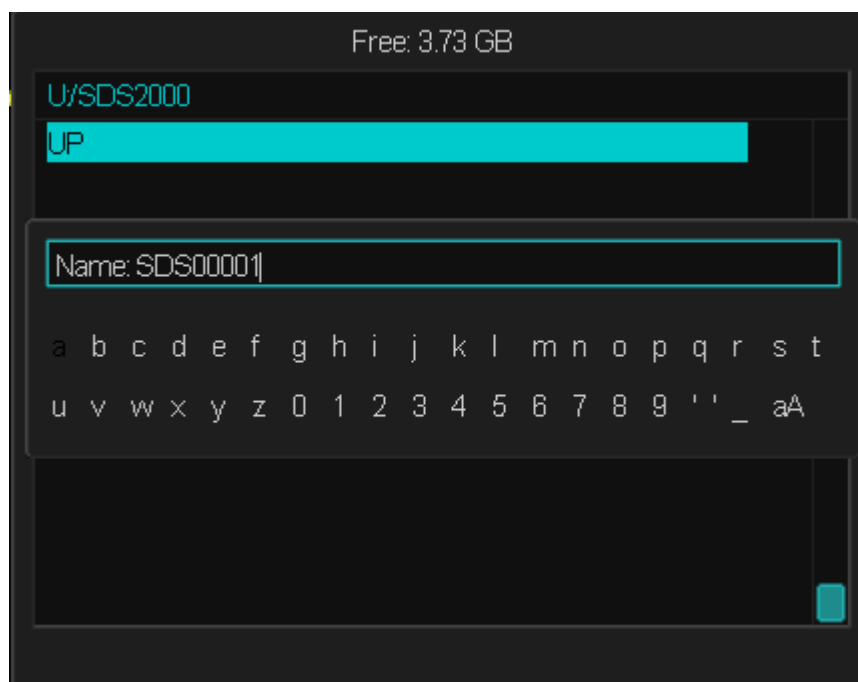


Figure 93 File Name Dialogue

6. Press the Enter softkey to save the current waveform to the external USB storage device,

- **Load the specified type of file in the external USB storage device.**
 7. Press the **Save/Recall** button on the front panel to enter the SAVE/RECALL function menu.
 8. Press the **Type** softkey to select **Waveforms** or **Setup**
 9. Press the **Recall** softkey to enter the SAVE/RECALL file system.
 10. Turn the **Universal Knob** to select the file to be recalled, press the **Load** softkey to recall the waveform or setup.

Disk Management

Disk management aim at the save and recall operations after the oscilloscope is connected with a USB storage device. Before using external disk, make sure that the USB storage device is connected correctly.

Execute the following operations through the disk management menu:

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

To Create a New File or Folder

This operation is only valid in external storage. SDS200 supports English input method. The file name or folder name can contain letters, numbers, underscores and spaces. Let's use an example to introduce how to create a file or folder.

Example: create a file or folder named "SDS2013ab"

1. Press the **Save/Recall** button on the front to enter the SAVE/RECALL function menu.
2. Press the **Type** softkey, and then turn the **Universal Knob** to select one of the type (if select **Setups**, please set the **Save To** option to **External**).
3. Press the **Save** or **Recall** softkey to enter the SAVE/RECAL file system.
Press the **Modify** softkey to select **File** or **Folder**, and then press the **New** softkey to open the interface shown as the picture below. It divides into two parts: name input area and keyboard area. The default is name input area. As the picture shows below, "aA" is used to switch upper-lower case.

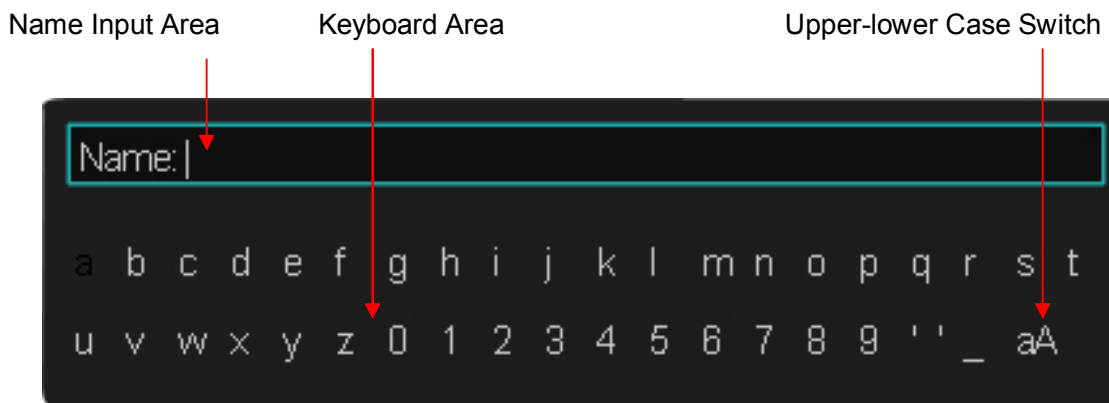


Figure 94 Input Keyboard

4. To delete the name in the name input area, press the **Delete** softkey continuously to delete the character one by one.
5. To input a new name, press the **Switch To** softkey to switch to the keyboard area. The default input method is lower case.
6. Press the ← or the → softkey or turn the **Universal Knob** to select "aA", and then press the **Enter** softkey or push down the **Universal Knob** to set it to "Aa" to set the input method to upper case.
7. Press the ← or the → softkey or turn the **Universal Knob** to select "S" and then press the Enter softkey or the Universal Knob to input the character. Use the same steps to enter the characters "DS2013".
8. Press the ← or the → softkey or turn the **Universal Knob** to select "Aa", and then press the **Enter** softkey or the **Universal Knob** to set it to "aA" to set the input method to lower case.
9. Use the method in step 8 to enter the characters "ab".
10. Press the **Enter** softkey to ensure the file name. You will see a file or folder named "SDS2013ab".

To delete a file or folder

This operation is only valid in external storage.

1. Press the **Save/Recall** button on the front to enter the SAVE/RECALL function menu.
2. Press the **Type** softkey, and then turn the **Universal Knob** to select one of the type (if select Setups, please set the **Save To** option to **External**).
3. Press the **Save** or **Recall** softkey to enter the SAVE/RECAL file system.
4. Turn the **Universal Knob** to select the file or folder to be deleted, and then press the **Delete** softkey. Then the file or folder will be deleted.

To rename a file or a folder

This operation is only valid in external storage.

1. Press the **Save/Recall** button on the front to open the SAVE/RECALL function menu.
2. Press the **Type** softkey, and then turn the Universal Knob to select one of the type (if select Setups, please set the **Save To** option to **External**).
3. Press the **Save** or **Recall** softkey to enter the USB storage device interface.
4. Press the **Next Page** softkey to go to the second page of the SAVE/RECALL function menu.
5. Press the **Rename** softkey, and then refer to the descriptions in “**To Create a new file or fold**” to create a new file name.

System Function Setting

This function module supports the oscilloscope's system-related function, such as system status, language, sound and some other advanced setting, such as do self cal, update and remote interface configure

The contents of this chapter:

- ◆ To View the System Status
- ◆ To Do Self Calibration
- ◆ IO Set
- ◆ To Enable or Disable the Sound
- ◆ To Specify the Language
- ◆ To do Pass/Fail Test
- ◆ To Use the History Function
- ◆ To Print the Screen Image
- ◆ To Enable Quick-Cal
- ◆ To Update Firmware or configuration
- ◆ To Do self Test
- ◆ To Specify the Screen Saver Time
- ◆ Option Management
- ◆ To Set and Display Data/Time

To View the System Status

Do the following steps to view the system status:

1. Press the **Utility** button on the front to enter the UTILITY function menu.
2. Press the **System Status** softkey to view the system status of the oscilloscope. The system status includes the information below:
 - **Startup Times:** record the boot-strap times of the oscilloscope.
 - **Software Version:** list the current software version of the oscilloscope.
 - **Fpga Version:** list the current fpga version of the oscilloscope.
 - **Hardware Version:** list the current hardware version of the oscilloscope.
 - **Product Type:** display the product type of the oscilloscope.
 - **Serial NO.:** list the serial number of the oscilloscope.
 - **Scope ID:** display the scope identification of the oscilloscope.



Figure 95 System Status

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

To Do Self Calibration

The self-calibration program can quickly make the oscilloscope reach the best working state to get the most precise measurement values. You can perform self-calibration at any time especially when the change of the environment temperature is up to or more than 5 °C. Make sure that the oscilloscope has been warmed up or operated for more than 30 minutes before the self-calibration.

Do the following steps to do self calibration:

1. Disconnect all the input channels.
2. Press the **Utility** button on the front panel, and then press the **Do Self Cal** softkey, and the oscilloscope will pop-out the message box shown as below:



Figure 96 Do Self Cal

3. Press the **Single** button on the front panel to perform the self calibration program. During the calibration, most of the keys are disabled.
4. When the self calibration program is finished, it will pop-out the message “**press Run/Stop key to exit**”. Press the **Run/Stop** button on the front panel to exit the calibration interface.

To Use the History Function

The history function can record the waveforms of the input channels before press the **Run/Stop** button or turn the history **View** function. In run state, the oscilloscope records input waveform continually; when fill up the memory (reach the maximal frame), the new frames will cover the old frames and keep the latest frames.

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

Do the following steps to record and replay waveform:

1. Press the **Utility** button on the front to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **History** softkey to enter the HISTORY function menu.

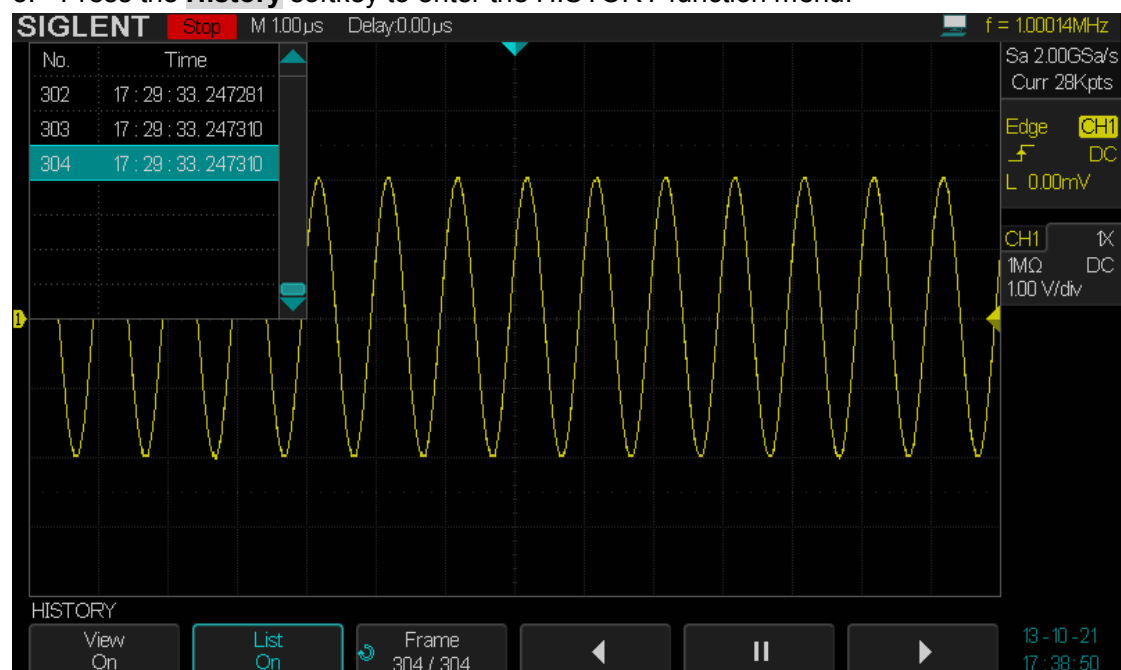





Figure 97 History

4. Press the **View** softkey to select **On** to turn on the history function.
 - Enable the View function will enter the stop state.
 - When in stop state, and then enable the **View** function, the oscilloscope will keep the stop state and the waveforms.
 - When enable the View function, The oscilloscope will stop acquisition.
5. Press the **List** softkey to turn on or off the list display. The list records the timestamp of every frame. It is accurate to microseconds.
6. Press the **Frame** softkey; then turn the **Universal Knob** to select the frame to display.
 - The format of the **Frame** is A/B; A is the frames number that displaying on the screen and B is the maximal frame number you can set.
 - The maximal frame number is determined by the current sampling point (**Curr** value) and sampling rate.
 - When press the **Run/Stop** button or enable the history function, you may not get the maximal frames, because the memory is not filled. So if you want to get the maximal frames, please wait for enough time for acquisition.



The table shows the maximal frame according to the sampling rate and current sampling point's number.

