



USER'S MANUAL

IDS01070 Series

2CH WLAN Digital Storage Oscilloscope

V1.0.2

Content

General Safety Summary	I
Brief Introduction to IDSO	III
Chapter 1 Quick Start	1
1.1 Connections and Indications	2
1.2 Set Connection Mode.....	3
1.3 Charging the battery.....	4
1.4 General Check	4
1.5 Probe Compensation	4
1.6 Function Check	6
1.7 Self Calibration.....	7
Chapter 2 PC Application	8
2.1 Install Software	9
2.2 Install Driver	12
2.3 Set Connection Mode with PC	18
2.4 The User's Interface.....	20
2.5 The Menu System	22
2.6 Setup the Oscilloscope	22
2.7 Set Horizontal System.....	23
2.7.1 Horizontal Setting	23
2.7.2 The Application of the X-Y Operation	24
2.8 Set Vertical System	26
2.9 Set Trigger System	28
2.9.1 Trigger Setting	28
2.9.2 Capturing a Single-Shot Signal	29
2.10 Math and FFT	30
2.10.1 MATH	30
2.10.2 Fast Fourier Transform Function	31
2.11 Save and Load	32
2.11.1 Save and load file	32
2.11.2 Set Reference.....	33
2.11.3 Save/Load Setup	34
2.12 Display System	34
2.13 Measure System	36
2.13.1 Scale measurement.....	36
2.13.2 Cursor Measure.....	37

2.13.3 Automatic Measurement	40
2.14 Acquisition Modes	42
2.15 Interpolation	42
2.16 Utility Function	44
2.16.1 Record.....	45
2.16.2 Pass/Fail.....	45
2.16.3 Self Calibration	47
2.16.4 Factory Setup	47
2.16.5 Language	47
2.17 Print And Print Preview	48
Chapter 3 IOS Application	49
3.1 User Interface	50
3.2 Menu System	52
3.3 Connect IDSO With Apple/Android	52
3.4 Set Horizontal System.....	53
3.4.1 Horizontal Setting	53
3.4.2 The Application of the X-Y Operation	54
3.5 Set Vertical System	54
3.6 Set Trigger System	56
3.7 Save and Load.....	57
3.7.1 Save/Load File	57
3.7.2 Save/Load Reference.....	57
3.7.3 Save/Load Setup	59
3.8 Display System	59
3.9 Measure Signal	59
3.9.1 Cursor.....	59
3.9.2 Measure	60
3.10 Acquire	61
3.10.1 Capture Mode.....	61
3.10.2 Interpolation.....	61
Chapter 4 Android Application.....	62
4.1 User Interface	62
4.2 Menu System	64
4.3 Connect IDSO with Android in direct mode	64
4.4 Set Vertical System	65
4.5 Set Trigger System	66
4.6 Measure Signal	66

Chapter 5 Appendix.....67

Appendix A: Specifications.....68

Appendix B: General Maintenance70

Appendix C: Accessories70

General Safety Summary

Review the following safety precautions carefully before operate the device to avoid any personal injuries or damages to the device and any products connected to it. To avoid potential hazards use the device as specified by this user's guide only.

- ◆ **To Avoid Fire or Personal Injury.**
- ◆ **Connect and Disconnect Properly.** Do not connect or disconnect probes or test leads while they are connected to a voltage source.
- ◆ **Connect and Disconnect Properly.** Connect the probe output to the measurement device before connecting the probe to the circuit under test. Disconnect the probe input and the probe reference lead from the circuit under test before disconnecting the probe from the measurement device.
- ◆ **Observe All Terminal Ratings.** To avoid fire or shock hazard, observe all ratings and markings on the product. Consult the product manual for further ratings information before making connections to the product.
- ◆ **Use Proper Probe.** To avoid shock hazard, use a properly rated probe for your measurement.
- ◆ **Avoid Circuit or Wire Exposure.** Do not touch exposed connections and components when power is on.
- ◆ **Do Not Operate with Suspected Failures.** If suspected damage occurs with the device, have it inspected by qualified service personnel before further operations.
- ◆ **Provide Proper Ventilation.** Refer to the installation instructions for proper ventilation of the device.
- ◆ **Do not operate in Wet/Damp Conditions.**
- ◆ **Do not operate in an Explosive Atmosphere.**
- ◆ **Keep Product Surfaces Clean and Dry.**

Safety Information

In order to ensure the correct using and the best efficient service, please carefully read the user's manual.

System Symbols

These symbols may appear in this manual or on the instrument.



Warning: "Warning" identifies conditions and actions that pose hazards to the users.



Caution: "Caution" identifies conditions and actions that may damage the product or other properties.



DANGER:
High Voltage



Refer to
the manual



Protective Conductor
Terminal



Chassis grand



Earth(grand) Terminal

System Terms

The following terms may appear on the instrument:

Danger: The term "Danger" is used in the manual to indicate that when you read this mark, personal injury may be caused to immediately.

Waning: The term "Danger" is used in the manual to indicate that when you read this mark, personal injury may not be caused to you immediately, but you need to be cautionary.

Notice: The term "Noise" is used in this manual to indicate that damages may be caused on this product or other properties.

Brief Introduction

The IDSO is WLAN dual-channel digital storage oscilloscope geared towards tablet computers, it's a small, lightweight, portable, battery powered, fully featured two channel oscilloscope. The oscilloscope is ideal for production test, research and design and all of the applications involving analog circuits test and troubleshooting, as well as education and training.

Instead of a built-in screen it uses your tablet iOS, Android or PC (Windows) to display the measurements. Data exchange between the tablet and the oscilloscope is via WLAN, there are no cables. Also, as there is no physical connection between the oscilloscope and the tablet, it is a very safe instrument to use. Due to the wireless principle, the oscilloscope and the 'screen' do not need to be at the same location.

The communication protocol between the oscilloscope and the tablet is freely available, so it is easy for third-party developers to develop applications for other purposes. Finally, it is equipped with a USB port for charging of the internal battery. If desired, it can also connect to a PC via the USB port, instead of wireless.

Model	Channels	Bandwidth	Sample Rate
IDSO1070A	2	70MHz	250Msa/s

Features

- ◆ Excellent industrial design-the same anodized aluminium casing as iPad, which is not only beautiful and tasteful but also greatly enhanced the hardness of aluminum alloy surface, and has good heat resistance and strong wearability.
- ◆ USB2.0 interface, USB power source required, easy to use.
- ◆ Li-ion rechargeable battery include
- ◆ DVM readouts
- ◆ 20 measurement functions, PASS/FAIL Check, be suitable for technical application.
- ◆ Waveform average, persistence, intensity, invert, addition, subtraction, multiplication, division, X-Y plot.
- ◆ Save waveform in the following: text file, jpg/bmp graphic file, MS excel/word file.
- ◆ FFT

Minimum system requirements

iOS (*)
 Android™ 4.0 (*)
 Windows XP or higher (*)
 Tablet or PC

Download the IDSO APP

iOS: Search for IDSO from **App Store**

Windows: 1. CD Disc

2. Download from Our Website

Android: 1. CD Disc

2. Download from Our Website

3. Search for IDSO1070 from Google Play



Chapter 1 Quick Start

In addition to the list of general features on the next page, this chapter describes how to do the following tasks:

[Connections and Indications](#)

[Set Connection Mode](#)

[Charging the battery](#)

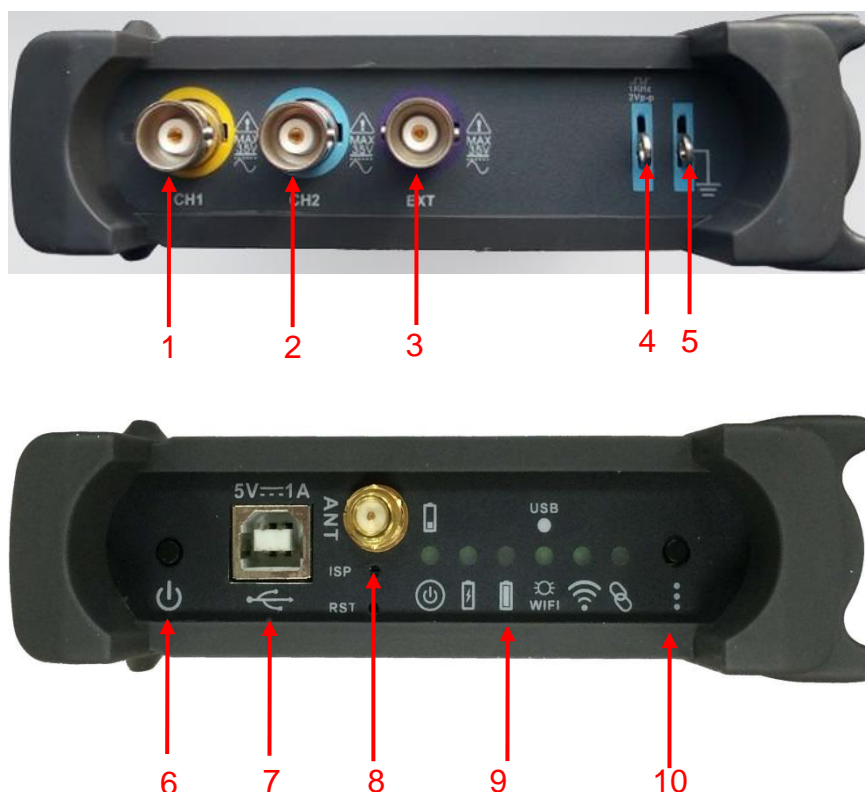
[General Check](#)

[Probe Compensation](#)

[Function Check](#)

[Self Calibration](#)

1.1 Connections and Indications



1. **CH 1:** Input connectors for waveform display.
2. **CH 2:** Input connectors for waveform display.
3. **EXT.:** Input connector for an external trigger source. Use the Trigger menu to select the Ext. source.
4. **CAL.:** Probe compensation output.
5. **GND.:** Ground terminal.
6. **Power Button:** Power on/off button.
7. **USB PORT:** Connect the B-Type Plug of USB cable to this port.
8. **ANT:** Antenna
9. **Indications**



Power on/off ①.



Battery charging



Battery full



USB/Wi-Fi indication



Hotspot, creates it's own wireless local area network (WLAN). Yellow light is always flashing.



When the device is connected with PC and communicates with PC successfully via USB cable, the light blinks in green for long.



Send signals when the connection is successful with the tablet/PC. ③



App connection indicates when there is connection over the WLAN network with the tablet application. ④

10. **Easylink:** Press this button to restart the IDSO and set Wi-Fi connection mode.

- ① *Power on for IDSO. Keep pressing power button until power indicator lights up. The indicator is blinking when it get battery level. When the battery is lower than 10%, the indicator is quick blinking. When the battery is not less than 10%, the indicator is slow blinking.*
- ② *The indicator keeps lighting in USB mode and slow blinking in Wi-Fi mode.*
- ③ *If there is data transfer between IDSO device and IDSO APP, this indicator is quick blinking. Otherwise, the indicator is off.*
- ④ *In Wi-Fi mode, if there is communication link between IDSO device and IDSO APP, the indicator keeps lighting normally.*

1.2 Set Connection Mode

There are two connection modes of the IDSO Wi-Fi mode and USB mode.

Wi-Fi mode: Keep pressing power button until power indicator lights up. Then the communication indicator USB/Wi-Fi lights up and slow blinking. You can connect the IDSO with tablet/PC via Wi-Fi.

USB mode: Keep pressing power button until power indicator lights up. And press Easy-Link button immediately, then the USB/Wi-Fi communication indicator lights up and keeping light. You can connect the IDSO with PC via USB cable.

Note:

1. In Wi-Fi mode: The IDSO device will power off when the time of no communication between IDSO device and IDSO application more than ten minutes. There are three situations about no communication:
 - 1) When the software of Windows is close.
 - 2) When iOS APP quits or works on background.
 - 3) When Android APP quits or the screen is lock.

2. In USB mode, if the driver is installed successfully on a PC, the software can communicate with IDSO automatically, so the IDSO won't power off when connect it with the PC. If the driver isn't installed successfully on a PC, there is no communication, so the IDSO will power off more than ten minutes.

1.3 Charging the battery

There are two ways to charge the battery:

1. Connect B-Type Plug of USB cable to IDSO's USB port. Connect the A-type Plug of USB cable to power adapter (5V, 1A).
2. Connect B-Type Plug of USB cable to IDSO's USB port. Connect the A-type Plug of USB cable to PC's USB port.

Note:

It is recommended to charge the rechargeable battery completely before using the oscilloscope for the first time.

When the battery is out of power, please use power adapter to charge it for a period of time before using.

1.4 General Check

Please check the instrument as following steps after receiving an oscilloscope:

Check 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 been checked mechanically and electrically.

Check the accessories:

Accessories supplied with the instrument are listed in "Accessories" in this guide. If the contents are incomplete or damaged, please notify the franchiser.

Check the instrument:

In case there is any mechanical damage or defect, or the instrument does not operate properly or fails performance tests, please notify the franchiser.