Sample Rate	Curr (pts)	Max. Frame	Sample Rate	Curr (pts)	Max. Frame
2GSa/s	≅ 560	> 50000	1GSa/s	≅ 700	80000
	1.4K	48000		1.4K	63157
	2.8K	32432		2.8K	36585
	5.6K	16997		7K	16043
	14K	8075		14K	8108
	28K	4067		28K	4067
	56K	2037		70K	1630
	140K	815		140K	815
	280K	408		280K	408
	560K	204		700K	163
	1.4M	81		1.4M	81
	2.8M	40		2.8M	37
	5.6M	18		7M	14
	14M	7		≅ 1GSa/s	14M
≅ 2GSa/s	28M	3			

7. Press the  softkey to replay the waveform from the current frame to 1.
8. Press the  softkey to stop replay.
9. Press the  softkey to replay the waveform from the current frame to the last frame.

To Enable or Disable the Sound

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

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

To Specify the Language

SDS2000 supports multiple language menu, Chinese/English help and prompt messages.

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

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

To Do Pass/Fail Test

One way to verify a waveform's compliance to a particular set of parameters is to use pass/fail testing. A pass/fail defines a region of the oscilloscope's display in which the waveform must remain in order to comply with chosen parameters. Compliance to the mask is verified point- by- point across the display.

The test results can be displayed on the screen as well as be declared through the system sound or the pulse signal output from the **[TRIG OUT]** connector at the rear panel.

Pass/Fail test operates on displayed analog channels; it does not operate on channels that are not displayed.

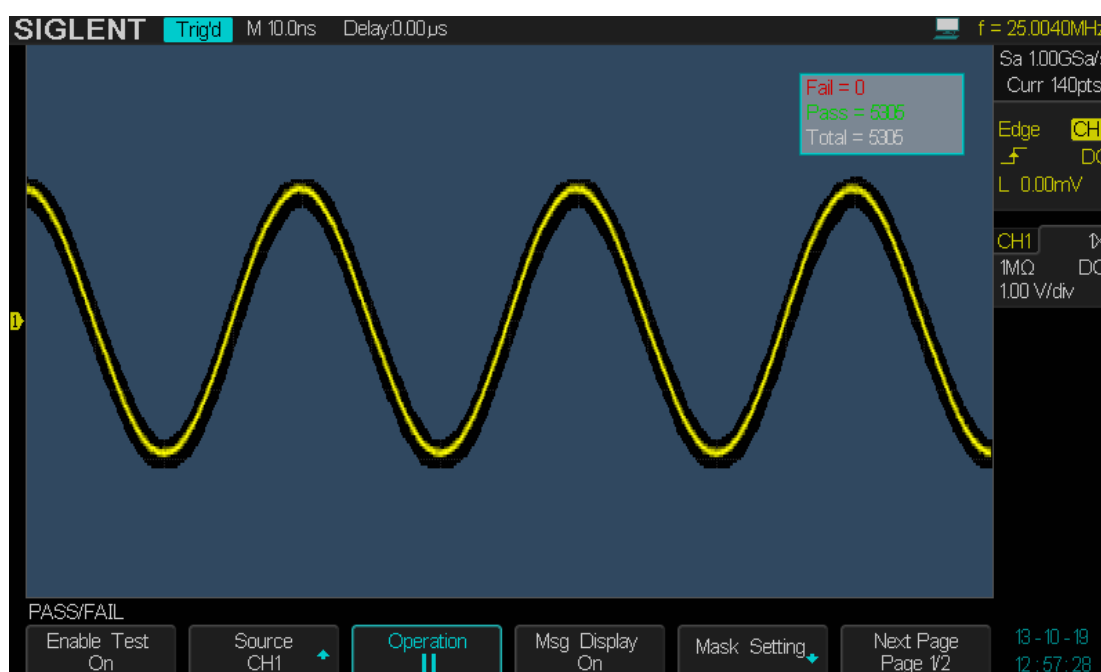
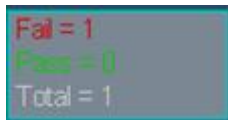






Figure 98 Pass/Fail Test

To Set and Perform Pass/Fail Test

Do the following steps to set and perform pass/fail test:

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Pass/Fail** softkey to enter the PASS/FAIL function menu.
4. Press the **Enable Test** softkey to select **On** to enable the pass/ fail test.
5. Press the **Source** softkey to select the desired channel.
6. Press the **Mask Setting** softkey to enter the MASK function menu.
7. Press the **X Mask** or **Y Mask** softkey; and then turn the **Universal Knob** to select the desired value. The range is from 0.02div to 4div.
8. Press the **Create Mask** button to create the mask. Whenever the **Create Mask** softkey is pressed the old mask is erased and a new mask is created.
9. Press the **UP** softkey to return to the PASS/FAIL function menu.
10. Press the **Msg Display** to select **On** or **Off** to turn on or off the message display. When **On** is selected, the test result will be displayed in the upper-right message box of the screen.



- **Fail:** the frame number of the channel waveform that fails to match the mask.
 - **Pass:** the frame number of the channel waveform that matches the mask.
 - **Total:** the frame number of the channel waveform that have been acquired. It is the total of **Pass** and **Fail**.
11. Press the Next Page softkey to go to the second page of the PASS/FAIL function menu.
 12. Press the Fail softkey to select On or Off to turn on or off the function.
 - **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]** connector (if enabled) 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 **[Trigger Out]** connector at the rear panel each time a failed waveform is detected.
 13. Press the Output softkey to select turn on or off the sound.
 -  : When the failed waveforms are detected, the beeper sounds.
 -  : The beeper will not sound even if the failed waveforms are detected,
 14. Press the Next Page softkey to return to the first page of the PASS/FAIL function menu.
 15. Press the Operate softkey to perform the test.
 -  : Current state is stop; press the softkey to start the pass/fail test.
 -  : Current state is running; press the softkey to stop the pass/fail test.

To Save and Recall Test Mask

Users can save the current test mask to the internal Flash memory or external USB flash device. The file format of the test mask file is “*.RGU”.

➤ Save Test Mask to Internal Memory

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Pass/Fail** softkey to enter the PASS/FAIL function menu.
4. Press the **Enable Test** softkey to select **On** to enable the pass/ fail test.
5. Press the **Source** softkey to select the desired channel.
6. Press the **Mask Setting** softkey to enter the MASK function menu.
7. Press the **X Mask** or **Y Mask** softkey; and then turn the **Universal Knob** to select the desired value.
8. Press the **Location** softkey to select **Internal**
9. Press the **Save** softkey to save the mask to internal memory.
10. A few second later, it will pop-out the message “**Store Data Success!**”

Note: the internal memory can only save one test mask; save a new test mask will cover the old one.

➤ Recall Test Mask from Internal Memory

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Pass/Fail** softkey to enter the PASS/FAIL function menu.
4. Press the **Enable Test** softkey to select **On** to enable the pass/ fail test.
5. Press the **Mask Setting** softkey to enter the MASK function menu.
6. Press the **Location** softkey to select **Internal**.
7. Press the **Load** softkey to recall the saved interval memory.
8. A few second later, it will pop-out the message “**Read Data Success!**” and the saved mask will be displayed on the screen.

➤ **Save Test Mask to External Memory**

Do the following steps to save the test mask to external memory:

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Pass/Fail** softkey to enter the PASS/FAIL function menu.
4. Press the **Enable Test** softkey to select **On** to enable the pass/ fail test.
5. Press the **Source** softkey to select the desired channel.
6. Press the **Mask Setting** softkey to enter the MASK function menu.
7. Press the **X Mask** or **Y Mask** softkey; and then turn the **Universal Knob** to select the desired value.
8. Press the **Location** softkey to select **External**
9. Press the **Save** softkey to enter the file SAVE/RECALL system.
10. Save the test mask file refer to the “**Save and Recall**” chapter.

➤ **Recall Test Mask from External Memory**

Do the following steps to save the test mask f external memory:

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Pass/Fail** softkey to enter the PASS/FAIL function menu.
4. Press the **Enable Test** softkey to select **On** to enable the pass/ fail test.
5. Press the **Mask Setting** softkey to enter the MASK function menu.
6. Press the **Location** softkey to select **External**
7. Press the **Load** softkey to enter the file SAVE/RECALL system.
8. Select the desired test mask file with a RGU postfix using the **Universal Knob**; and then press the **Load** softkey.

IO Set

SDS2000 provides abundant IO interfaces, including: USB Device, LAN and Aux Output.

To Set the USB Device

Do the following steps to set the oscilloscope to communicate with PC via USB:

1. Install the USBTMC device driver on PC. Suggest you install NI Vista.
2. Connect the oscilloscope with PC using a standard USB cable
3. Press the **Utility** button on the front panel to enter the UTILITY function menu.
4. Press the **IO Set** softkey to enter the I/O SET function menu.
5. Press the **USB Device** softkey to select **USBTMC**.
6. Use EasyScopeX or NI Vista to communicate with the oscilloscope.

Do the following steps to set the oscilloscope to print the screen image:

1. Connect the oscilloscope with print using a standard USB cable.
2. Press the **Utility** button on the front panel to enter the UTILITY function menu.
3. Press the **IO Set** softkey to enter the I/O SET function menu.
4. Press the **USB Device** softkey to select **Printer**.
5. Press the Print button on the front panel to print the screen image.

To Set the LAN

Do the following steps to set the oscilloscope to communicate with PC via LAN:

1. Connect the oscilloscope to your local area network using the network cable.
2. Press the **Utility** button on the front panel to enter the UTILITY function menu.
3. Press the **IO Set** softkey to enter the I/O SET function menu.
4. Press the **LAN** softkey to enter the LAN setting interface, see the picture below:

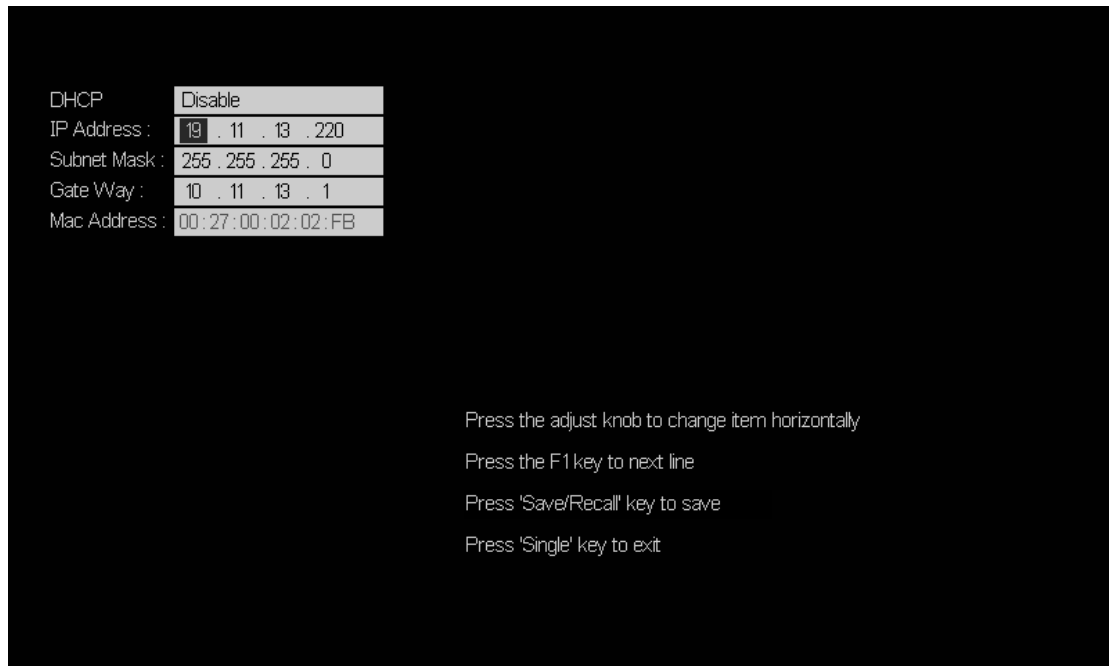


Figure 99 LAN Setting Interface

5. Press the F1 softkey (the first softkey below the screen) continually to go to the DHCP line; then turn the **Universal Knob** to select Enable or Disable.
 - **Enable:** the DHCP server in the current network will assign the network parameters (such as the IP address) for the oscilloscope.
 - **Disable:** you can set the IP address, subnet mask, gate way manually.
 - Turn the **Universal Knob** to select the desired value.
 - Push the **Universal Knob** to change item horizontally.
 - Press the **F1** softkey to go to the next line.
 - Press the **Save/Recall** softkey to save the current setting.
 - Press the **Single** button to exit the setting interface.
6. Use EasyScopeX or NI Vista to communicate with the oscilloscope.

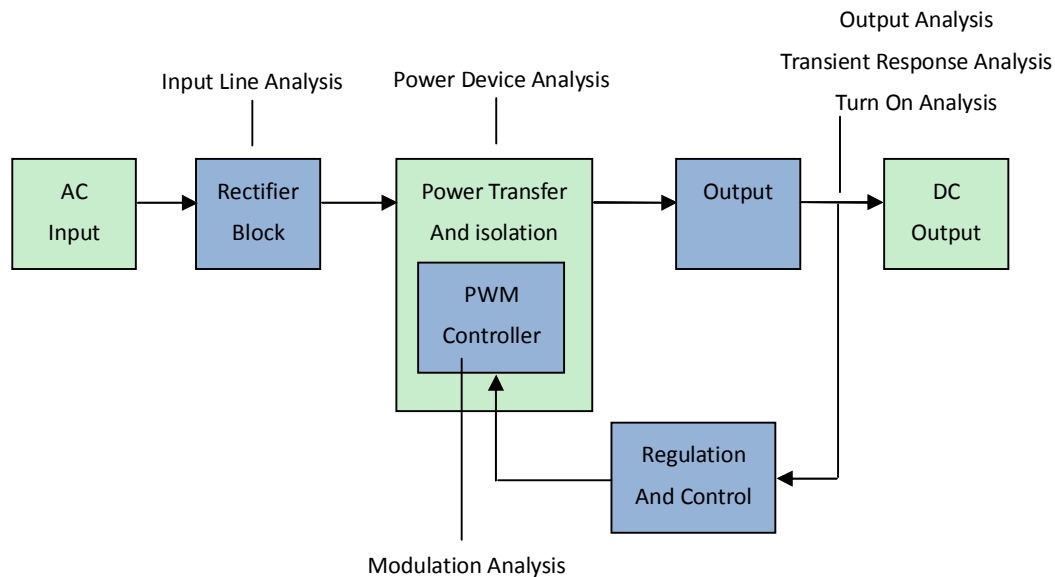
To Set Aux Output

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

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **IO Set** softkey to enter the I/O SET function menu.
3. Press the Aux Output softkey to select Trig Out or Pass/Fail. The default setup is **Trig Out**.
 - **Trig Out:** if this type is selected, the oscilloscope outputs a signal that can reflect the current capture rate of the oscilloscope at each trigger. The maximal capture rate of SDS2000 is 110k wfs/s.
 - **Pass/Fail:** if this type is selected, the oscilloscope will output a pulse signal when failed waveforms are detected. This signal can be connected to other control systems to conveniently view the test results.

To Do Power Analysis

The Power Analyze application for the SDS2000 serials oscilloscopes lets you quickly and easily analyze switching power supply efficiency and reliability.



Switch-Mode Power Supply (SMPS) Block Diagram and Types of Measurements

With the Power Analyze application, you can:

- Measure switching loss and conduction loss at the switching device.
- Analyze di/dt slew rate.
- Automate oscilloscope setup for ripple measurements.
- Perform pre-compliance testing to IEC 61000-3-2 standards.
- Analysis line power with total harmonic distortion, true power, apparent power, power factor tests.
- Measure output noise.
- Analysis modulation using the on-time information of a Pulse Width Modulation (PWM) signal.
- Measure how well a circuit rejects ripple coming from the input power supply at various frequencies with the Power Supply Rejection Ratio (PSRR) measurement.

The Power Analyze application license, along with the oscilloscope, high-voltage differential probe, current probe, probe deskew fixture, and passive probe, form a complete power measurement system for power supply design and testing.

Perform Channel Deskew

To make accurate power loss measurements, you must perform current and voltage channel deskew using the DF2001A deskew fixture. The channel deskew procedure calibrates the time delay between current and voltage probes. This step is certainly important since a small delay in the timing of the voltage and current traces can cause a large error in the instantaneous power reading.

The deskew accuracy is a function of the electrical characteristics of the voltage and current probes as well as the rise time of the signal under test. In general, higher bandwidth probes will allow better rise time fidelity and better deskew resolution.

You need to perform the deskew procedure once initially, and you should re-run the procedure when any part of the hardware setup changes (for example, a different probe, different oscilloscope channel, etc.) or when the ambient temperature changes.

To perform the channel deskew, do the following steps:

1. First, demagnetize and zero-adjust the current probe.
2. Make connections to the DF2001A deskew fixture:

	Small Loop	Large Loop
For current probes:	CP4060 (100KHz, 60A) CP4200 (150KHz, 200A) CP5050 (40MHz , 50A)	CP5300 (12MHz, 300A)
Connect high-voltage differential probe to either:	J2: differential measuring terminal J7: signal terminal J6: earth terminal	J1: differential measuring terminal J3: signal terminal J4: earth terminal

- a. Connect D+ and D- of the high-voltage differential probe to the deskew fixture.
- b. Connect the current probe to the loop with the direction of the arrow pointing towards the current flow.
- c. Make sure the switch on the deskew fixture is set to the appropriate side of the fixture (either “small loop” or “large loop”).
- d. Connect the deskew fixture to a USB port on your oscilloscope or a PC using a USB cable. The USB port supplies power to the deskew fixture.

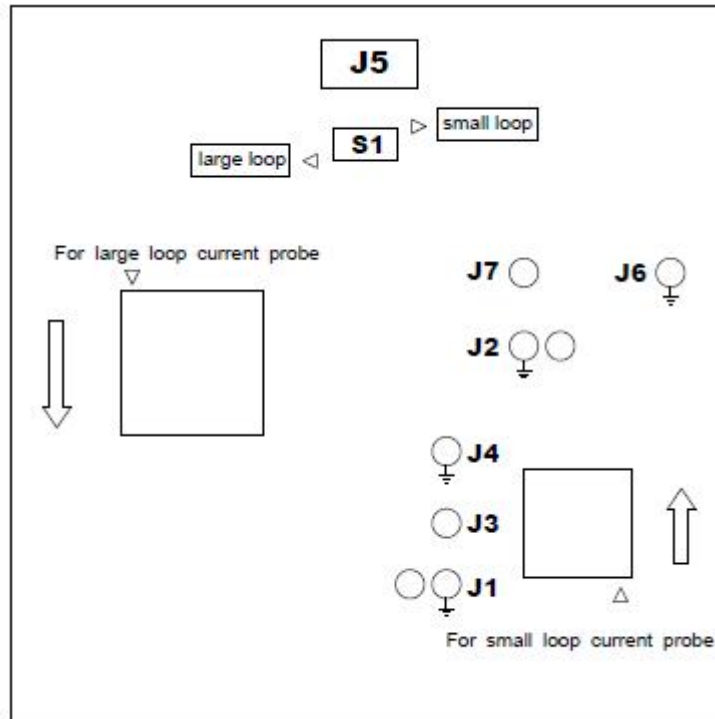


Figure 100 DF2001A Deskew Fixture

3. Press the **Utility** on the front panel to enter the UTILITY function menu.
4. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.

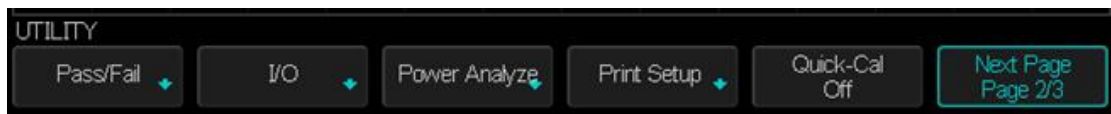


Figure 101 Second Page of the UTILITY Function Menu

5. Press the Power Analyze softkey to enter the ANALYSIS function menu.

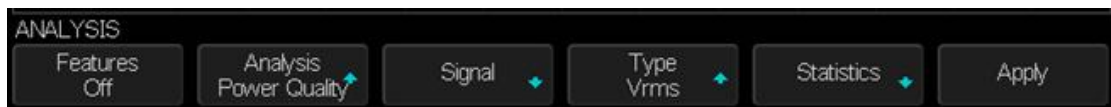


Figure 102 UTILITY Function Menu

6. Press the Signal softkey to enter the SIGNAL setting menu.



Figure 103 Power Analysis SIGNAL Menu

7. Press the **Voltage** softkey, and then turn the universal knob to select the analog channel probing the voltage signal.
8. Press the **Current** softkey, and then turn the universal knob to select the analog channel probing the current signal.
9. Press the **Deskew** softkey to enter the DESKEW function menu.



Figure 104 DESKEW Menu

10. Press the **Auto Deskew** softkey. When the process completes, you see a message indicating whether the deskew was successful, and if so, the settings being used.

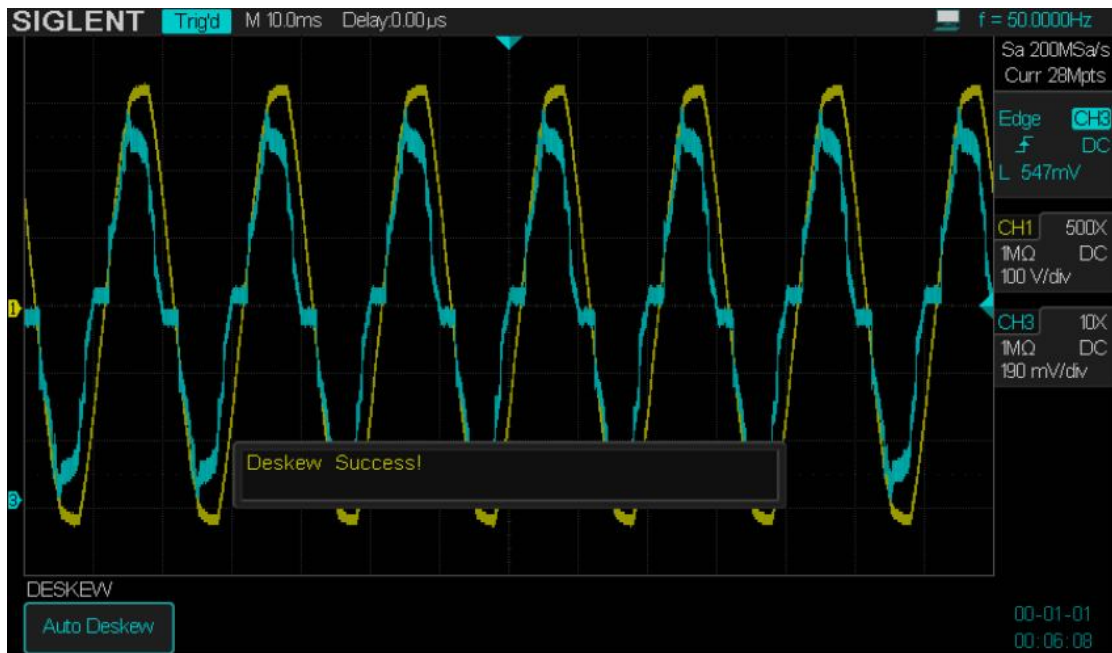


Figure 105 Deskew Success

General, you need to perform the deskew again when part of the test setup changes (for example, a different oscilloscope probe, different channel, etc.) or when the ambient temperature has changed.

Perform Power Analysis

This chapter describes the types of power analysis you can perform with the Power Analyze application, the proper probing connections to the device under test, signal setup, settings, and results.

Power Analysis application mainly contains the following items:

- Power Quality
- Harmonics
- Inrush
- Switching
- Slew Rate
- Modulation
- Output Ripple
- Turn On/Off
- Transient
- Efficiency

Power Quality

The Power Quality analysis shows the quality of the AC input line.

Some AC current may flow back into and back out of the load without delivering energy. This current, called reactive or harmonic current, gives rise to an “apparent” power which is larger than the actual power consumed. Power quality is gauged by these measurements: RMS voltage, RMS current, power factor, apparent power, true power, reactive power, crest factor, and phase angle of the current and voltage of the AC line.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Power Quality** and press down the knob to confirm.



Figure 106 ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect D+ of the voltage probe to the live wire of the AC input.
 - b. Connect D- of the voltage probe to the neutral wire of the AC input.
 - c. On the voltage probe, select the appropriate attenuation ratio.
 - d. Connect the current probe to the live wire of the AC input with the direction of the arrow pointing towards the current flow.
 - e. Connect the voltage and current probes to the desired oscilloscope channels.
6. Press the **Signal** softkey to enter the SIGNAL menu.



Figure 107 Power Quality SIGNAL Menu

7. Press the **Voltage** and **Current** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Cycles** softkey, and then turn the universal knob to select the desired number of cycles to capture in one acquisition.
10. Press the **Auto Setup** softkey to automatically scale and position the voltage and current channels and set the appropriate time/div.
11. Press the **UP** button to return to the ANALYSIS function menu.
12. Press the **Type** softkey, and then turn the universal knob to select the type of measurement to make in the power quality analysis:
 - **Real Power**: the portion of power flow that averaged over a complete cycle of the

AC waveform, result in net transfer of energy in one direction.

- **Apparent Power:** the portion of power flow due to stored energy, which returns to the source in each cycle.
- **Reactive Power:** the difference between apparent power and real power due to reactance.
- **Voltage Crest Factor:** ratio between the instantaneous peak voltage required by the load and the RMS voltage (RMS stands for Root Mean Square, which is a type of average).
- **Current Crest Factor:** ratio between the instantaneous peak current required by the load and the RMS current (RMS stands for Root Mean Square, which is a type of average).
- **Phase Angle:** In the power triangle (the right triangle where $\text{apparent_power}^2 = \text{real_power}^2 + \text{reactive_power}^2$), phase angle is the angle between the apparent power and the real power, indicating the amount of reactive power.