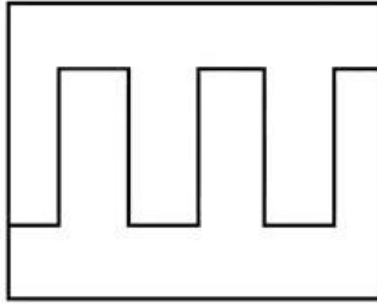
1.5 Probe Compensation

Perform this function to match the characteristics of the probe and the channel input. This should be performed whenever attaching a probe to any input channel at the first time.

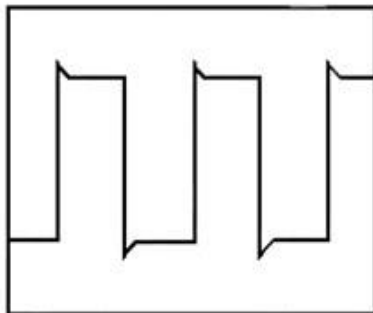
1. From the "Probe" menu, select attenuation to 1:10. Set the switch to "X10" on the probe and connect it to CH1 of the oscilloscope. When using the probe hook-tip, insert the tip onto the probe firmly to ensure a proper connection.
2. Connect the probe tip with a 1 KHz, 2 Vp-p square, select CH1, and then press the

“**AUTOSET**” button into the menu or the toolbar.

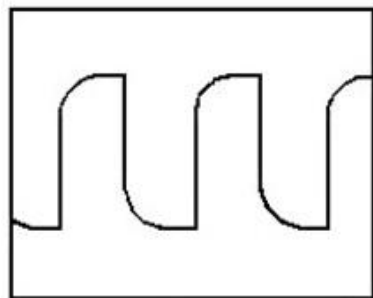
3. Check the shape of the displayed waveform.



Correctly Compensated

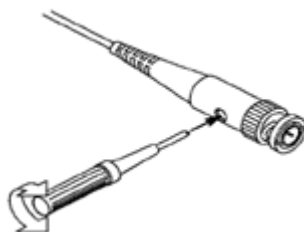


Over compensated



Under Compensated

4. If necessary, use a non-metallic tool to adjust the trimmer capacitor of the probe for the fattest square wave being displayed on the oscilloscope.



5. Repeat if necessary.

WARNING: To avoid electric shock while using the probe, be sure the perfection of the insulated cable, and do not touch the metallic portions of the probe head while it is connected with a voltage source.

1.6 Function Check

Perform this functional check to verify that your oscilloscope is operating correctly.

◆ Connect the oscilloscope with PC, iOS or Android

1. Connect IDSO with PC via USB cable or Wi-Fi. Please refer to [Connect IDSO with PC](#).
2. Connect IDSO with IOS via Wi-Fi. Please refer to [Connect IDSO WITH APPLE](#).
3. Connect IDSO with Android via Wi-Fi. Please refer to [Connect WITH Android](#).

◆ Input a signal to a channel of the oscilloscope

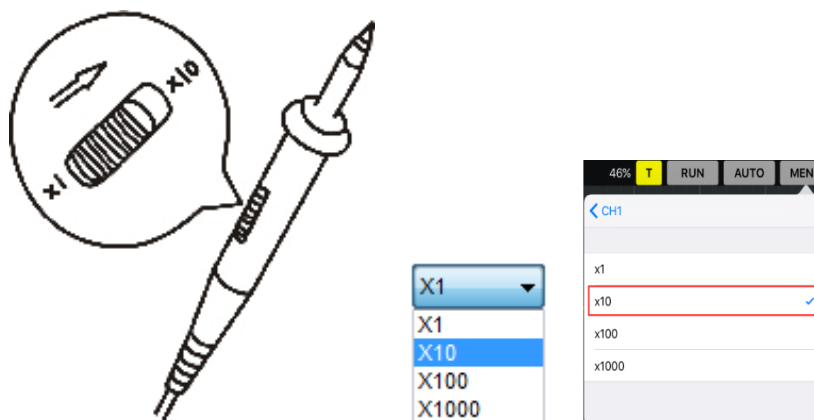
The oscilloscope is equipped with two channels plus external trigger.

Please input signal in the following steps:

1. Set the attenuation switch on the probe as 10X and connect the probe on the oscilloscope with CH1. Aim the slot in the probe connector at the faucet on BNC of CH1 and insert, then, turn right to lock the probe. Finally, attach the tip of probe and ground nip to the Connector of Probe compensator.



2. Set the CH1 probe attenuation of the oscilloscope to X10. (The default is X1).



3. Attach the tip of probe and ground nip to the Connector of Probe compensator. Click the **AUTO** button. A square wave will be displayed within a several seconds. (Approximately 1 KHz, 2V, peak- to- peak).
4. Inspect CH2 with the same method. Repeat steps 2 and 3.

1.7 Self Calibration

The self calibration routine lets you optimize the oscilloscope signal path for maximum measurement accuracy. You can run the routine at any time but you should always run the routine if the ambient temperature changes by 5°C or more. For accurate calibration, power on the oscilloscope and wait twenty minutes to ensure it is warmed up. To compensate the signal path, disconnect any probes or cables from the input connectors. Then, access the **“Utility -> Calibrate/Self Calibration”** option and follow the directions on the screen. The self calibration routine takes about several minutes.

Chapter 2 PC Application

- ◆ [Install Software](#)
- ◆ [Install Driver](#)
- ◆ [Set Connection Mode with PC](#)
- ◆ [The User's Interface](#)
- ◆ [The Menu System](#)
- ◆ [Set Oscilloscope](#)
- ◆ [Set Horizontal System](#)
- ◆ [Set Vertical System](#)
- ◆ [Set Trigger System](#)
- ◆ [MATH an FFT](#)
- ◆ [Save and Load](#)
- ◆ [Display System](#)
- ◆ [Measure System](#)
- ◆ [Acquisition Modes](#)
- ◆ [Interpolation](#)
- ◆ [Utility Function](#)
- ◆ [Print](#)

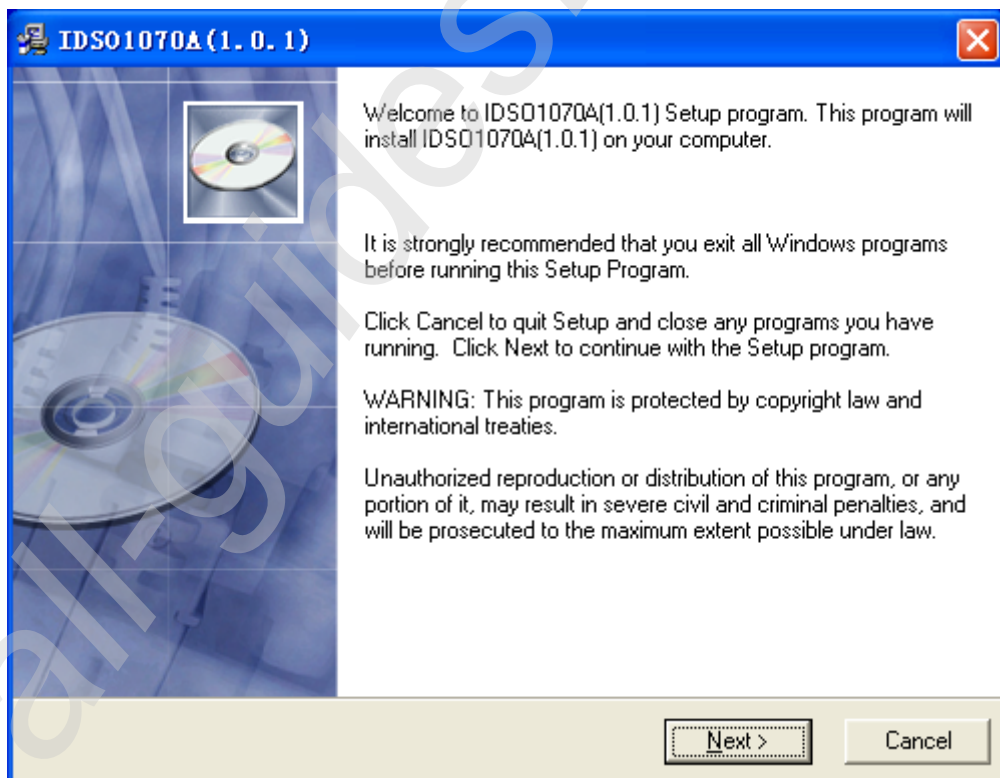
2.1 Install Software

Caution: You must install the software before using the oscilloscope on Windows PC.

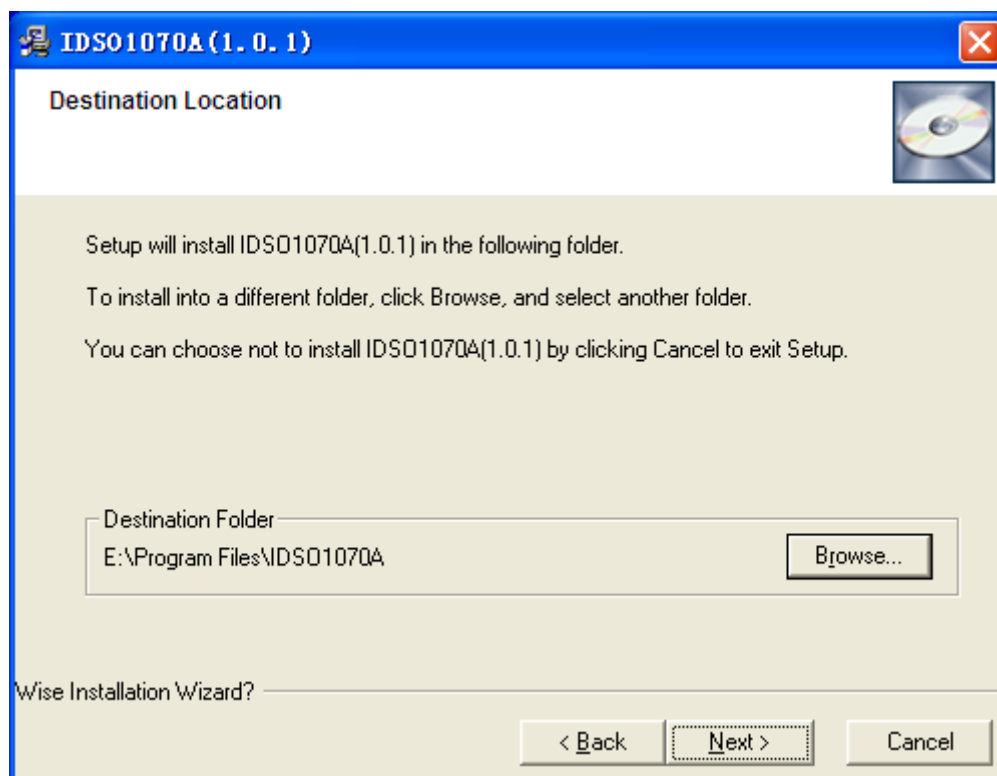
1. While in Windows, insert the installation CD into the CD-ROM drive.
2. The installation should start up automatically. Otherwise open Windows Explorer, and switch to the CD-ROM driver and run Setup.exe.



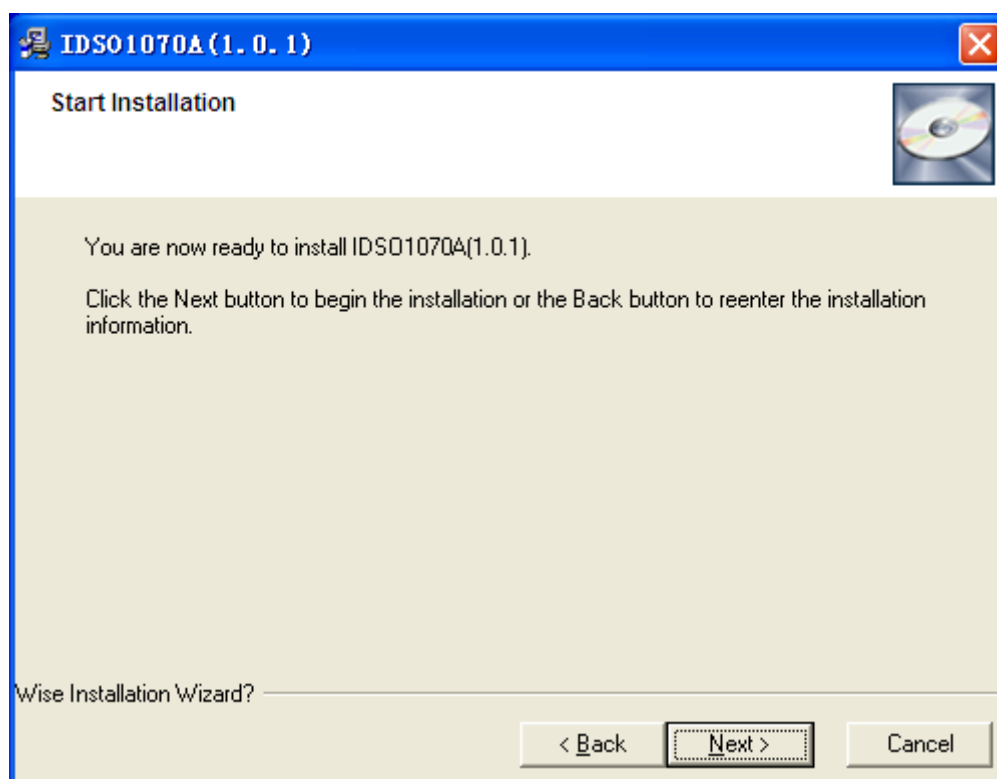
3. The software Installation is started. Click 'Next' to continue.