13. Press the **Apply** softkey to perform analysis to perform power quality analysis.



Figure 108 Perform Power Quality Analysis

The voltage and current waveforms are displayed as well as the input power waveform (waveform math multiply of the voltage and current). Also displayed are the power quality measurements you have selected and applied. There are at most displaying 4 measurements at a time.

14. Press the **Statistics** softkey to enter the STATISTIC menu.



Figure 109 STATISTIC Menu

15. Press the **Display** softkey to select **On** or **Off** to turn on or off the statistic function.

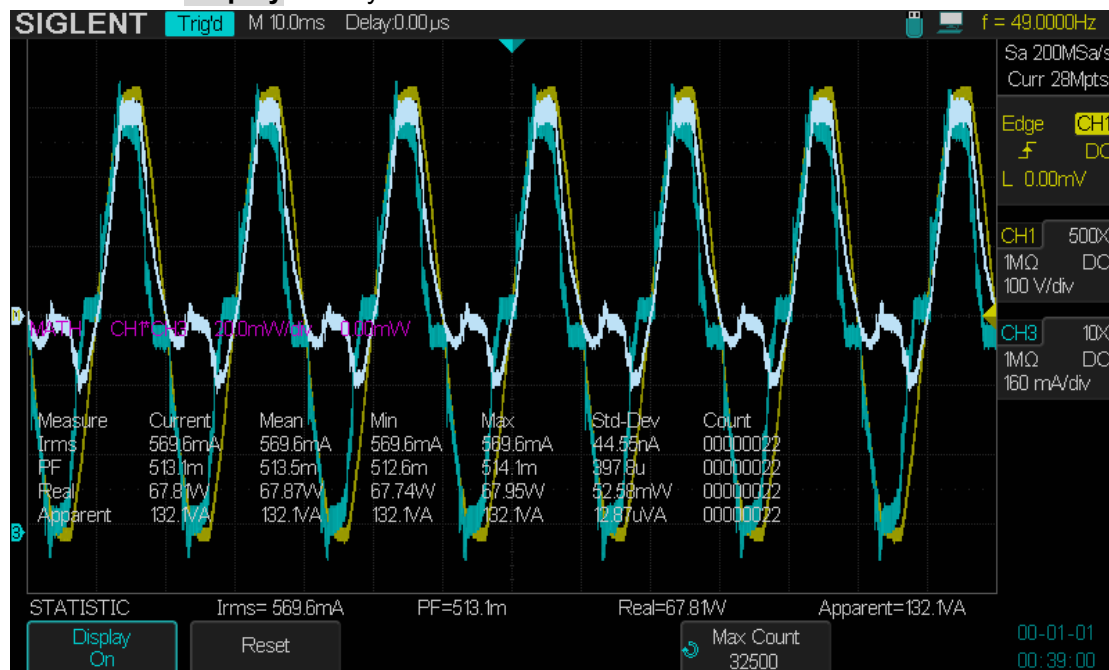


Figure 110 Turn on Statistic Function

16. Press the **Rest** softkey to clear the count value.

17. Press the **Max Count** softkey, and then turn the universal knob to specify the maximal count value.

Current Harmonics

Switching power supplies draw a range of harmonics from the AC mains. Standard limits are set for these harmonics can travel back to the supply grid and cause problems with other devices on the grid.

Use the current harmonics analysis to test a switching power supply's current harmonics to pre-compliance standard of IEC61000-3-2(Class A, B, C or D). The analysis present up to 40 harmonics.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Harmonics** and press down the knob to confirm.



Figure 111 ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect D+ of the voltage probe to the live wire of the AC input.
 - b. Connect D- of the voltage probe to the neutral wire of the AC input.
 - c. On the voltage probe, select the appropriate attenuation ratio.
 - d. Connect the current probe to the live wire of the AC input with the direction of the arrow pointing towards the current flow.
 - e. Connect the voltage and current probes to the desired oscilloscope channels.
6. Press the **Signal** softkey to enter the SIGNAL menu.



Figure 112 Harmonics SINGAL Menu

7. Press the **Voltage** and **Current** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Cycles** softkey, and then turn the universal knob to select the desired number of cycles to capture in one acquisition.
10. Press the **Auto Setup** softkey to automatically scale and position the voltage and current channels and set the appropriate time/div.
11. Press the **UP** button to return to the ANALYSIS function menu.
12. Press the **Settings** softkey to enter the SETTINGS menu.



Figure 113 Harmonics SETTING Menu

13. Press the **Line Freq** softkey, and then turn the universal knob to select the appropriate line frequency 50Hz, 60Hz or 400Hz.
14. Press the **Standard** softkey, the turn the universal knob to select the desired standard to perform compliance testing on the current harmonics.
 - IEC 6100-3-2 Class A – for balanced three- phase equipment, household appliances (except equipment identified as Class D), tools excluding portable tools, dimmers for incandescent lamps, and audio equipment.
 - IEC 6100-3-2 Class B – for portable tools.
 - IEC 6100-3-2 Class C– for lighting equipment.
 - IEC 6100-3-2 Class D – for equipment having a specified power according less than or equal to 600W, of the following types: personal computers and personal computer monitors, television receivers.
15. Press the **UP** button to return to the ANALYSIS function menu.
16. Press the **Apply** Softkey to perform the harmonics analysis. When the analysis has completed, results are displayed.

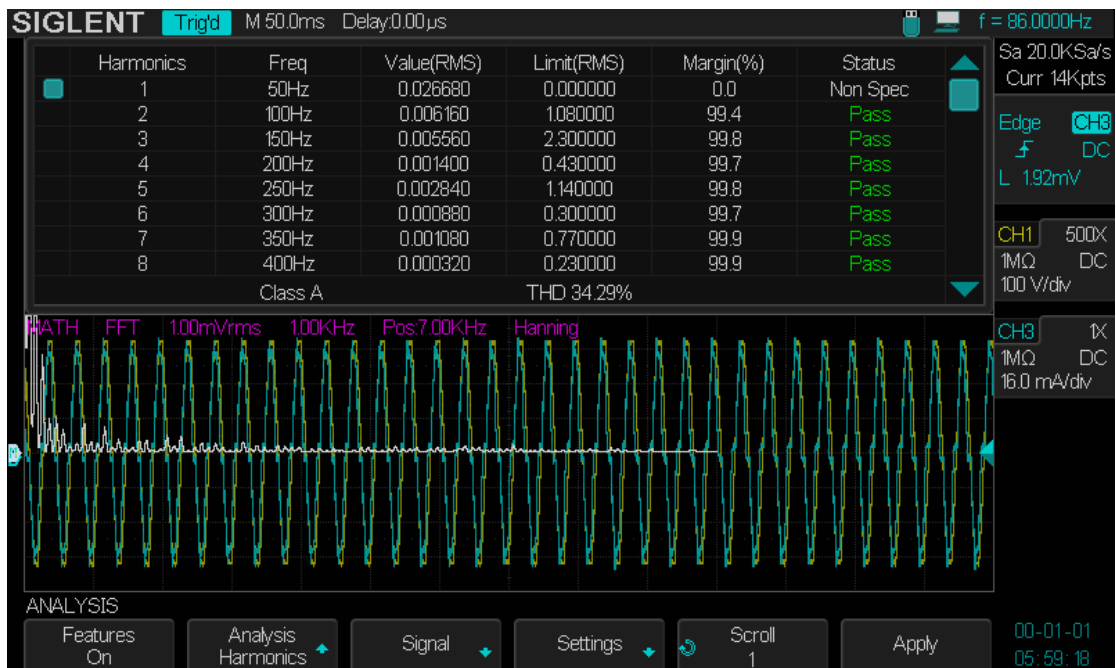


Figure 114 Perform Harmonics Analysis

FFT waveform	Shows the frequency components in the input current. The FFT is calculated using the Hanning window.
Harmonic, Value (RMS), Limit(RMS), Margin, Status	<p>For the first 40 harmonics, these values are displayed:</p> <ul style="list-style-type: none"> ● Value (RMS): the measured value in the units specified by the Harmonics Unit parameter. ● Limit (RMS): the limit specified by the selected Current Harmonics Standard parameter. ● Margin: the margin specified by the selected Current Harmonics Standard parameter. ● Status: whether the value passes or fails according to the selected Current Harmonics Standard. <p>Marginal results are greater than 85% of the limit but less than 100% of the limit.</p>
THD (Total Harmonics Distortion)	$\text{THD} = 100 \times \frac{\sqrt{X_2^2 + X_3^2 + X_4^2 + \dots + X_n^2}}{X_1}$ <p>Where :</p> <p>X_n = voltage or current of each harmonic X_1 = fundamental voltage or current value</p>

After the analysis has been performed, you can see the **Scroll** option under the Harmonics ANALISIS function menu. Press the **Scroll** softkey, and then turn the universal knob to scroll current harmonics analysis results.

Inrush

The inrush current analysis measures the peak inrush current of the power supply when the power supply is first turned on.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Inrush** and press down the knob to confirm.



Figure 115 ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect D+ of the voltage probe to the live wire of the AC input.
 - b. Connect D- of the voltage probe to the neutral wire of the AC input.
 - c. On the voltage probe, select the appropriate attenuation ratio.
 - d. Connect the current probe to the live wire of the AC input with the direction of the arrow pointing towards the current flow.
 - e. Connect the voltage and current probes to the desired oscilloscope channels.
6. Press the **Signal** softkey to enter the SIGNAL menu.



Figure 116 Inrush Analysis SIGNAL Menu

7. Press the **Voltage** and **Current** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Expected** softkey, and then turn the universal knob to specify the expected inrush current amplitude. This sets the vertical scale of the channel probing current.
10. Press the **Max Vin** softkey, and then turn the universal knob to specify the maximum input voltage. This sets the vertical scale of the channel probing voltage.
11. Press the **UP** button to return to the ANALYSIS function menu.
12. Press the **Apply** softkey to perform the analysis. Follow the onscreen instructions showing as below:
 - a. Turn off the power supply, then press Next.
 - b. Turn on the power supply, then press Next.



Figure 117 Perform Inrush Analysis

When the analysis has completed, results are displayed at the bottom of the screen.

- **Peak Current:** the peak current can be a positive or negative value, so the result is larger of the measured maximum or minimum.
The peak current measurement is made on one source input that is the current waveform.

Switching

The switching loss analysis calculates the power dissipated in the switching cycle across the switching device. Typical power losses include:

- Switching loss that occur during switching of V_{ds} and I_d .
- Conduction losses that occur when the switching device (MOSFET) is ON.

Design engineers use this information to improve the power conversion efficiency of the power supply.

Switching loss is also used to quantify the power loss that is transferred to the heat sink of the power device.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Switching** and press down the knob to confirm.



Figure 118 ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect D+ of the voltage probe to the source of the MOSFET.
 - b. Connect D- of the voltage probe to the drain of the MOSFET.
 - c. On the voltage probe, select the appropriate attenuation ratio.
 - d. Connect the current probe to the I drain of the MOSFET with the direction of the arrow pointing towards the current flow.
 - e. Connect the voltage and current probes to the desired oscilloscope input channels.
6. Press the **Signal** softkey to enter the SIGNAL menu.



Figure 119 Switching Analysis Signal Menu

7. Press the **Voltage** and **Current** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Auto Setup** softkey to automatically scale and position the voltage and current channels and set the appropriate time/div.
10. Press the **UP** button to return to the ANALYSIS function menu.
11. Press the **Settings** softkey to enter the SETTINGS menu.

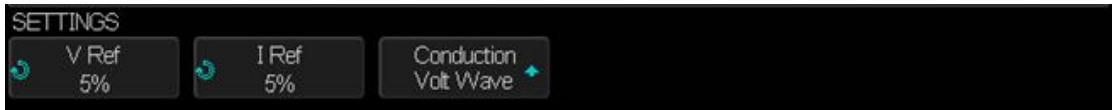


Figure 120 Switching Analysis SETTINGS Menu

12. Press the **V Ref** softkey, and then turn the universal knob to select the desired value.
 - Enter the switching level for the switching edges. The value is in percentage of the maximum switch voltage.
You can adjust this value to ignore noise floors.
This value specifies the threshold that is used to determine the switching edges.
13. Press the **I Ref** softkey, and then turn the universal knob to select the desired value.
 - Enter the switching level for the start of switching edges. The value is in percentage of the maximum switch current.
You can adjust this value to ignore noise floors or null offset that is difficult to eliminate in current probes.
This value specifies the threshold that is used to determine the switching edges.
14. Press the **Conduction** softkey, and then turn the universal knob to select the way to calculate conduction.
 - Volt Waveform: the power waveform uses the original data, and the calculation is :
 $P = V \times I$
 - Rds (on): the power waveform includes error correction:
 - In the on zone (where the voltage level is below V Ref), the power calculation is: $P = I_{d2} \times R_{ds} (on)$
Specify **Rds (on)** using the additional softkey.
 - In the off zone (where the current level is below I Ref), the power calculation is: $P = 0$ Watta.
 - Vce (sat): the power waveform includes error correction:
 - In the on zone (where the voltage level is below V Ref), the power calculation is: $P = V_{ce} (sat) \times I_c$
Specify **Vce (sat)** using the additional softkey.
 - In the off zone (where the current level is below I Ref), the power calculation is: $P = 0$ Watta.
15. Press the **UP** button to return to the ANALYSIS function menu.
16. Press the **Apply** softkey to perform the analysis. When the analysis has completed, results are displayed.



Figure 121 Perform Switching Analysis

The voltage and current waveforms are displayed, as well as the power waveform (waveform math multiply of the voltage and current). Also displayed are the Power Loss and Energy Loss measurements.

- **Power Loss:** $P_n = V_{ds_n} * I_{d_n}$, where n is in each sample. The power loss measurement is made on one source input that represents power. This is typically a math multiply waveform of the voltage and current waveform.
- **Energy Loss:** $E_n = \sum (V_{ds_n} * I_{d_n}) * \text{sample size}$, where n is in each sample. The energy loss measurement is made on one source input that represents power. This is typically a math multiply waveform of the voltage and current waveforms.

Slew Rate

The slew rate analysis measures the rate of the voltage or current change during switching.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Slew Rate** and press down the knob to confirm.



Figure 122 Slew Rate ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect D+ of the voltage probe to the source of the MOSFET.
 - b. Connect D- of the voltage probe to the drain of the MOSFET.
 - c. On the voltage probe, select the appropriate attenuation ratio.
 - d. Connect the current probe to the I drain of the MOSFET with the direction of the arrow pointing towards the current flow.
 - e. Connect the voltage and current probes to the desired oscilloscope input channels.
6. Press the **Signal** softkey to enter the SIGNAL menu.



Figure 123 Slew Rate Signal Menu

7. Press the **Voltage** and **Current** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Auto Setup** softkey to automatically scale and position the voltage and current channels and set the appropriate time/div.
10. Press the **UP** button to return to the ANALYSIS function menu.
11. Press the **Source** softkey to select either Voltage or Current as the source for the slew rate analysis.
12. Press the **Apply** softkey to perform the analysis. When the analysis has completed, results are displayed.



Figure 124 Perform Slew Rate Analysis

The voltage and current waveform are displayed along with a differentiate math function waveform that shows the slew rate.

Max and Min measurements on the differentiate math function waveform are added and displayed.

- Voltage slew rate: $dV/dt = [y_{(n)} - y_{(n-1)}] / [x_{(n)} - x_{(n-1)}]$, measures the slew rate of Vds of the power device (MOSFET).
- Current slew rate: $di/dt = [y_{(n)} - y_{(n-1)}] / [x_{(n)} - x_{(n-1)}]$, measures the slew rate of Id of the power device (MOSFET).

Modulation

The modulation analysis measures the control pulse signal to a switching device (MOSFET) and observes the trending of the average, RMS-AC, period, frequency, positive width, negative width, duty cycle, rise time and fall time of the control pulse signal in response to different events.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Modulation** and press down the knob to confirm.



Figure 125 Modulation ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect D+ of the voltage probe to the gate of the MOSFET.
 - b. Connect D- of the voltage probe to the source of the MOSFET.
 - c. On the voltage probe, select the appropriate attenuation ratio.
 - d. Connect the current probe to the I drain of the MOSFET with the direction of the arrow pointing towards the current flow.
 - e. Connect the voltage and current probes to the desired oscilloscope input channels.
6. Press the **Signal** softkey to enter the SIGNAL menu.

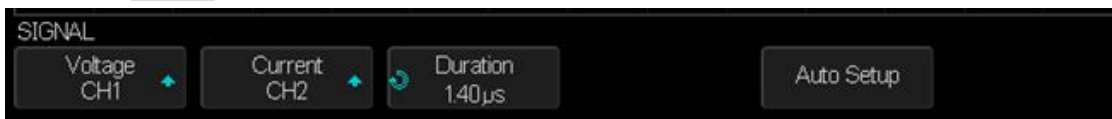


Figure 126 Modulation SIGNAL Menu

7. Press the **Voltage** and **Current** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Duration** softkey, and then turn universal knob to specify the time to capture signals. This sets the time scale of the oscilloscope.
10. Press the **Auto Setup** softkey to automatically scale and position the voltage and current channels and set the appropriate time/div.
11. Adjust the trigger level to capture waveforms at the same place in each cycle (in other words, stabilize the waveform display).
12. Press the **UP** button to return to the ANALYSIS function menu.
13. Press the **Source** softkey to select either **Voltage** or **Current** as the source for the

modulation analysis.

14. Press the **Type** softkey, and then turn the universal Knob to select the type of measurement to make in the modulation analysis.
 - Average
 - RMS –AC
 - Period
 - Frequency
 - Pos Width
 - Neg Width
 - Duty Width
 - Rise Time
 - Fall Time
15. Press the **Apply** softkey to perform the analysis. When the analysis has completed, results are displayed. There are at most displaying 4 sets of measurements at the same time. Measurements are displayed at the bottom of the screen.



Figure 127 Perform Modulation Analysis

The measurement trend math waveform visualization is used to plot the measurement variation for each cycle of the modulation waveforms.

Output Ripple

The output ripple analysis measures the ripple noise of the power supply output.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Output Ripple** and press down the knob to confirm.



Figure 128 Output Ripple ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect the voltage probe (passive or differential) to the DC output of the power supply.
 - b. Connect the voltage probe to an oscilloscope input channel.
6. Press the **Signal** softkey to enter the SIGNAL menu.

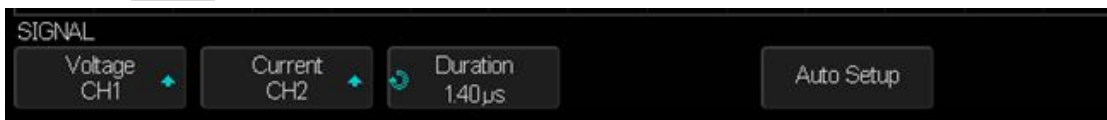


Figure 129 Output Ripple SIGNAL Menu

7. Press the **Voltage** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factor is set in the oscilloscope for the voltage probe.
9. Press the **Duration** softkey, and then turn universal knob to specify the time to capture signals. This sets the time scale of the oscilloscope.
10. Press the **Auto Setup** softkey to automatically scale and position the voltage and current channels and set the appropriate time/div.
11. Adjust the trigger level to capture waveforms at the same place in each cycle (in other words, stabilize the waveform display).
12. Press the **UP** button to return to the ANALYSIS function menu.
13. Press the **Apply** softkey to perform the analysis. When the analysis has completed, results are displayed.

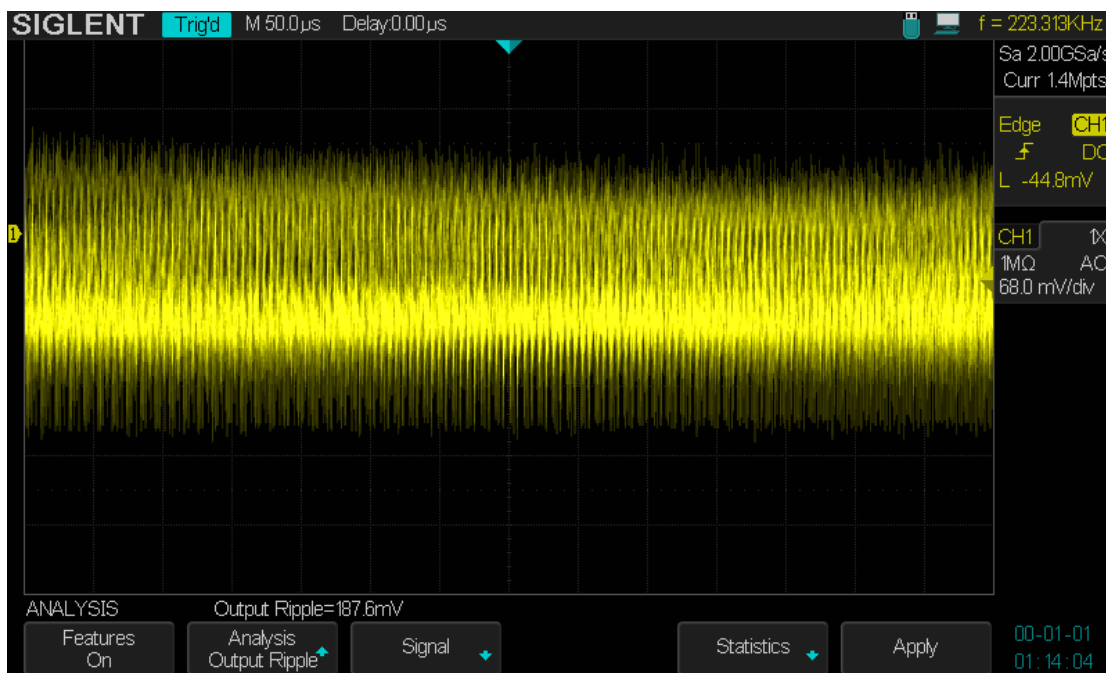


Figure 130 Perform Output Ripple Analysis

The output voltage waveform is displayed along with this automatic power measurement: Output Ripple: $V_{Max} - v_{mIN}$. The output ripple measurement is made on one source input that is the output voltage waveform.

14. Press the **Statistics** softkey to enter the STATISTIC menu.



Figure 131 Output Ripple STATISTIC Menu

15. Press the **Display** softkey to select **On** or **Off** to turn on or off the statistic function.



Figure 132 Turn on Statistic Function

16. Press the **Rest** softkey to clear the count.
17. Press the **Max Count** softkey, and then turn the universal knob to specify the maximal count value.

Turn On/ Off

The turn on analysis determines how fast a turned on power supply takes to reach 90% of its steady state output.

The turn off analysis determines how fast a turn off power supply takes to reduce its output voltage to 10% of maximum.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Turn On/Off** and press down the knob to confirm.



Figure 133 Turn On/Off ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect D+ of the voltage probe to the live wire of the AC input.
 - b. Connect D- of the voltage probe to the neutral wire of the AC input.
 - c. On the voltage probe, select the appropriate attenuation ratio.
 - d. Connect the output voltage probe (passive or differential) to the DC output of the power supply.
 - e. Connect the voltage probes to the desired oscilloscope channels.
6. Press the **Signal** softkey to enter the SIGNAL menu.



Figure 134 Turn On/Off SIGNAL Menu

7. Press the **Input V** and **Output V** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Duration** softkey, and then turn the universal knob to select the time scale of the measurement.
10. Press the **Max Vin** softkey, and then turn the universal knob to specify the maximum input voltage.
Enter the maximum (peak to peak) source voltage amplitude. The source voltage will be used to trigger the oscilloscope in “Turn On Time” test.
This value is used to adjust the vertical scale of the channel probing the oscilloscope input voltage.
11. Press the **Steady Vout** softkey, and then turn the universal knob to specify the expected steady state output DC voltage of the power supply.

This value is used to adjust the vertical scale of the channel probing the oscilloscope output voltage.

12. Press the **UP** button to return to the ANALYSIS function menu.
13. Press the **Test** softkey to select whether turn on or turn off analysis is performed:
 - **Turn On**: measure the time taken to get the output voltage of the power supply after the input voltage is applied.
 - **Turn Off**: measure the time taken for the output voltage of the power supply to turn off after the input voltage is removed.
14. Press the **Apply** softkey to perform turn on/off analysis.

Follow the instruction on screen. When the analysis has completed, results are displayed.

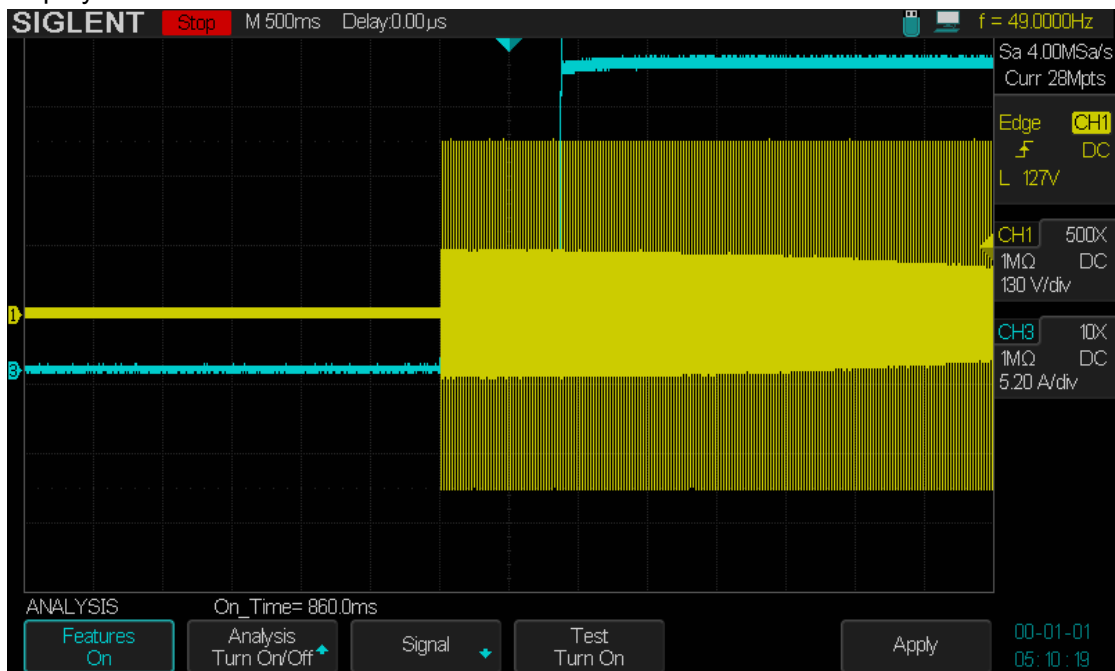


Figure 135 Perform Turn On/Off Analysis

The input and output voltage waveforms are displayed. Also displayed is this automatic power measurement:

- Turn On Time = $t_2 - t_1$, where:
 - t_1 = AC input voltage rise to 10% of its maximum amplitude (Start Time).
 - t_2 = DC output voltage rise to 90% of its maximum amplitude (End Time).
- Turn Off Time = $t_2 - t_1$, where:
 - t_1 = AC input voltage goes below 10% of its positive peak (or negative peak whichever occurs first) (start time).
 - t_2 = DC output voltage drops to 10% of its steady state value (End Time).