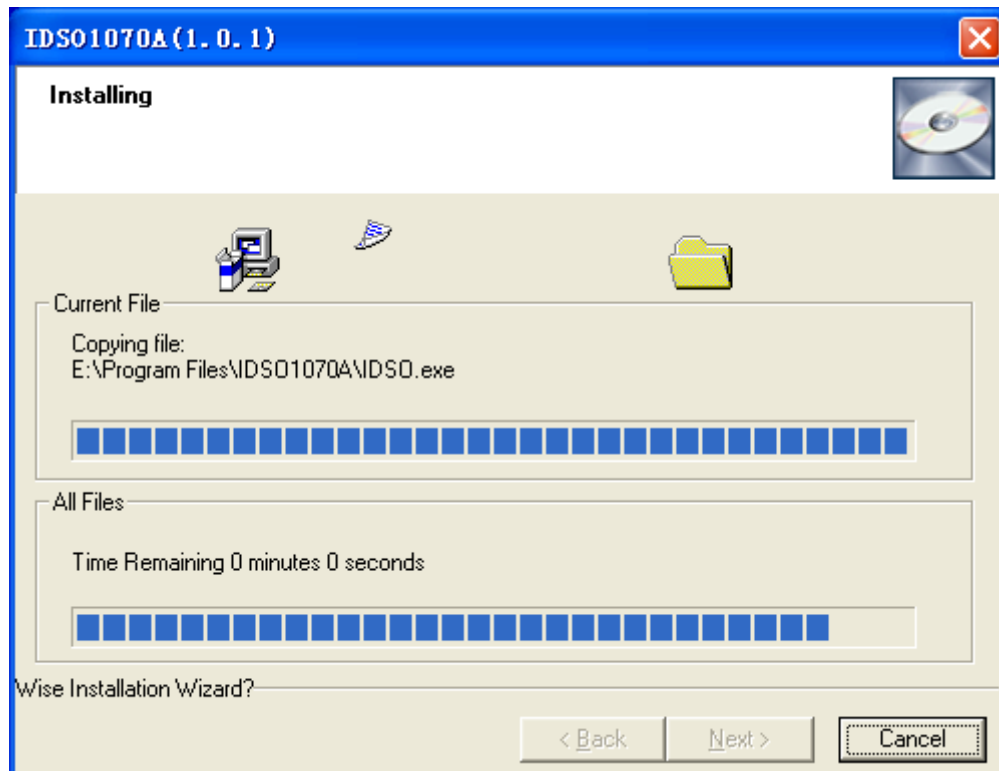
4. Choose a destination directory. Click 'Next' to continue.



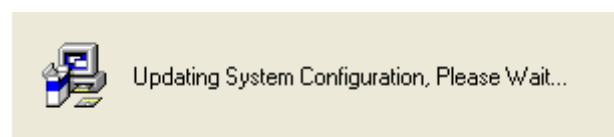
5. Check the setup information. Click Next to start copying of files.



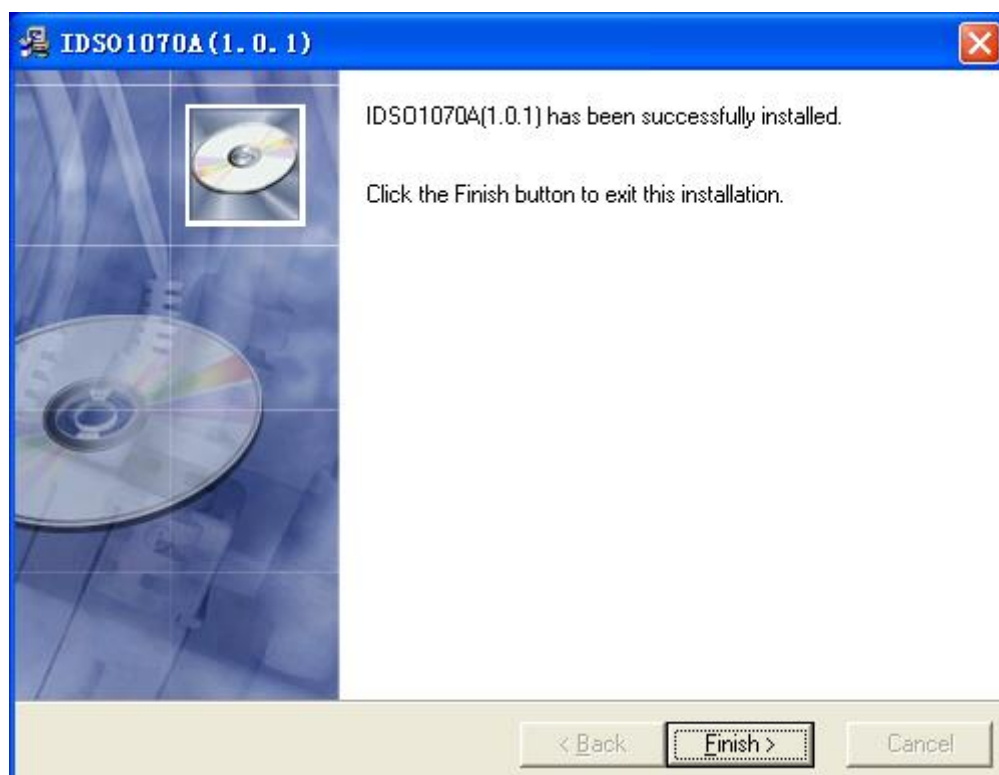
6. This Status dialog is displayed during copying of files.



7. Updating your system configuration.



8. The installation is complete.



2.2 Install Driver

Caution: You must install the driver before connect the device with Windows PC via USB cable. Not need to install driver if you connect IDSO with PC via Wi-Fi.

1. Connect the A-Type Plug of USB cable to your PC's USB port.



2. Connect the B-Type Plug of USB cable to IDSO's USB port.



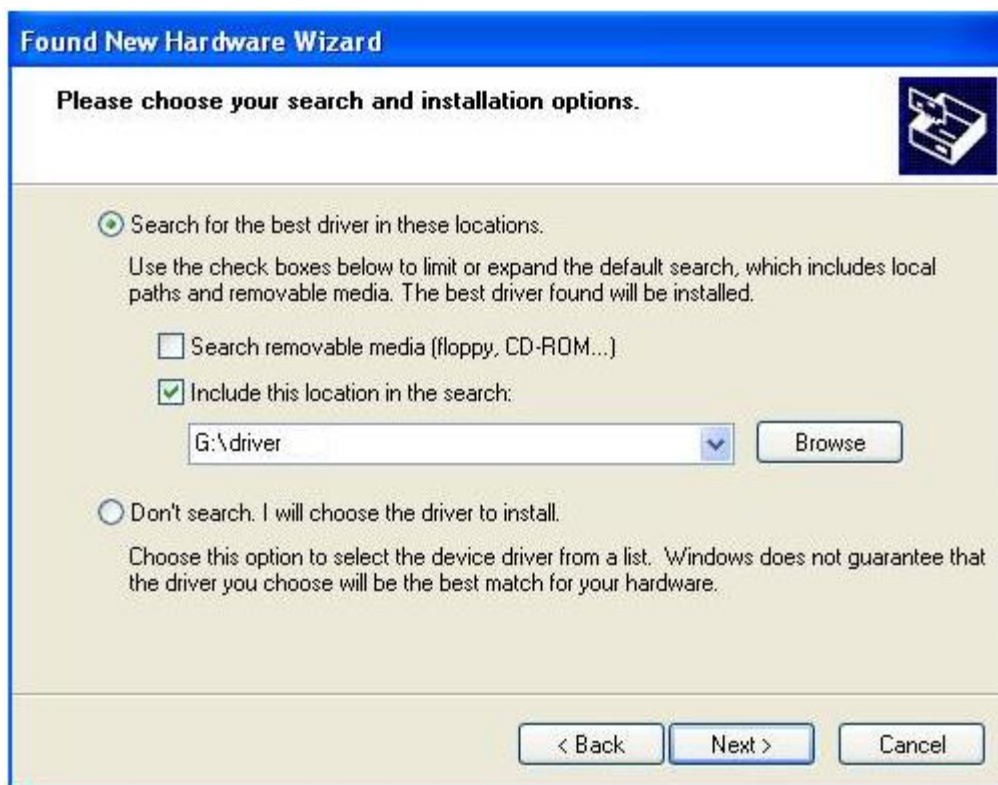
3. New hardware is found.



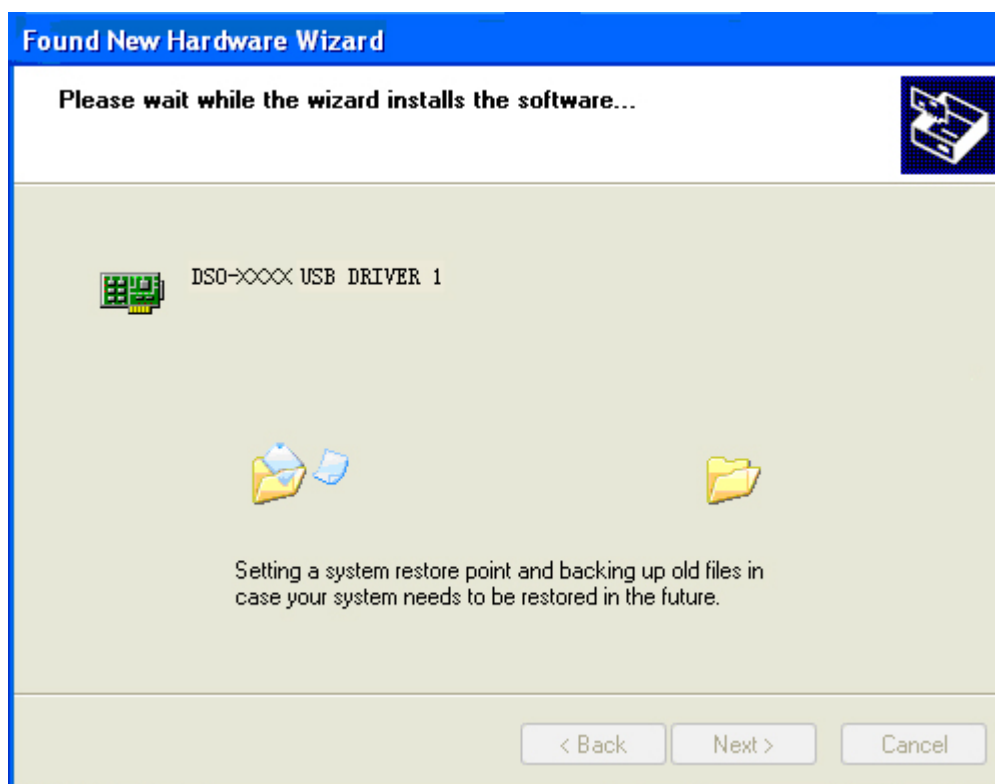
4. New hardware search wizard starts.



5. New hardware search wizard starts to search the drive.



6. New hardware wizard installs software



7. Finish new hardware search wizard.



8. New hardware is found.



9. New hardware wizard installs software and finish new hardware search wizard.



10. Your new hardware is installed and ready to use.



Note: Simply plug this cable's USB-B end into the device, and plug the two USB-A ends into open ports on one PC. This cable then pulls power from both USB port, and thus increases the output to the external driver. The red molded hood on the auxiliary USB-A connector enables the user to quickly distinguish between main and auxiliary power.

If the DSO instrument does not work, the possible causes are:

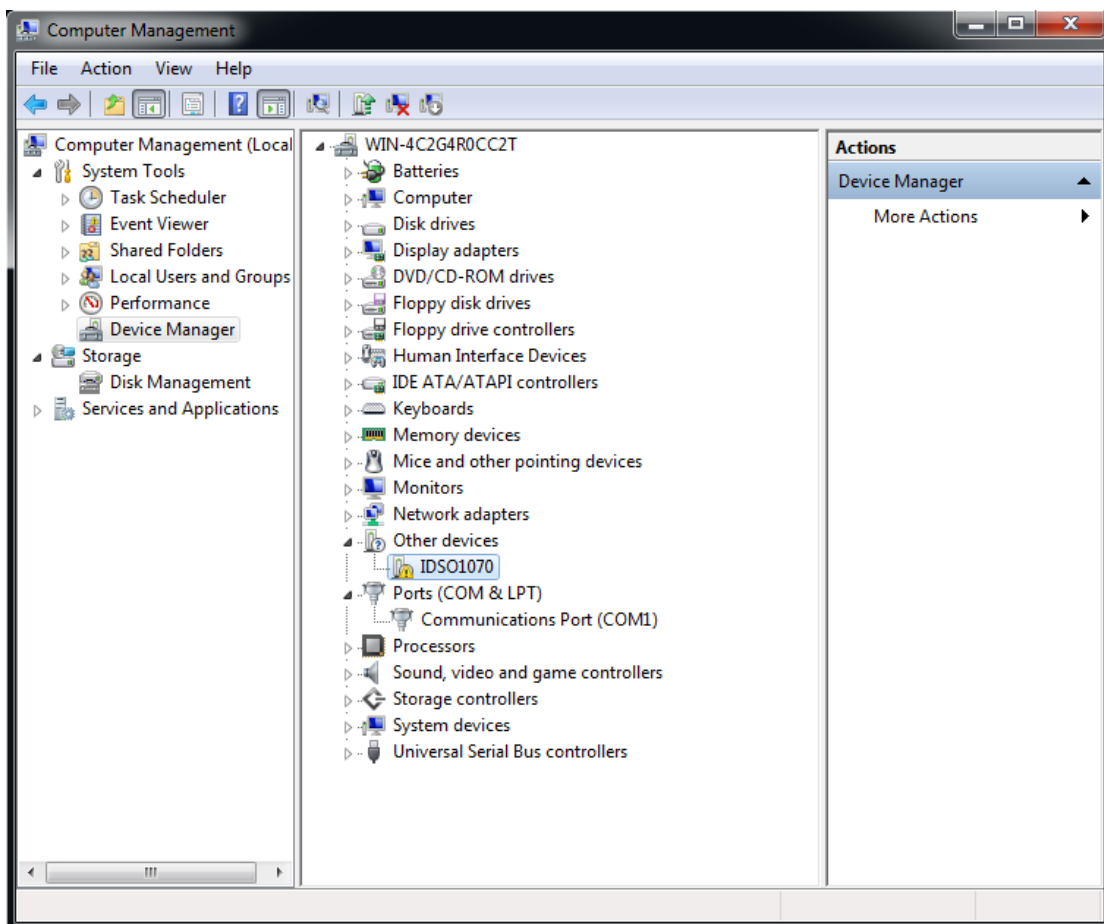
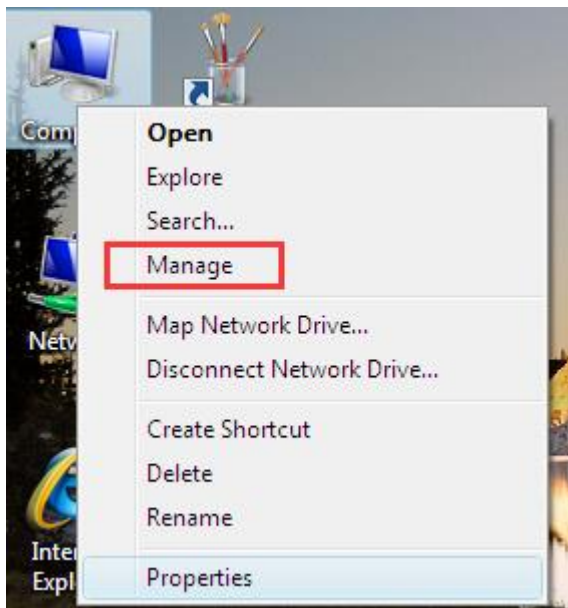
1. The instrument is not connected to PC.
2. The driver is not found.
3. The driver is wrong.

Method: Update Driver

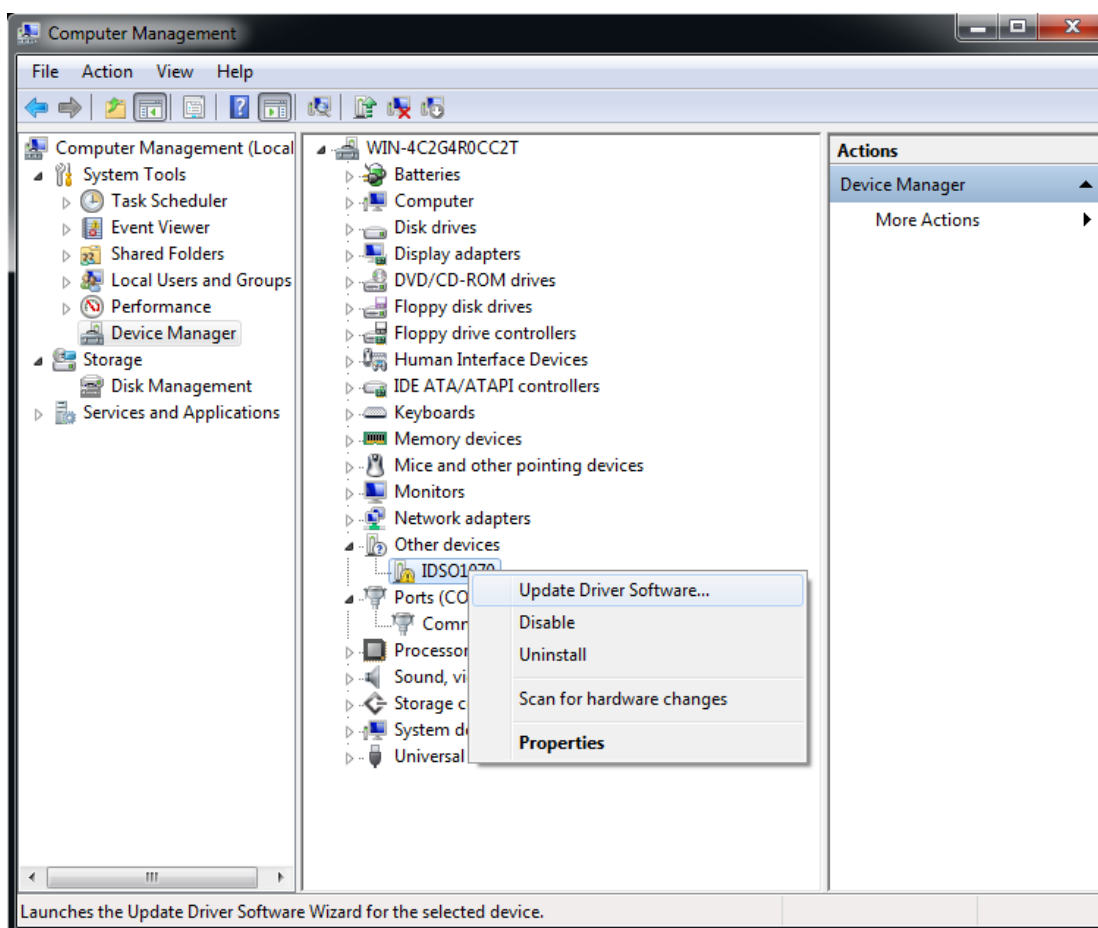
Open "Device Manager" and see the driver is right or not.

If the driver is wrong, please update the driver!

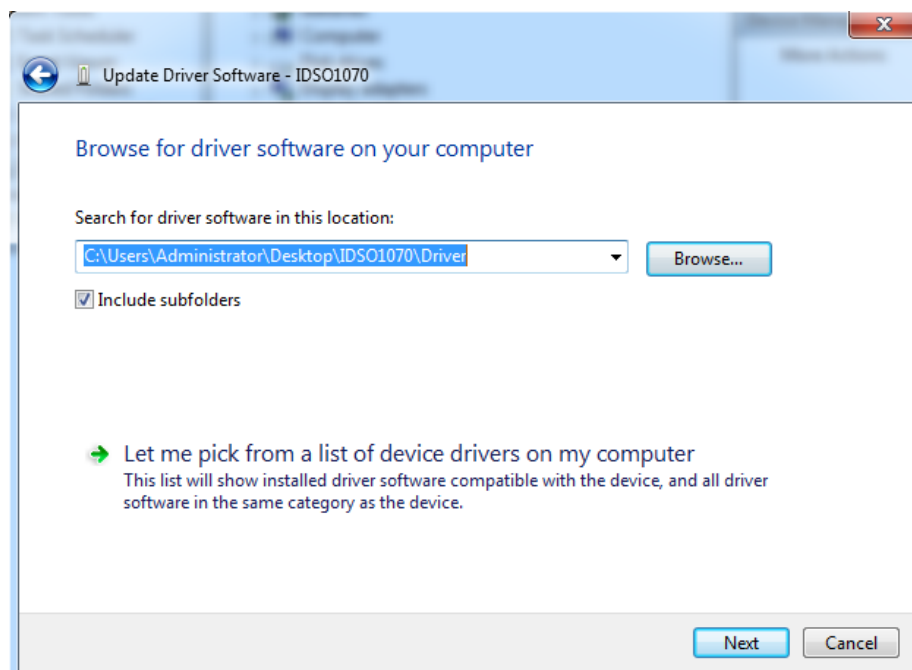
1. Right click "Computer->Manager" to open the device manager.



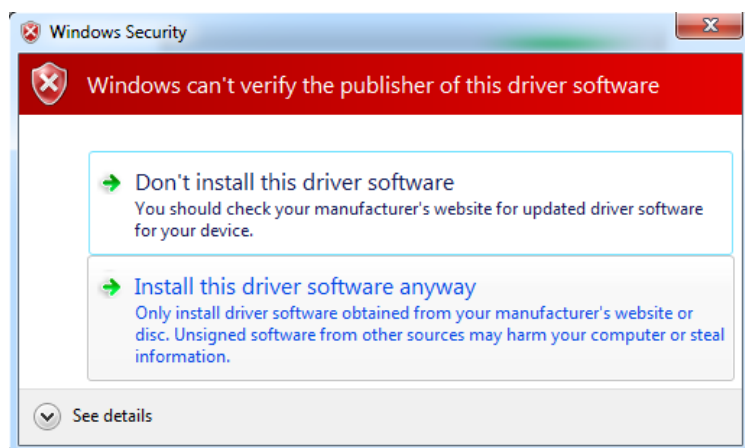
2. Right click "IDSO1070", select "Update driver software". According to the installation wizard to install.



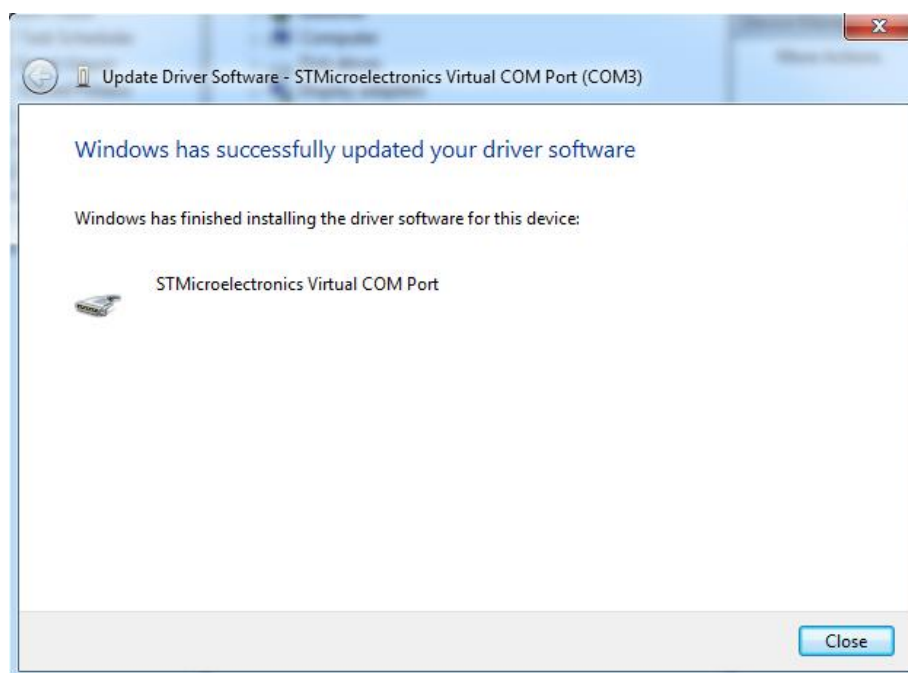
3. Select the driver path, click “Next”.



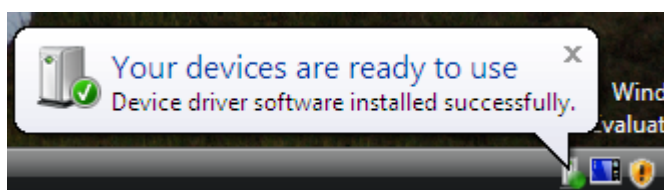
4. Click “Install this driver software anyway”.



5. The driver has been successfully installed.



Now your device is ready to use.