Transient

The transient response analysis determines how fast a power supply's output voltage responds to change at the output load. This time is from when the output voltage first exits the setting band to when it last enters the setting band.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Transient** and press down the knob to confirm.



Figure 136 Transient ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect the voltage probe (passive or differential) to the DC output of the power supply.
 - b. Connect the voltage probe to an oscilloscope input channel.
 - c. On the voltage probe, select the appropriate attenuation ratio.
 - d. Connect the current probe to the output load of the power supply.
The change in the load current will be used to trigger the oscilloscope to capture the transients.
 - e. Connect the current probe to an oscilloscope input channel.
6. Press the **Signal** softkey to enter the SIGNAL menu.

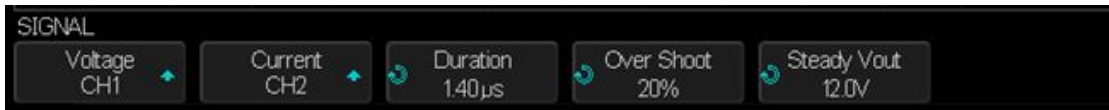


Figure 137 Transient SIGNAL Menu

7. Press the **Voltage** and **Current** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Duration** softkey, and then turn the universal knob to select the time scale of the measurement.
10. Press the **Over Shoot** softkey, and then turn the universal knob to specify the % of overshoot of the output voltage.
This value is used to determine the setting band value for the transient response and adjust the vertical scale of the oscilloscope.
11. Press the **Steady Vout** softkey, and then turn the universal knob to specify the expected steady state output DC voltage of the power supply.
This value is used to along with the overshoot percentage to specify the setting band for the transient response and to adjust the vertical scale of the oscilloscope.
12. Press the **UP** button to return to the ANALYSIS function menu.

13. Press the **Settings** softkey to enter the SETTINGS menu.



Figure 138 Transient SETTING Menu

14. Press the **Initial I** softkey, and then turn the universal knob to specify the initial load current value. The initial load current will be used as a reference and to trigger the oscilloscope.
15. Press the **New I** softkey, and then turn the universal knob to specify the new load current value. The new load current will be used as a reference and to trigger the oscilloscope.
16. Press the **UP** button to return to the ANALYSIS function menu.
17. Press the **Apply** softkey to perform turn on/off analysis.

Follow the instruction on screen. When the analysis has completed, results are displayed.

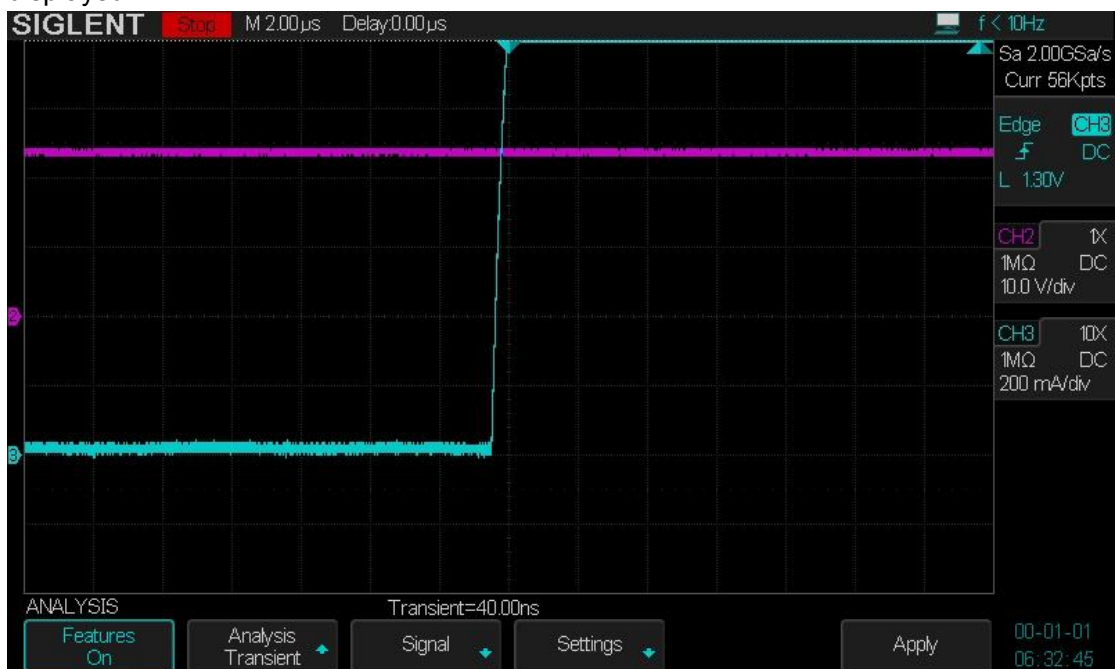


Figure 139 Perform Transient Analysis

The voltage and current waveforms are display. Also displayed is the transient measurement, which displaying at the bottom of the screen.

Transient response time = $t_2 - t_1$, where:

- t_1 = The first time a voltage waveform exits the setting band.
- t_2 = The last time it enters into the setting band.
- Setting band = +/- overshoot% of the steady state output voltage.

Efficiency

Efficiency analysis tests the overall efficiency of the power supply by measuring the output power over the input power.

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to enter the second page of the UTILITY function menu.
3. Press the **Power Analyze** softkey to enter the ANALYSIS function menu.
4. Press the **Analysis** softkey, and then turn the universal knob to select **Efficiency** and press down the knob to confirm.



Figure 140 Efficiency ANALYSIS Function Menu

5. Connect your probes to the device under test and to the oscilloscope as well.
 - a. Connect D+ of the input voltage probe to the live wire of the AC input.
 - b. Connect D- of the input voltage probe to the neutral wire of the AC input.
 - c. On the input voltage probe, select the appropriate attenuation ratio.
 - d. Connect the input current probe to the live wire of the AC input with the direction of the arrow pointing towards the current flow.
 - e. Connect D+ of the output voltage probe to the input path of the load.
 - f. Connect D- of the output voltage probe to the return path of the load.
 - g. On the output voltage probe, select the appropriate attenuation ratio.
 - h. Connect the output current probe to the input path of the load with the direction of the arrow pointing towards to the current flow.
 - i. Connect the voltage and current probes to the desired oscilloscope channels.
6. Press the **Signal** softkey to enter the SIGNAL menu.



Figure 141 Efficiency SIGNAL Menu

7. Press the **Input V**, **Input I**, **Output V** and **Output I** softkeys and make sure the proper analogy channels are selected.
8. Make sure the proper probe attenuation factors are set in the oscilloscope for the voltage and current probes.
9. Press the **Duration** softkey, and then turn the universal knob to specify the time to capture signals. This sets the time scale of the oscilloscope..
10. Press the **Auto Setup** softkey to automatically scale and position the voltage and current channels and set the appropriate time/div.
11. Press the **UP** button to return to the ANALYSIS function menu.
12. Press the **Apply** Softkey to perform the efficiency analysis. When the analysis has completed, results are displayed.



Figure 142 Perform Efficiency Analysis

The input voltage, input current, output voltage and output current waveforms are displayed, as well as the input power waveform (waveform math multiply of the input voltage and current). All displayed are these automatic power measurements:

- **Input Power = Input V * Input I**
The input power measurement is made using two source inputs, the input voltage waveform and the input current waveform, and it also requires a math multiply waveform of the voltage and current waveforms.
- **Output Power = Output V * Output**
The output power measurement is made using two source inputs, the output voltage waveform and the output current waveform, and it also require a math multiply waveform of the voltage and current waveforms.
- **Efficiency = Input Power/ Output Power**
The efficiency measurement is made on one source input that represents input power. This is typically a math multiply waveform of the input voltage and input current waveforms. This measurement also requires the output voltage waveform and the output current waveform specified in signals setup for the efficiency power analysis.

To Print the Screen Image

SDS2000 supports PictBridge compatible printers. You can print the screen image of the oscilloscope.

1. Connect a USB printer to either the USB host on the front panel or the USB host device on the rear panel.
2. Press the **Utility** button on the front panel to enter the UTILITY function menu.
3. Press the **IO Set** softkey to enter the I/O SET function menu.
4. Press the **USB Device** softkey to select **Print**.
5. Press the **UP** softkey to return to the UTILITY function menu.
6. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
7. Press the **Print Setup** softkey to enter the PRINT function menu.
8. Press the **InkSaver** softkey to select **On** or **Off**. The default setup is **Off**.
 - **On**: to reduce the amount of black ink it takes to print screen image by changing the black background to white.
 - **Off**: to print the color screen image as it is.
9. Press the **Layout** softkey to select the desired output direction of the print.
10. Press the **PaperSize** softkey to select the desired paper size.
11. Press the **Image Size** softkey to select the desired image size.
12. Press the **Print Key** to select Print **Picture**.
 - **Print Picture**: select the option when the oscilloscope connects to the printer; press the Print button on the front panel to print the screen image.
 - **Save Picture**: select the option when the oscilloscope connects with a USB flash driver; press the Print button on the front panel to save the screen image to the USB flash driver with a .BMP postfix.
13. Press the **Next Page** softkey to go to the second page of the PRINT function menu.
14. Press the **ID Print** softkey to select the whether to print the screen image with an ID.
15. Press the **Print** button on the front panel to perform.

To Enable Quick-Cal

Quick-Cal can calibrate the measure deviation which cause by the temperature. If the temperature that the SDS2000 is working is not stable, do the following steps:

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Quick-Cal** to select **On**.

If the Quick-Cal has been set to **On** before powering on, and the current vertical scale is 2mV or 5mV of any channel, the oscilloscope will perform the quick calibration procedure when it powers on. During the calibration, most of the buttons are disabled.

Also, when the Quick-Cal is **On** and the current vertical scale is 2mV or 5mV of any channel, the oscilloscope will perform the quick calibration procedure if the operating temperature changes by more than 2 degree.

To Update Firmware and Configuration

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

Do the following steps to update the firmware:

1. Insert the USB flash driver which with the firmware and the configure files inside into the USB host interface on the front panel of the oscilloscope.
2. Press the **Utility** button on the front panel to enter the UTILITY function menu.
3. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
4. Press the **Next Page** softkey to go to the third page of the UTILITY function menu.
5. Press the **Update** softkey to enter the UPDATE function menu.
6. Press the **Firmware** softkey and it will pop-out the message “**Press ‘Single’ to continue and press ‘Run/Stop’ to exit!**”
7. Press the **Single** button to enter the SAVE/RECALL file system.
8. Turn the **Universal Knob** to select the update file which should be with an ADS postfix; and then press the **Load** softkey to start update the firmware. The process needs about 7 minutes. And during the update, do not cut off the oscilloscope, otherwise the oscilloscope will not restart again.
9. After finish the update, the screen will pop-out the message “**Update success, please restart the DSO**”, and you will heard a sound of buzzer.
10. Restart the oscilloscope to finish the firmware update.

Do the following steps to update the configuration:

1. Insert the USB flash driver which with the firmware and the configure files inside into the USB host interface on the front panel of the oscilloscope.
2. Press the **Utility** button on the front panel to enter the UTILITY function menu.
3. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
4. Press the **Next Page** softkey to go to the third page of the UTILITY function menu.
5. Press the **Update** softkey to enter the UPDATE function menu.
6. Press the **Configure** softkey to enter the SAVE/RECALL file system.
7. Turn the **Universal Knob** to select the update file which should be with an CFG postfix; and then press the **Load** softkey to start update the firmware. The process needs about 30 seconds.
8. After finish the update, the screen will pop-out the message “**Update success, please restart the DSO**”, and you will heard a sound of buzzer.
9. Restart the oscilloscope to finish the configuration update.

Do Self Test

Self tests include screen test, keyboard test, and LED test. Self tests used to test the screen, buttons, knobs and LED lights whether works well.