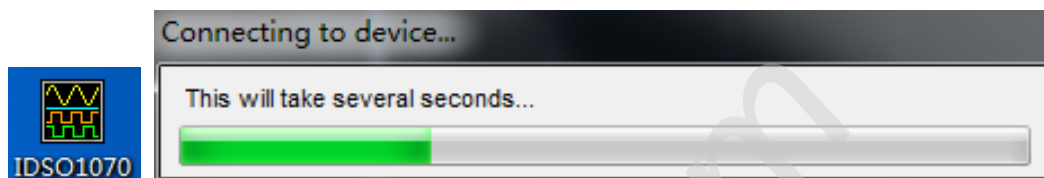


2.3 Set Connection Mode with PC

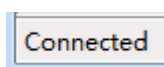
Communicate with PC via USB cable

1. Connect the A-Type Plug of USB cable to your PC's USB port. Connect the B-Type Plug of USB cable to IDSO's USB port.

2. Power on the IDSO.
3. Set connect mode to USB mode. Please refer to [Set Connection Mode](#).
4. Double click shortcut icon on desktop to open IDSO software. It will communicate with IDSO device automatically.



5. If the device is connected with PC successfully, the status information will display **“Connected”** on bottom left status bar.




Note: Only IDSO windows software can support USB mode.

Communicate with PC via Wi-Fi

1. Power on the IDSO.
2. Set connect mode to Wi-Fi. Please refer to [Set Connection Mode](#).
3. Connect your PC with the Wi-Fi access point.

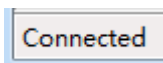


4. Double click shortcut icon on desktop to open IDSO software. And click  in toolbar to connect via Wi-Fi.



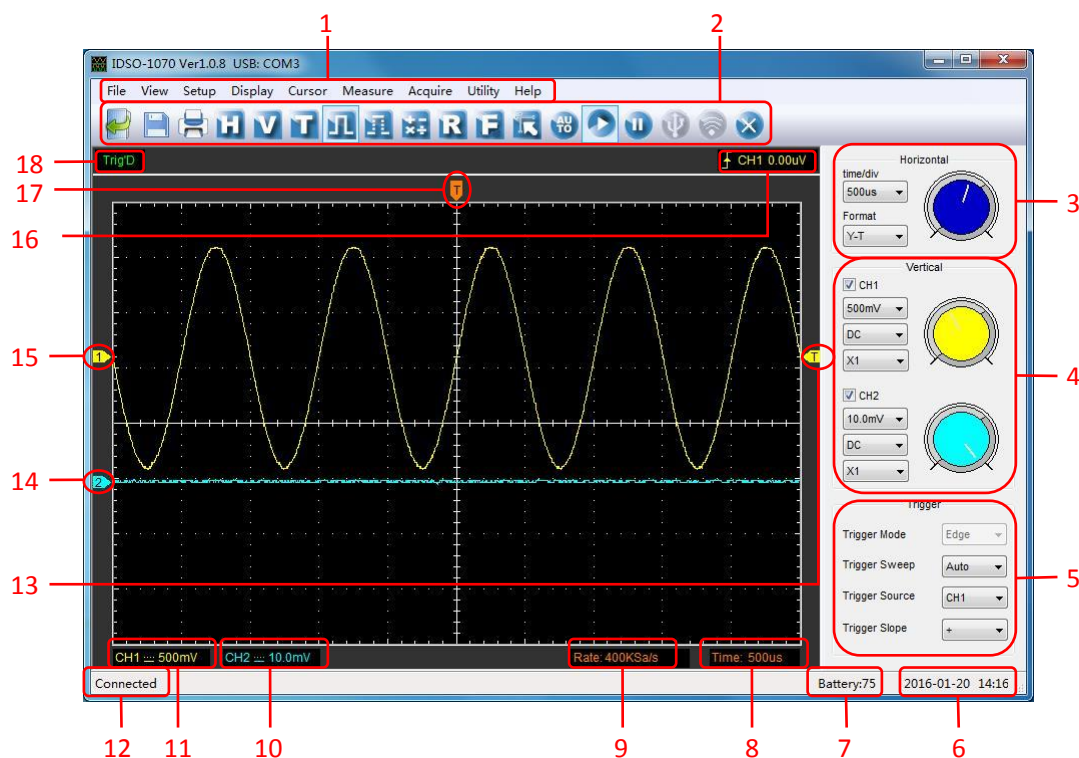
5. If the device is connected with PC successfully, the status information will display **“Connected”**

on bottom status bar.



2.4 The User's Interface

Click the software icon on the desk after you finished the software setting and equipment connecting. Then a user interface will be showed as follows:



1. The Main Menu

All setting can be found in the main menu.

2. The Toolbar

3. The Horizontal Panel

The user can change Time/Div, format in the panel.

4. The vertical Panel

The user can turn on/off the CH1/CH2. Also the user can change the CH1/CH2 volt/div, coupling and probe attenuation.

5. The Trigger Panel

In this panel, the user can change the trigger mode, sweep, source and slope.

6. Displays the system time.

7. Battery Level.**8. Time/Div.****9. Sample Rate.****10. Displays the CH2 information**

Readout shows the coupling of the channels.

Readout shows the vertical scale factors of the channels.

A “B” icon indicates that the channel is bandwidth.

11. Displays the CH1 information

Readout shows the coupling of the channels.

Readout shows the vertical scale factors of the channels.

A “B” icon indicates that the channel is bandwidth.

12. Status Information.

Demo: Don't communicate with PC.

Disconnected: The device is not connected with PC successfully.

Connected: The device is connected with PC successfully.

13. Trigger Level.**14. The marker shows the reference points of the displayed waveforms.**

If there is no marker, the channel is not displayed.

15. The same as 14.**16. Displays the trigger information**

Display the edge trigger slope, source and level.

17. Marker shows horizontal trigger position.**18. Trigger status indicates the following.**

Auto: The oscilloscope is in auto mode and is acquiring waveform in the absence of triggers.

Trig'D: The oscilloscope has seen a trigger and is acquiring the post trigger data.

WAIT: All pretrigger data has been acquired and the oscilloscope is ready to accept a trigger.

Stop: The oscilloscope has stopped acquiring waveform data.

Run: The oscilloscope is running.

Play: The oscilloscope is displaying the record waveform.

2.5 The Menu System

The Main Menu:

File View Setup Display Cursor Measure Acquire Utility Help

1. **File:** Load or save CSV, waveform and setup
2. **View:** Change the user interface
3. **Setup:** Set Horizontal, vertical, trigger system.
4. **Display:** Change wave display type, intensity and color.
5. **Cursor:** Set cursor measure source and measure type
6. **Measure:** Set measurement source and measure parameters.
7. **Acquire:** Start, Stop, Autoset and other operation setting
8. **Utility:** Utility setting, record waveform, pass/fail test, self calibration, factory setup and language setting.
9. **Help:** Turn on help file.

2.6 Setup the Oscilloscope

Use “AUTOSSET” to display a signal automatically.

Auto setup functions one time each time you click the “**AUTO**” button. The function obtains a stable waveform display for you. It automatically adjusts the vertical scale, horizontal scale and trigger settings. Auto setup also displays several automatic measurements in the graticule area, depending on the signal type.

Connect a signal to the CH1 input:

1. Connect a signal to the oscilloscope as described above.
2. Click the “**Acquire ->Autoset**” button.

The oscilloscope will change the current settings to display this signal.

Save Setup

The oscilloscope software saves the current setup before you close the oscilloscope software. The oscilloscope recalls this setup when you run the software next time. You can use the “**File->Save Setup**” in main menu to permanently save setting.

Load Setup

The oscilloscope can recall the last setup before the oscilloscope software was running, any saved

setups, or the factory setup. You can use the “**File->Load Setup**” in main menu to permanently recall setup.

Factory Setup

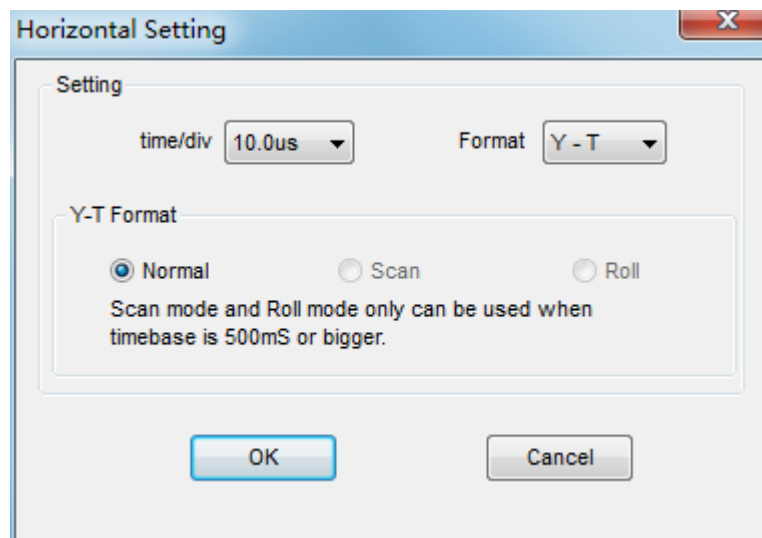
The oscilloscope software is set up for normal operation when it is shipped from the factory. This is the factory setup. Click the “**Utility->Factory Setup**” in main menu to recall this setup.

2.7 Set Horizontal System

2.7.1 Horizontal Setting

Click “**Setup->Horizontal**” menu.

The following figure shows the Horizontal System window. It shows the horizontal parameters settings.



➤ Horizontal Setting

1. **Time/DIV:** lead the setting of the time base parameters

Select the time setting for the beam to sweep one major division on the screen. By selecting different time/div settings it is possible to zoom in on the frozen waveform on the screen.

2. **Format:** lead the setting of the horizontal format parameters.

Set the waveform display format (**Y-T**, **X-Y**, **Scan** and **Roll**).

Y -T: Show the relative relation between vertical voltage and horizontal time.

X -Y: Show CH1 value at X axis; CH2 value at Y axis. The XY format is used to analyzer phase difference, such as those represented by Lissajous patterns.

➤ Y-T Format

Scan: In Scan Mode, the waveform display scan updates from left to right.

In **Scan** mode, trigger and horizontal offset control of waveforms are available during Scan Mode.


This mode is only available when set to 500ms/div or slower. Generally, used for measuring low frequency signal.

Roll: In Roll Mode, the waveform display updates from right to left.

In **Roll** mode, the waveform display rolls from right to left. No trigger or horizontal offset control of waveforms is available during Roll Mode. This is only available when set to 500ms/div or slower.

Caution: If the time/div bigger than or equal to 500ms, the format will change to Roll mode automatically. If you want to select Scan mode, please click "**Setup -> Horizontal -> Scan**" in main menu to set Scan mode manually.

➤ **Change Horizontal Position**

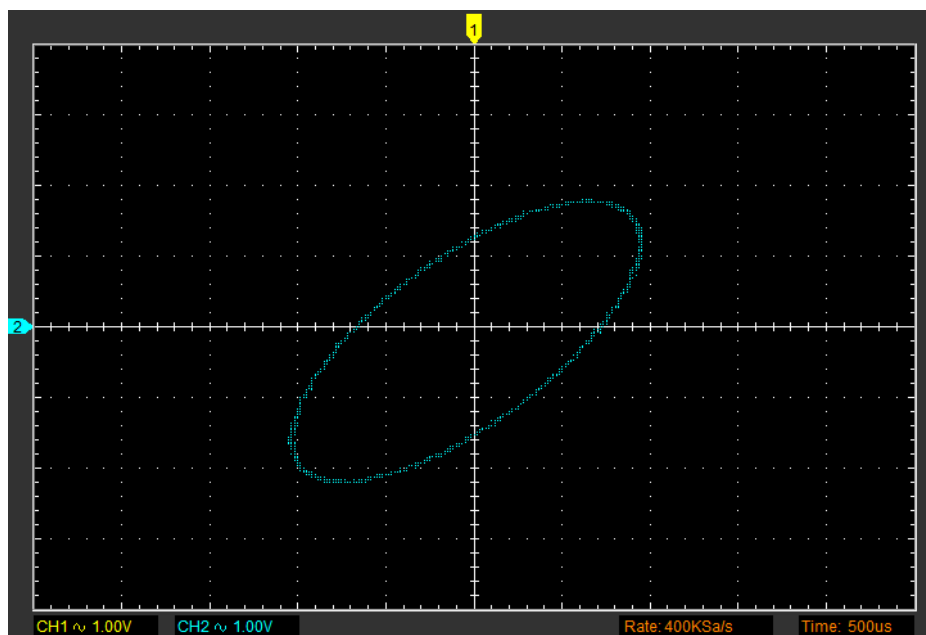
Horizontal Position icon: Drag Horizontal Position icon  on screen to adjust the trigger position against screen. Double click the icon to reset the trigger point back to the screen center.

2.7.2 The Application of the X-Y Operation

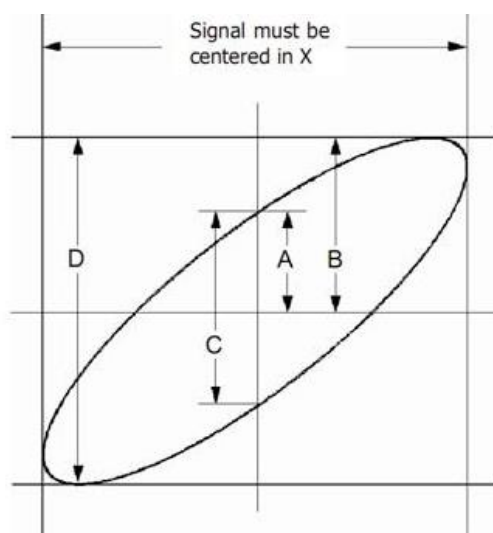
X-Y Plot acts to analyze correlation of data of two channels. Lissajous diagram is displayed in the screen when you use **X-Y** Plot, which enables to compare frequencies, amplitudes and phases of counterpart waveform against the reference waveform. This makes it possible to compare and analyze frequency, amplitude and phase between input and output.

Do these steps as follows:

1. Set the probe attenuation to "**x10**". Set the switch to "**x10**" on the probes.
2. Connect the CH1 probe to the input of the circuit, and connect the CH2 probe to the output of the circuit.
3. Click **AUTO** button.
4. Adjust the vertical scale and offset to display approximately the same amplitude signals on each channel.
5. Select X-Y format at Horizontal window. The oscilloscope will displays a Lissajous pattern representing the input and the output characteristics of the circuit.
6. Adjust the scale and offset of the horizontal and vertical to a desirable waveform display. The following picture shows a typical example.
7. Apply the Ellipse Method to observe the phase difference between the two channels.

Signal in X-Y Format:

Instruction of the Ellipse Method:



$\sin\theta = A/B$ or C/D , where θ = phase shift (in degrees) between the two signals.

From the formula above:

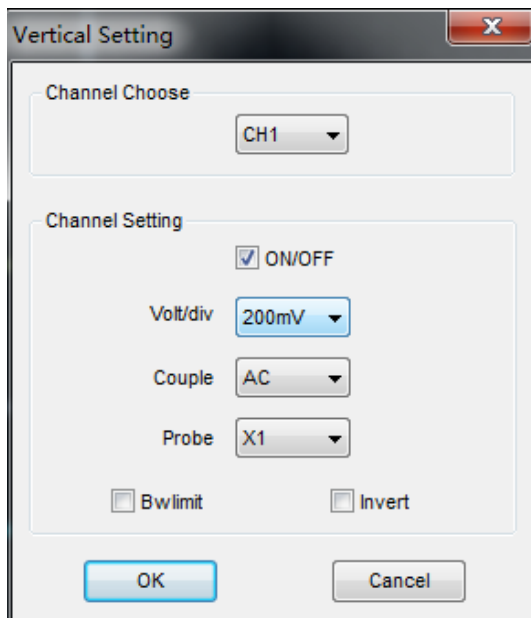
$$\theta = \pm \arcsine (A/B) \text{ or } \pm \arcsine (C/D)$$

θ must be in the range of $(0 \sim \pi/2)$ or $(3\pi/2 \sim 2\pi)$ if the main axis of the ellipse is between I and III quadrant. If the main axis is at II and IV quadrant, θ must be in the range of $(\pi/2 \sim \pi)$ or $(\pi \sim 3\pi/2)$.

2.8 Set Vertical System

Click **“Vertical”** in **“Setup”** menu.

The following figure shows the Vertical System window. It shows the vertical parameters settings.



Channel Choose: User can select the channel by clicking the Combo box.

ON/OFF: Turn on/off the channel



Volt/div: Select the channel voltage/div

Couple: Select the channel coupling
DC/AC/GND.

Probe: Set the selected one according to the
probe attenuation factor to ensure correct vertical

Also user can also set parameters in the right sidebar.

Vertical Position icon

Drag Vertical Position icon   to move waveform up and down on the screen. Double click the icon to reset the waveforms to the vertical center position on the screen.

Change Volt/Div

You can click **“Volt/div”** in **“Vertical Setting”** window to select the voltage.

You can left click and drag the mouse on the knob to change the voltage.

The signal on the screen can be enlarged or reduced vertically by adjusting the displayed voltage per division. (V/div = voltage per division).

Set Channel Coupling

Click **“Couple”** in **“Vertical Setup”** window.

You can set the coupling to **DC**, **AC** or **GND**. If you set the coupling to **DC**, it blocks the **AC** component of the input signal.

AC: the input signal is capacitive coupled to the input amplifier/attenuator. Only the AC components are measured.

GND: the input signal is broken and the input amplifier/attenuator is connected to earth. Use

this position for selecting a reference point on the display.

DC: the input signal is directly connected to the input amplifier/attenuator. Both AC and DC voltage are measured.

Probe Attenuation Setting

Select the attenuation factor for the probe. To check the probe attenuation setting, toggle the probe menu to match the attenuation factor of the probe.

This setting remains in effect before you changed again. Click **“Probe”** in **“Vertical Setting”** window to select the probe attenuation.

Note: The attenuation factor changes the vertical scale of the oscilloscope so that the measurement results reflect the actual voltage levels at the probe tip.

➤ Bandwidth Limit

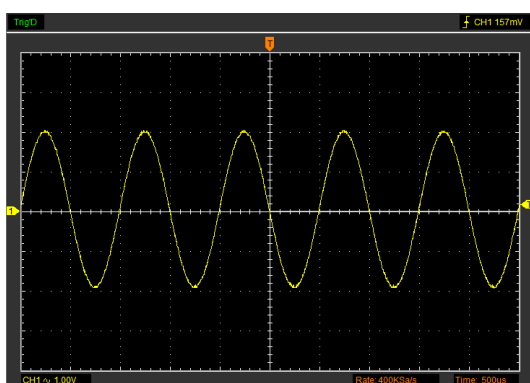
The oscilloscope is set to full bandwidth and will pass the high frequency component in the signal if the **“BW Limit”** was turned off.

The oscilloscope will reject the frequency component higher than 20MHz if the **“BW Limit”** was turned on.

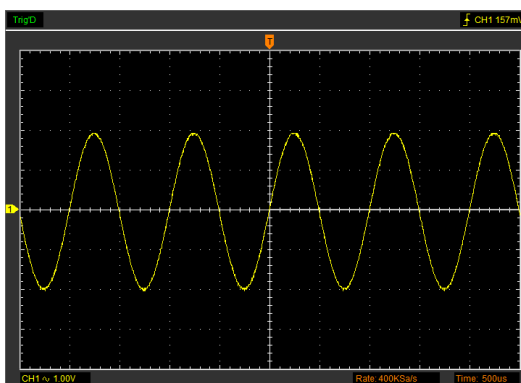
➤ Invert

The invert function turns the displayed waveform 180 degrees, with respect to the ground level. When the oscilloscope is triggered on the inverted signal, the trigger is also inverted.

Click **“Invert”** in **“Vertical Setting”** window.



Not select “Invert”



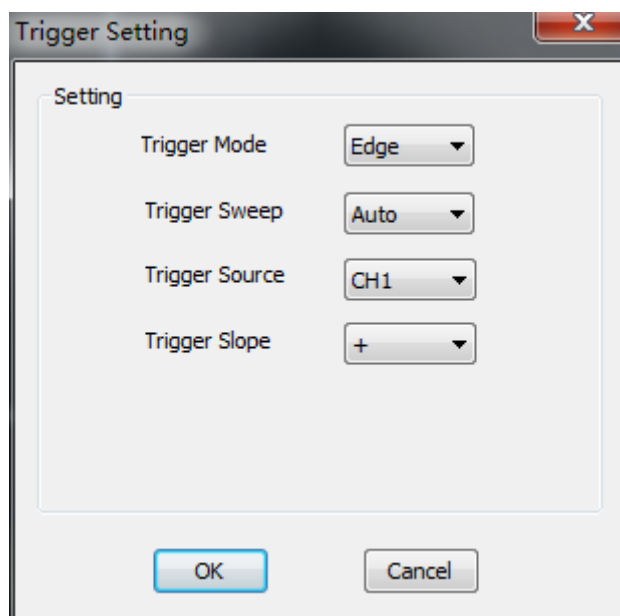
Select “Invert”

2.9 Set Trigger System

2.9.1 Trigger Setting

Click “**Setup-> Trigger**” menu.

The following figure shows the trigger system control.



➤ Trigger Setting

The trigger determines when the oscilloscope starts to acquire data and display a waveform. When a trigger is set up properly, it can convert unstable displays or blank screens into meaningful waveforms.

If the oscilloscope wants to acquire a waveform, it collects enough data so that it can draw the waveform to the left of the trigger point. The oscilloscope continues to acquire data while waiting for the trigger condition to occur. The oscilloscope continues to acquire enough data so that it can draw the waveform to the right of the trigger point after it detects a trigger.

The Edge trigger determines whether the scope finds the trigger point on the rising or the falling edge of the signal. Select Edge trigger mode to trigger on Rising edge or Falling edge.

Trigger Mode: Edge.

Trigger Sweep: Set the sweep mode to **Auto**, **Normal** or **Single**.

Auto: Performs the acquisition freely in absence of valid trigger. It allows the generation of untriggered waveforms.

Normal: Updates the displayed waveforms only when the oscilloscope detects a valid trigger condition. Before this update, the oscilloscope still displays the old waveforms. This mode shall be used when you want to only view the effectively triggered waveforms. In this mode, the oscilloscope displays waveforms only after the first trigger.

Single: Acquire waveform when trigger occurred, then stop to acquire.

Trigger Source: You can use the trigger source options to select the signal that the oscilloscope uses as a trigger. Select the trigger source signal (CH1, CH2 or EXT).



Trigger Slope: Set the slope to **Rising (+)** or **Falling (-)**.

Rising: Trigger on rising edge

Falling: Trigger on falling edge

Also user can also set trigger parameters in the bottom right sidebar.

➤ Change Trigger Level

Trigger Level icon: User can also drag the trigger icon   on screen to adjust the trigger level. Double click trigger icon to set trigger level to the vertical midpoint between the peaks of the trigger signal.

2.9.2 Capturing a Single-Shot Signal

To capture a single event, it needs to gather some pre-test knowledge of the signal in order to set up the trigger level and slope correctly. For example, if the event is derived from 3.3V COMS logic, a trigger level of 1.2 or higher Volts should work on a rising edge.

Do these steps as follows:

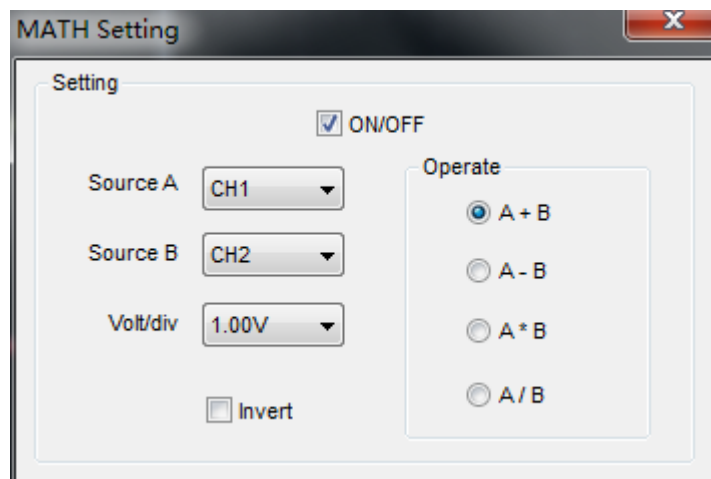
1. Set the probe and the channel attenuations to “**x10**”.
 2. Set up the trigger in the Trigger Menu, or in the Trigger Setting window.
 - (1) Adjust the Trigger Mode to **Edge**.
 - (2) Set the Trigger Sweep to **Single**.
 - (3) Set the Trigger Source to **CH1**.
 - (4) Set the Trigger Slope to “**+**” which means you select the rising edge.
 - (5) Adjust the **Volt/Div** and the time base in a proper range for the signal.
 - (6) Drag the trigger level sign on the waveform display screen to proper position. It ally higher a little above the normal level.
 - (7) Click **START** button to start capturing. When the trigger conditions are met, data appears on the display representing the data points that the oscilloscope obtained with one acquisition.
- This function helps to capture the signal occurrence easily, such as the noise with large amplitude; set the trigger level higher a little above the normal level and press and wait. When noise occurs, the instrument will record the waveform before and after the trigger.

2.10 Math and FFT

2.10.1 MATH

Click “**Setup->MATH**” to set **MATH** channel.

The **MATH Setting** window:



ON/OFF: Turn On/Off the MATH Channel.

Source A/B: Set the sources of the math channel.

Volt/DIV : Set the resolution of the math channel.

Operate: Set operates type of the math channel.

The mathematic functions include addition, subtraction, multiplication, division.