Screen Test

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Next Page** softkey to go to the third page of the UTILITY function menu.
4. Press the **Do Self Test** softkey to enter the SELFTEST function menu.
5. Press the **Screen Test** softkey to enter the screen test interface, as the picture shown below, the screen display pure red.

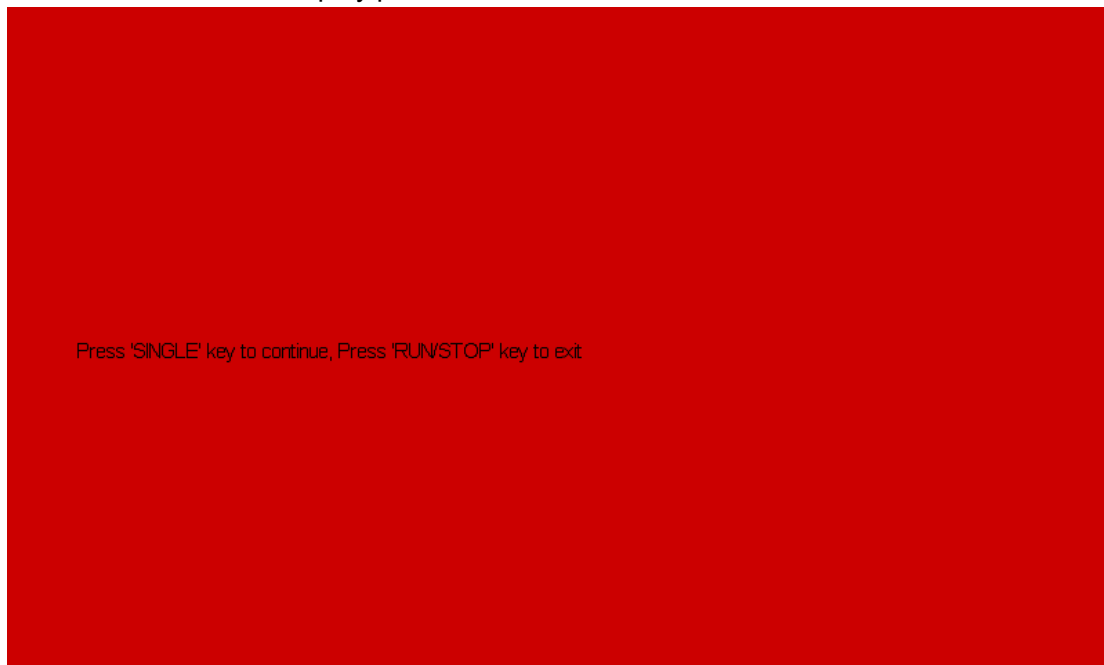


Figure 143 Screen Test

6. Press the **Single** button on the front panel continually as it says in the picture above. The screen displays green, blue and red again. It is easy to check chromatic aberration, stain and scratch of the screen under the condition.
7. Press the **Run/Stop** button on the front panel to exit the screen test program.

Keyboard Test

Keyboard test is used to test that if the keys or the knobs work well.

Do the following steps to do keyboard test:

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

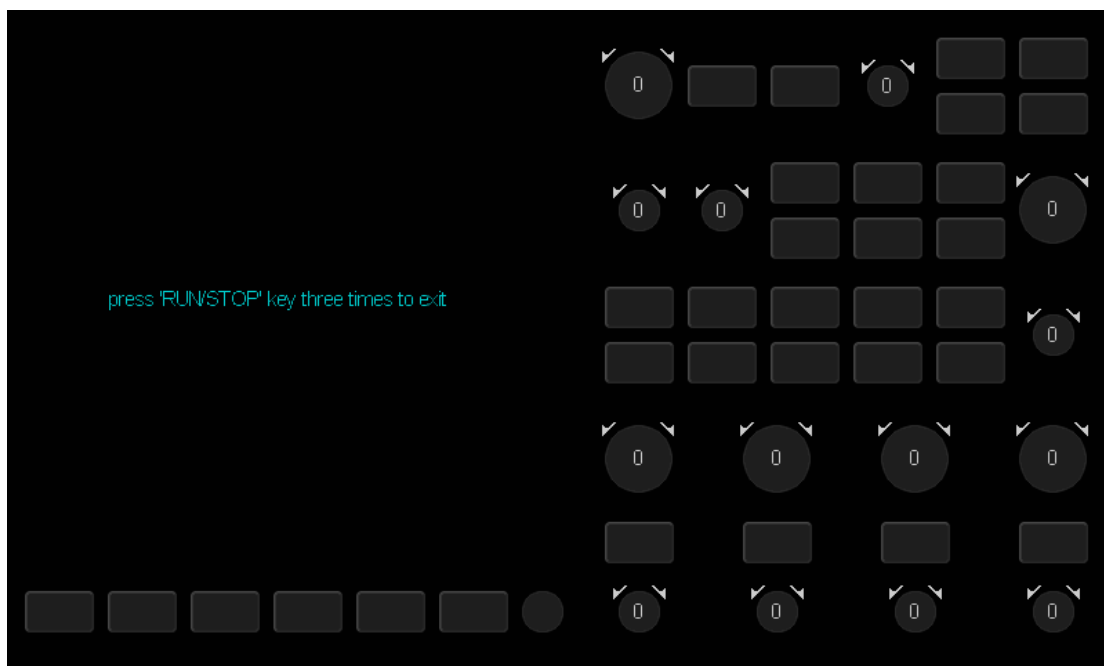


Figure 144 Keyboard Test

6. To perform the knobs and the buttons test.
Knobs test: the default value is 0. Turn left to increase the value while turn right to decrease; push the knob to set the value to 0. Test every knob randomly.
Keys test: the first time to press the key to light it up, and a second press to die out. Test every button randomly.
7. Press the **Run/Stop** button 3 times to exit the keyboard test program.

LED Test

LED test is used to test that if the button lights work well.

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



Figure 145 LED Test

6. According to the prompting information displaying on the screen, press the **Single** button continually to light the button lights one by one. The first time to press the **Single** button, the **Run/Stop** button displays red, and the second press the **Run/Stop** button displays green. Then the other button lights will be lighted one by one. At last all the lights will be lighted at the same time.
7. Press the **Run/Stop** button to exit the LED test program.

To specify Screen Saver Time

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

Do the following steps to set the screen saver time:

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Next Page** softkey to go to the third page of the UTILITY function menu.
4. Press the **Screen Saver** softkey; and then turn the **Universal Knob** to select the desired screen saver time. The screen saver time can be set to **1min**, **5min**, **10min**, **30min**, and **1hour**. Also you can select **Off** to turn off the screen saver function



Figure 146 Screen Saver Interface

5. Press the any button on the front to exit the screen saver program.

Option Management

SDS2000 provides multiple options to fulfill your measurement requirements. Please contact your **SIGLENT** sales representative or **SIGLENT** 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.

Do the following steps to install the option on the oscilloscope:

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Next Page** softkey to go to the third page of the UTILITY function menu.
4. Press the **Options** softkey to enter the OPTION function menu.



Figure 147 OPTION Function Menu

5. Press the **Type** softkey; then turn the **Universal Knob** to select the module to be installed and push down the knob to confirm.
6. Press the **License** softkey to enter the LABEL function menu to input the license.



Figure 148 LABEL Function Menu

- When the **Switch To** Softkey is set to **Keyboard**:
 - Turn the **Universal Knob** or press the **←** softkey and the **→** softkey to select character; Push down the **Universal Knob** or press the **Enter** softkey to input the character.
 - Press the **Delete** softkey to delete the incorrect character.
- When the **Switch To** softkey is set to **License**:

- Turn the **Universal Knob** or press the **←** softkey and the **→** softkey to select character;
- Push down the **Universal Knob** or press the **Enter** softkey to confirm the characters and the oscilloscope will activate the corresponding option using the license key currently input.
- Press the **Delete** softkey to delete the incorrect character.

7. Press the Information softkey to view the option information.

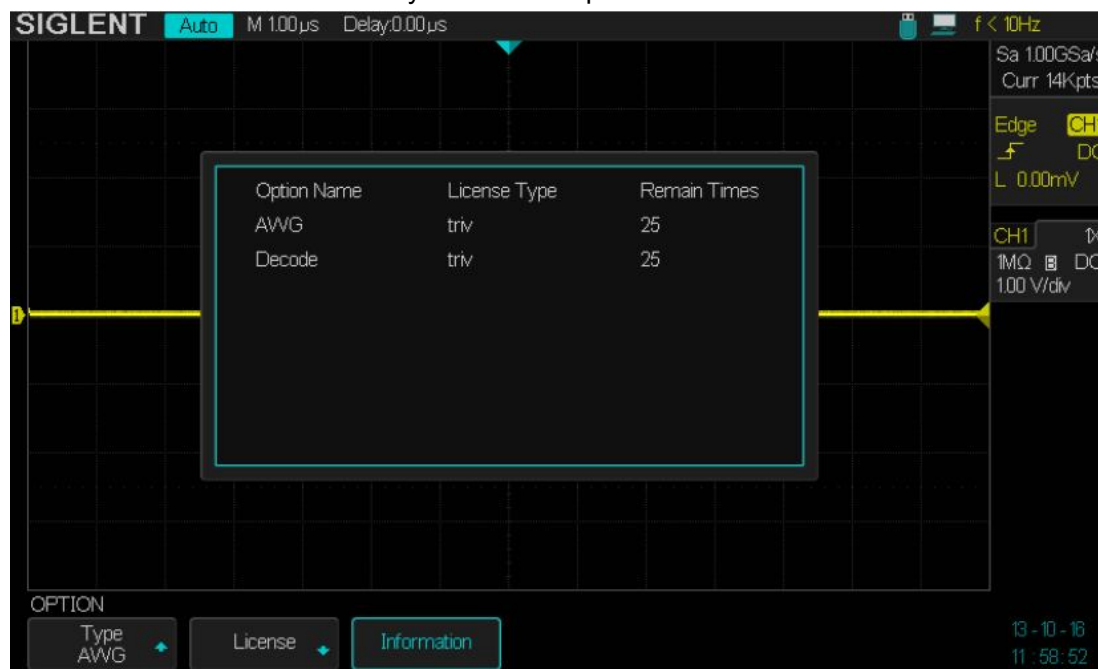


Figure 149 Option Information

To Set and Display Data /Time

SDS2000 supports real time display.

Do the following steps to set and display time:

1. Press the **Utility** button on the front panel to enter the UTILITY function menu.
2. Press the **Next Page** softkey to go to the second page of the UTILITY function menu.
3. Press the **Next Page** softkey to go to the third page of the UTILITY function menu.
4. Press the **Date/Time** softkey to enter the DATE/TIME function menu.



Figure 150 DATE/TIME Function Menu

5. Press the **Year** softkey; then turn the **Universal Knob** to select the desired value.
6. Press the **Month/Day** softkey once; then turn the **universal knob** to select the desired month. Press the **Month/Day** softkey for a second time; and then turn the **Universal Knob** to select the desired day.
7. Press the **Hour/Minute** softkey once; then turn the **Universal Knob** to select the desired hour. Press the **Hour/Minute** softkey for a second time; then turn the **Universal Knob** to select the desired minute.
8. Press the **Display** softkey to select **On**, and the real time will display at the down-right corner of the display.
9. Press the **Confirm** softkey to confirm the current setting.

Arbitrary Waveform Generator

SDS2000 supports build-in Arbitrary waveform generator.

SDS2000 build-in Arbitrary waveform generation provides sine, square, ramp, pulse, DC, noise, cardiac, Gauss pulse, exp rise, exp fall and Arbitrary waveforms.

The contents of this chapter:

- ◆ To Set Wave Type and Parameters
- ◆ To Output Arbitrary Waveform
- ◆ To Set Output Load
- ◆ To Recover Default Setup
- ◆ To Do AWG Self Cal

To Set Wave Type and Parameters

1. Press the **WaveGen** button on the front panel to open the build-in Arbitrary waveform generation function. The first press lights up the backlight of the button, opens the output and enters the WAVEFORM function menu. A second press of the **WaveGen** button will turn off the backlight and the output.
2. Press the **Wave Type** softkey; then turn the **Universal Knob** to select the desired waveform and push down the knob to confirm. Press the Wave Type softkey continually can also select the waveform type.
3. Press the **Frequency** softkey; then turn the **Universal Knob** to select **Frequency**, **Frequency Fine**, **Period** or **Period Fine** and push down the knob to confirm; turn the **Universal Knob** to select the desired value. Use the same method the set the other parameters.