Operate

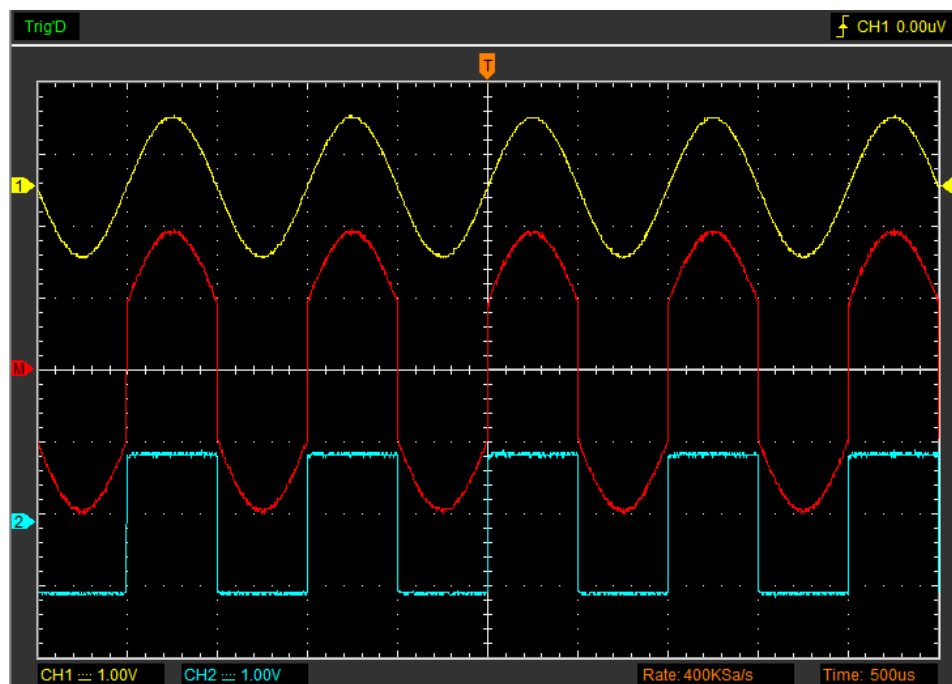
Four Types:

A + B	Add source A and source B
A - B	Subtract source B from source A
A x B	Multiply source A by source B
A / B	Divide source A by source B

In this function, use the addition, subtraction, multiplication, division, function to operate and analyze the waveform.

Select the operate type in the **Operate** menu. Select source A and B. Then adjust the vertical scale and offset to view the math channel clearly. The mathematic result can be measured by the measure and the cursor.

The Math Function Display:

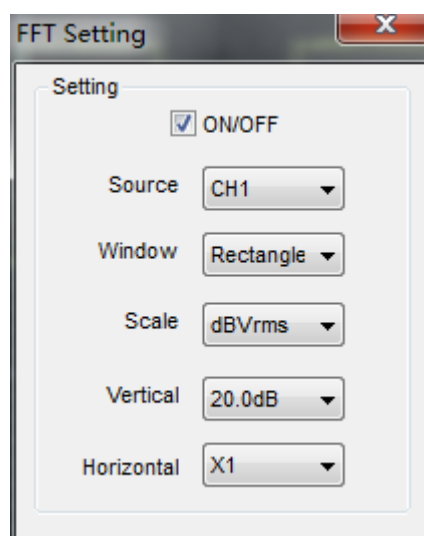


2.10.2 Fast Fourier Transform Function

User can use the FFT mode to convert a time-domain signal into its frequency components (spectrum). You can use the FFT mode for the following types of analysis:

To use the FFT mode, you need to click “**Setup-> FFT**” to open the FFT setting window.

The FFT setting window:



You can select the Source channel, Window algorithm, FFT number, and FFT Zoom factor. It displays only one FFT spectrum at a time.

Source: Selects the channel used as the FFT source

Window: Selects the FFT window type

The windows reduce spectral leakage in the FFT spectrum.

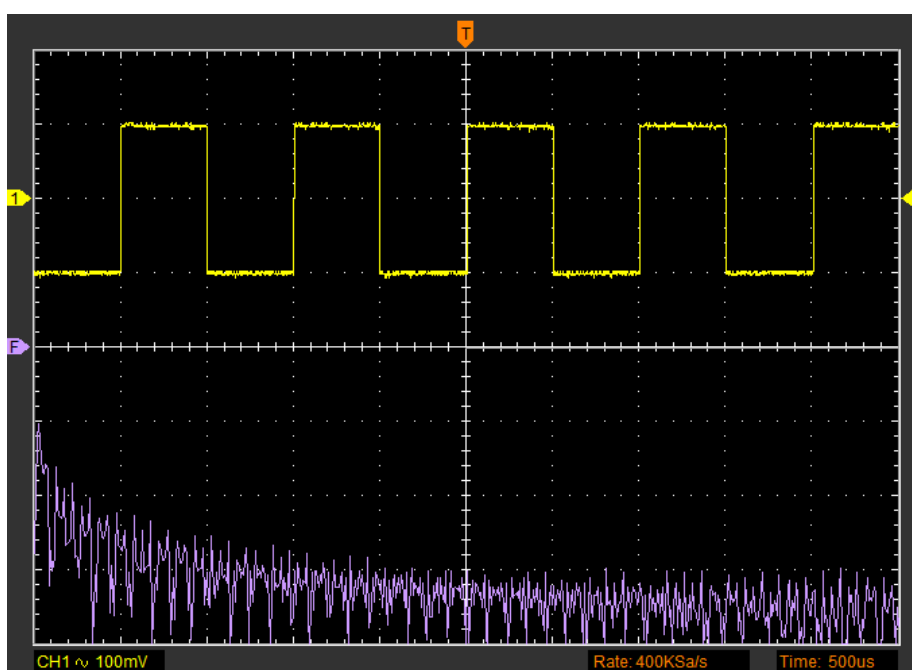
Scale: Selects the vertical scale units

Vertical: Selects the vertical scale

Horizontal: Selects the FFT zoom factor

Sets horizontal scale, If the information is more than the quantity displayed to the display window, more information is displayed by using scroll bar.

The FFT display Windows:



2.11 Save and Load

2.11.1 Save and load file

Click “File” in main menu to save data, setup and image.

1. Save Data

Save waveform data as .txt, .xls, .doc, .csv or .wms file.

2. Save Image

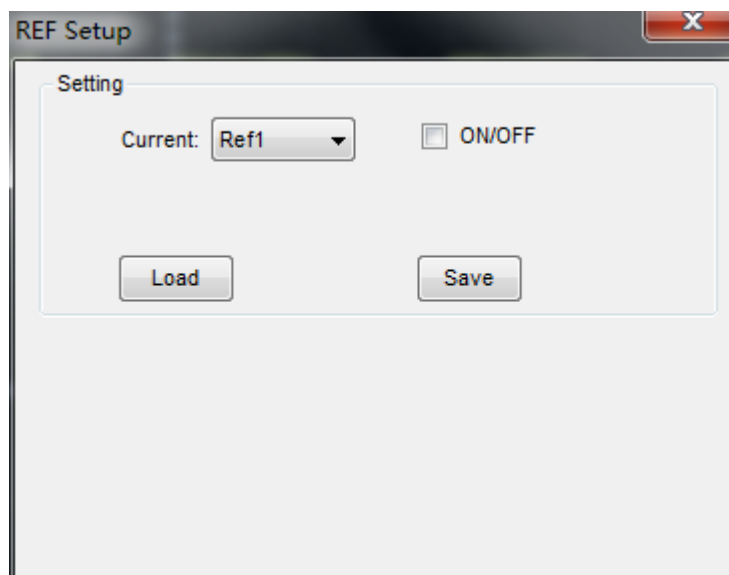
Save the software display window as a .bmp or .jpg file.

3. Load Data

Load waveform data as .wms file.

2.11.2 Set Reference

Click the button  in toolbar or “**Setup-> REF**” on menu to set REF channel.



There are two Ref channels, Ref1 and Ref2.

The Reference Channel Function:

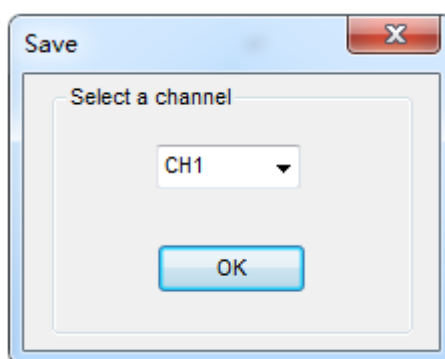
On/Off: Turn on/off the reference channel.

Load

Click “**Load**” to load the “*.rfc” file that was selected. The load file window will appear.

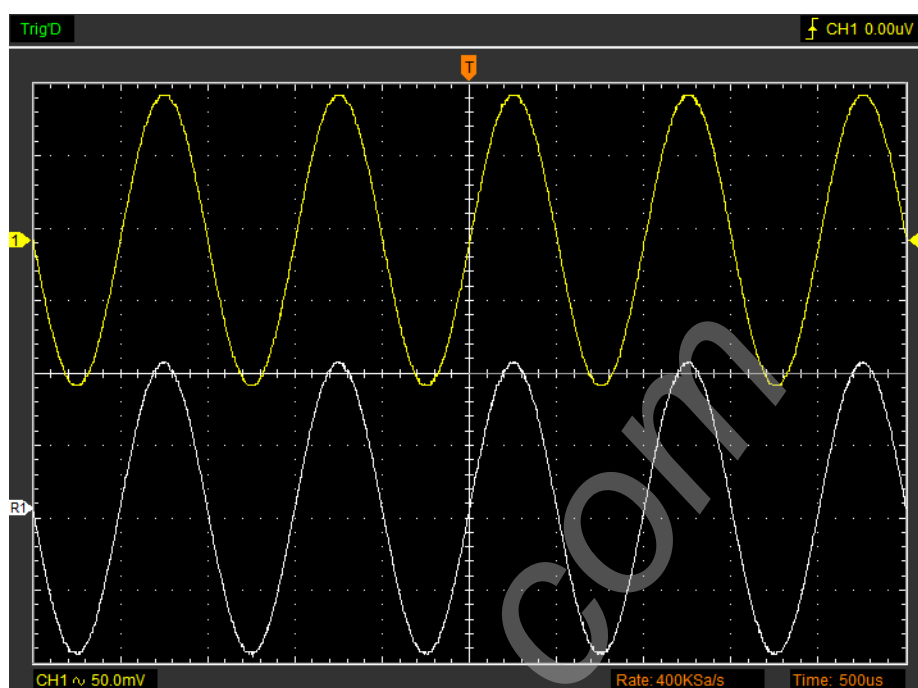
Save

Click “**Save**” to save the waveform to *.rfc file. The saved source window appears:



The save file window will appear after you selected the saved source.

The Reference Waveform Display Window:



Note: If you turn on the “Reference” channel, the load file window will appear.

2.11.3 Save/Load Setup

Save Setup

You can use the “File-> **Save Setups**” in main menu to permanently save setting.

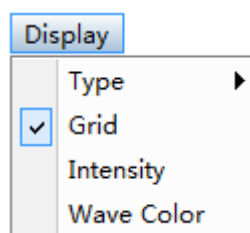
Load Setup

You can use the “File-> **Load Setups**” in main menu to permanently recall setup.

2.12 Display System

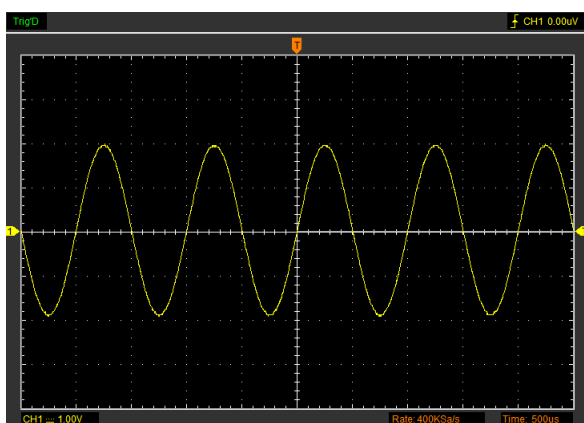
Click “**Type**” in “**Display**” menu.

The following figure shows the type parameters setting.

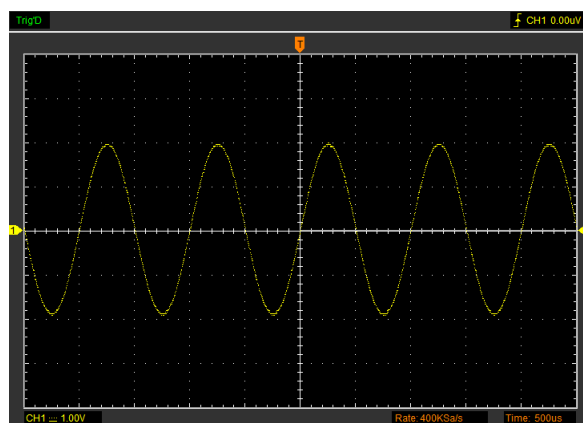


➤ Display Type

Click “**Display->Type**” in main menu. Select display type Vectors or Dots.