Wave Type	Character	Frequency Range	Amplitude (High-Z)	Offset (High-Z)
Sine	Use the Universal Knob to adjust the Frequency/ Frequency Fine/ Period/ Period Fine/ Amplitude/ Amplitude Fine/ High-Level/ High-Level-Fine/ Offset/ Offset Fine/ Low-Level/ Low-Level-Fine value.	1uHz~25MHz	4mV~6V	± 3.0V
Square	Use the Universal Knob to adjust the Frequency/ Frequency Fine/ Period/ Period /Fine/ Amplitude/ Amplitude Fine/ High-Level/ High-Level-Fine/ Offset/ Offset Fine/ Low-Level/ Low-Level-Fine/ Duty value.	1uHz~10MHz	4mV~6V	± 3.0V
Ramp	Use the Universal Knob to adjust the Frequency/ Frequency Fine/ Period/ Period Fine/ Amplitude/ Amplitude Fine/ High-Level/ High-Level-Fine/ Offset/ Offset Fine/ Low-Level/ Low-Level-Fine/ Symmetry value.	1uHz~300KHz	4mV~6V	± 3.0V
pulse	Use the Universal Knob to adjust the Frequency/ Frequency Fine/ Period/ Period /Fine/ Amplitude/ Amplitude Fine/ High-Level/ High-Level-Fine/	1uHz~10MHz	4mV~6V	± 3.0V

	Offset/Offset Fine/ Low-Level/ Low-Level-Fine/ Width/ Width Fine value.			
DC	Use the Universal Knob to adjust the Offset value.			± 3.0V
Noise	Use the Universal Knob to adjust the Stdev/ Mean value.			
Cardiac	Use the Universal Knob to adjust the Frequency/ Frequency Fine/ Period/ Period /Fine/ Amplitude/ Amplitude Fine/ High-Level/ High-Level-Fine/ Offset/Offset Fine/ Low-Level/ Low-Level-Fine value.	1uHz~5MHz	4mV~6V	± 3.0V
Gaus Pulse	Use the Universal Knob to adjust the Frequency/ Frequency Fine/ Period/ Period /Fine/ Amplitude/ Amplitude Fine/ High-Level/ High-Level-Fine/ Offset/Offset Fine/ Low-Level/ Low-Level-Fine value.	1uHz~5MHz	4mV~6V	± 3.0V
Exp Rise	Use the Universal Knob to adjust the Frequency/ Frequency Fine/ Period/ Period /Fine/ Amplitude/ Amplitude Fine/ High-Level/ High-Level-Fine/ Offset/Offset Fine/ Low-Level/ Low-Level-Fine value.	1uHz~5MHz	4mV~6V	± 3.0V
Exp Fai;	Use the Universal Knob to adjust the Frequency/ Frequency Fine/ Period/ Period /Fine/ Amplitude/ Amplitude Fine/ High-Level/ High-Level-Fine/ Offset/Offset Fine/ Low-Level/ Low-Level-Fine value.	1uHz~5MHz	4mV~6V	± 3.0V
When the Output Load is set to 50Ω , all the value will be the half.				

To Output Arbitrary Waveform

1. Press the **Wave Gen** button on the front panel to enable the AWG function and enter the WAVEFORM function menu.
2. Connect the SDS2000 with PC which has installed the EasyWave software using a USB cable.
3. Double click the EasyWave icon to run it.
4. Select a waveform file or draw an arbitrary waveform; then click the **Send wave** option which at the bottom of the interface and it will pop-up the dialogue which shows as the picture below:

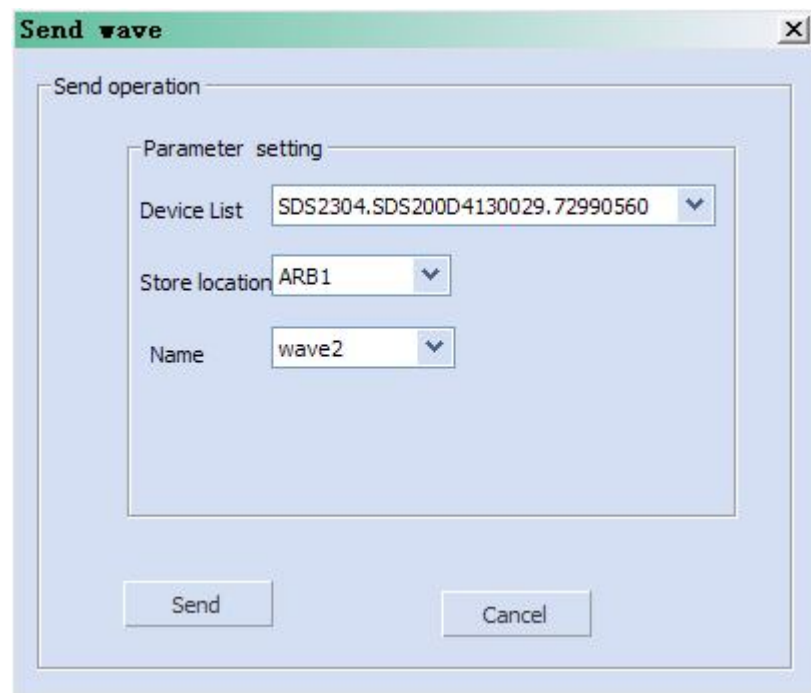


Figure 151 Send Wave Dialogue

5. Click the Store location area and select the location to keep the arbitrary waveform.
6. Click the Send option, then the oscilloscope will pop-out the message “Dealing with waveform data, please wait...”
7. A few seconds later the oscilloscope will prompt “Succeed to save waveform”, and it will enter the WAVEFORM function menu, the Wave Type sets to ArbX (the location chose above)



Figure 152 Save Waveform Successfully

8. If you want to delete the arbitrary waveform, press the **Delete** softkey.
9. Press the **Setting** softkey to enter the SETTING function menu.
10. Press the **Output Load** softkey to select **High-Z** or **50Ω**.

To Set Output Load

11. Press the **Wave Gen** button on the front panel to enter the WAVEFORM function menu.
12. Press the **Setting** softkey to enter the SETTING function menu.
13. Press the **Output Load** softkey to select **High-Z** or **50Ω**.

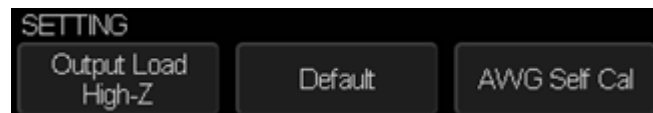


Figure 153 SETTING Function Menu

The output load of the Wave Gen must match with the oscilloscope, or the amplitude of the signal will be incorrect.

Recover the Default Values of Wave Gen.

1. Press the **Wave Gen** button on the front panel to enter the WAVEFORM function menu.
2. Press the **Setting** softkey to enter the SETTING function menu.
3. Press the **Default** softkey to recover the default values of Wave Gen (Sine, 1.000 KHz, 4.000Vpp, 0.0mVdc).

AWG Self Cal

If the operating temperature changes more than 5° and the instrument runs more than more 30 minutes, you should do the AWG Self Cal to decrease the mistake.

1. Press the **Wave Gen** button on the front panel to enter the WAVEFORM function menu.
2. Press the **Setting** softkey to enter the SETTING function menu.
3. Press the **AWG Self Cal** softkey to perform AWG self calibration program.





Figure 154 AWG Self Cal

4. When the self calibration program is finish, press the **Run/Stop** button on the front panel to exit the interface according to the information on the screen.

Default Setup

Press the Default button on the front to set the SDS2000 to the leave factory setup. The details show as below.

Horizontal	
T/div	1 μ S/div
Delay	0 S
Zoom	Off
Format	Y-T
Vertical	
Channel on/off	CH1
V/div	1 V/div
Offset	0 V
Coupling	DC
BW Limit	Full
Adjust	Coarse
Probe	1X
Impedance	1 M Ω
Unit	V
Invert	Off
Acquire	
Acquisition	Normal
Sinx/x	Sinx
Mem Depth	14 Mpts
Trigger	
Type	Edge
Source	CH1
Slope	Rising
Holdoff	Off
Coupling	DC
Noise Rejest	Off
Mode	Auto
Display	
Type	Vectors
Color	Off

Persist	Off
Grid	
Intensity	50%
Brightness	40%
Transparence	50%
Cursor	
Mode	Off
Type	X1
Source	CH1
X1	-3.5 μ s
X2	3.5 μ s
Save/Recall	
Type	Setups
Save To	Internal
Setup	NO.1
Utility	
IO Set	
USB Device	USBTMC
Aux Output	Trig Out
Sound	
Sound	On
Pass/Fail	
Enable Test	Off
Source	CH1
Operate	Off
Mes Display	Off
X Mask	0.2
Y Mask	0.2
Location	Internal
Fail To Stop	Off
Output	
System Setup	
Quick-Cal	Off
Screen Saver	30min
Math	

Operate	Off
+	
Source A	CH1
Source B	CH1
Invert	Off
V/div	1.00 V/div
offset	0V
-	
Source A	CH1
Source B	CH1
Invert	Off
V/div	1.00 V/div
offset	0 V
*	
Source A	CH1
Source B	CH1
Invert	Off
V/div	1.00 V ² /div
offset	0 V ²
/	
Source A	CH1
Source B	CH1
Invert	Off
V/div	1.0/div
offset	0
FFT	
Source	CH1
Window	Hanning
Horizontal	1X
Vertical Scale	20 dBVrms
Display	Split
Horizontal Scale	100MHz
d/dt	
Source	CH1
Vertical Scale	1.00 (MV/S)/div
Vertical Offset	0
dx	0.2 div
[dt	
Source	CH1
Offset	0

Vertical Scale	1.00 μ V/div
Vertical Offset	0
√	
Source	CH1
Vertical Scale	1.00 V ^{1/2} /div
Vertical Offset	0
REF	
Source	CH1
Location	REF A
Display	Off
Decode	
Serial 1	
Serial	I2C
Display	Off
List	Off
Serial 2	
Serial	SPI
Display	Off
List	Off
I2C	
SCK	CH1
Threshold	1.60V
SDK	CH2
Threshold	1.60V
Address	7 位
SPI	
CLK	CH1
Threshold	1.60V
Edge Select	Rising
MISO	CH2
Threshold	1.60V
MOSI	CH3
Threshold	1.60V
CS Type	CS
CS	CH4
Idle Level	Low
Bit Order	LSB
Data Length	8

UART/RS232	
RX	CH1
Threshold	1.60V
TX	CH2
Threshold	1.60V
Bound	9600
Parity Check	None
Stop Bit	1
Idle Level	Low
Data Length	8
CAN	
CAN-H	CH1
Threshold	1.60V
CAN-L	CH2
Threshold	1.60V
Bound	100kb/S
Decode Source	CAN_H
LIN	
Source	CH1
Threshold	1.60V
Baud	2400
Wave Gen	
Function	Off
Wave Type	Sine
Output Load	High-Z
Sine	
Frequency	1 KHz
Amplitude	4 Vpp
Offset	0 Vdc
Square	
Frequency	1 KHz
Amplitude	4 Vpp
Offset	0 Vdc
Duty	50%
Ramp	
Frequency	1 KHz
Amplitude	4 Vpp
Offset	0 Vdc
Symmetry	50%

Pulse	
Frequency	1 KHz
Amplitude	4 Vpp
Offset	0 Vdc
Width	200 μ s
DC	
Offset	0.0mVdc
Noise	
Stdev	448 mV
Mean	0 mV
Cardiac	
Frequency	1 KHz
Amplitude	4 Vpp
Offset	0 Vdc
Gaus Pulse	
Frequency	1 KHz
Amplitude	4 Vpp
Offset	0 Vdc
Exp Rise	
Frequency	1 KHz
Amplitude	4 Vpp
Offset	0 Vdc
Exp Fall	
Frequency	1 KHz
Amplitude	4 Vpp
Offset	0 Vdc

Troubleshooting

The commonly encountered failures and their solutions are listed below. When you encounter those problems, please solve them following the corresponding steps. If the problem remains still, please contact **SIGLENT** Company as soon as possible.

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 use the specified fuse.
 - 4) Restart the instrument after finishing the above inspections.
 - 5) If it still does not work correctly, please contact **SIGLENT**.

 2. **The signal is sampled but no waveform of the signal is displayed:**
 - 1) Check whether the probe is correctly connected to the signal connecting wire.
 - 2) Check whether the signal connecting wire is correctly connected to the BNC (namely channel connector).
 - 3) Check whether the probe is correctly connected to the item to be tested.
 - 4) 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).
 - 5) Resample the signal.

 3. **The tested voltage amplitude is greater or lower than the actual value (note that this problem usually occurs when probe is used):**

Check whether the attenuation coefficient of the channel complies with the attenuation ratio of the probe.

 4. **There is waveform display but not stable:**
 - 1) Check the trigger signal source: check whether the **Source** item at the 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) Change the trigger holdoff setting.

 5. **No display after pressing Run/Stop:**

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

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

6. **The display of waveform is ladder-like:**

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

7. **Fail to connect PC or PictBridge printer through USB:**

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

8. **The USB storage device cannot be recognized:**

- 1) Check whether the USB storage device can work normally.
- 2) Make sure whether the USB interface can work normally.
- 3) Make sure that the USB storage device being used is flash storage type. This oscilloscope does not support hardware storage type.
- 4) Make sure whether the system of the USB storage device is FAT32.
- 5) Restart the instrument and then insert the USB storage device to check it.
- 6) If the USB storage device still cannot be used normally, please contact **SIGLENT**.