Vectors

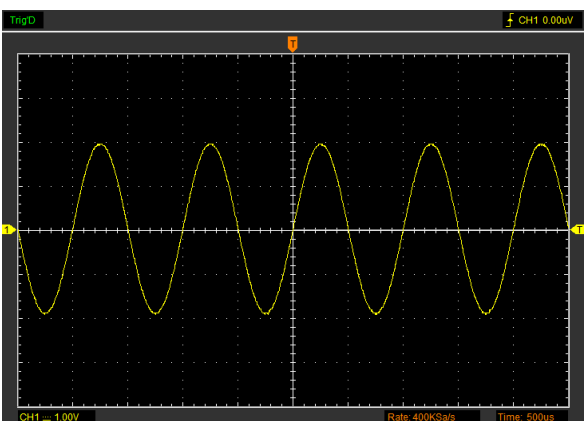


Dots

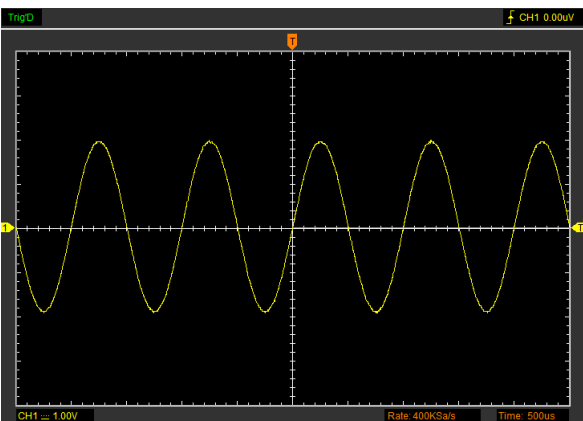
➤ Display Grid

Click **“Display->Grid”** in main menu.

The grid shows:



Grid

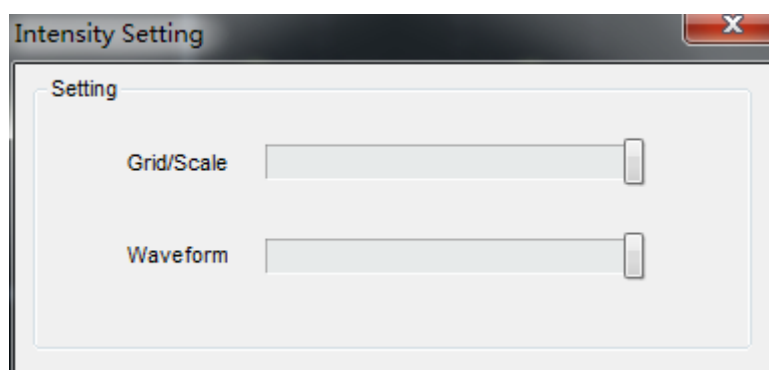


No Grid

➤ Intensity

Click **“Display->Intensity”** in main menu.

The following figure shows the intensity dialog. It shows the display parameters setting.

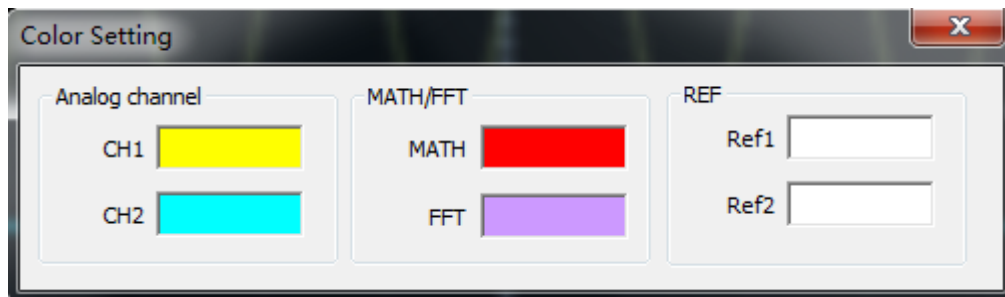


You can change the grid and waveform color intensity in this dialog.

➤ Wave Color

Click “**Display->Wave Color**” in main menu.

The following figure shows the waveform color dialog. It shows the waveform color setting.



If you click the color rectangle, the color dialog will be showed:



2.13 Measure System

The oscilloscope displays graphs of voltage versus time and can help to measure the displayed waveform. There are several ways to take measurements, using the graticule, cursor measurement or performing an automatic measurement.

2.13.1 Scale measurement

Graticule: This method allows you to make a quick, visual estimate and take a simple measurement through the graticule divisions and the scale factor.

For example, you can take simple measurements by counting the major and minor graticule divisions involved and multiplying by the scale factor. If you counted 6 major vertical graticule divisions between the minimum and maximum values of a waveform and knew you had a scale

factor of 50mV/division, you could easily calculate your peak-to-peak voltage as follows:

$$6 \text{ divisions} \times 50\text{mV/division} = 300\text{mV}.$$

2.13.2 Cursor Measure

Click "**Cursor**" in main menu.

This method allows you to take measurements by moving the cursors.

1. Source

The user can set the source to **CH1**, **CH2** and **MATH**.

When you use cursors, be sure to set the **Source** to the waveform on the display that you want to measure.

2. Type

There are four types of cursors: **Cross**, **Trace**, **Vertical** and **Horizontal**.

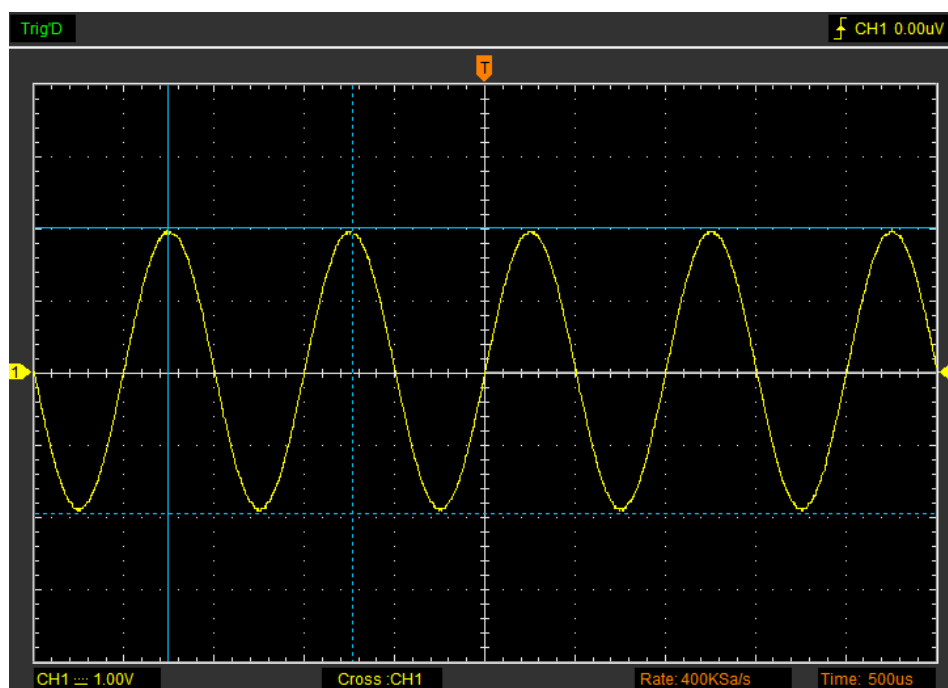
1) Cross

The **Cross** cursors appear as cross lines on the display and measure the vertical and horizontal parameters.

Do these steps:

1. Click "**Cursor**"-> "**Source**", select CH1 (select CH2 if you want measure CH2).
2. Click "**Cursor**"-> "**Type**", select Vertical.
3. Push left mouse button, and the vertical lines appear.
4. Drag the mouse button to the point you want to measure.
5. Release the left mouse button, the frequency difference and time difference will be shown at the status bar.

The **Cross** cursor display window:



The **Cross** measure result displays on status bar: Freq: 975.9Hz Time: 1.02ms Volt: 3.97V

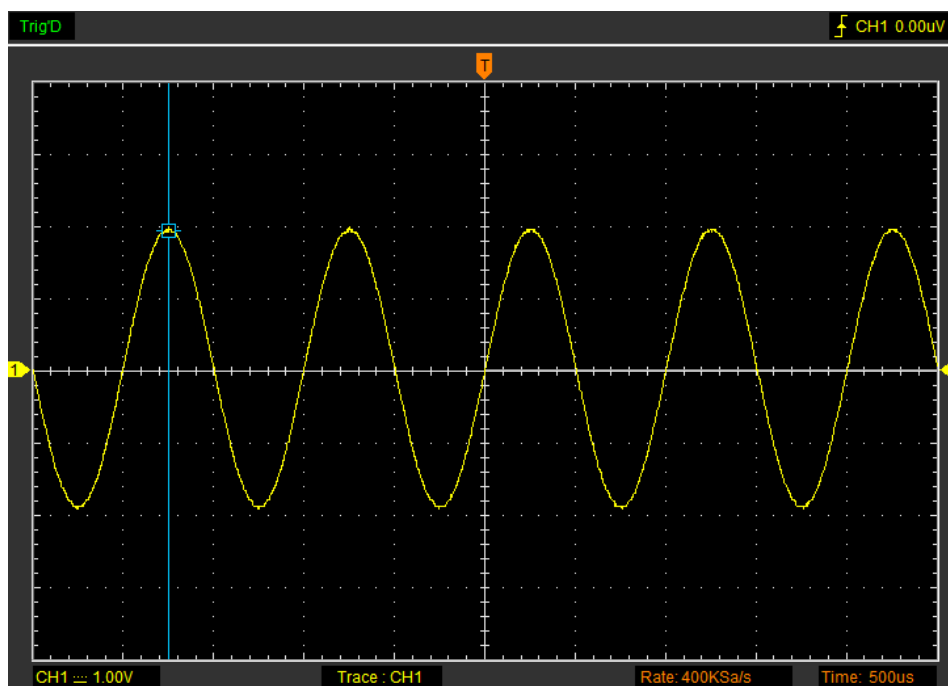
2) Trace

The **Trace** cursors appear as vertical lines on the display and measure the waveform amplitude at the point the waveform crosses the cursor.

Do these steps:

1. Click "**Cursor->Source**", select CH1 (select CH2 if you want trace CH2).
2. Click "**Cursor->Type**", select Trace.
3. Click the cursor at the position that you want traced of the wave in the waveform window.

The **Trace** cursor display window:



The **Trace** cursor measure result display on status bar: Volt: 1.95V

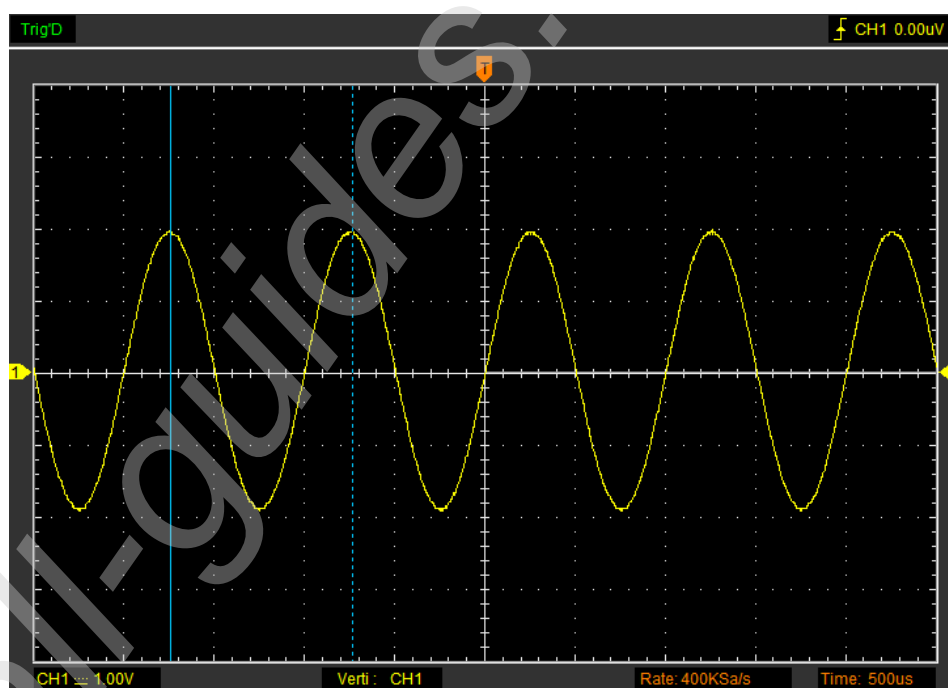
3) Vertical

The **Vertical** cursors appear as vertical lines on the display and measure the vertical parameters.

Do these steps:

1. Click "**Cursor**"-> "**Source**" select CH1 (select CH2 if you want measure CH2).
2. Click "**Cursor**"-> "**Type**", select **Vertical**.
3. Push left mouse button, and the vertical lines appear.
4. Drag the mouse button to the point you want to measure.
5. Release the left mouse button, the frequency and time difference will be shown at the status bar.

The **Vertical** cursor display window:



The **Vertical** cursor measure result display on status bar: Freq: 989.9Hz Time: 1.01ms

4) Horizontal

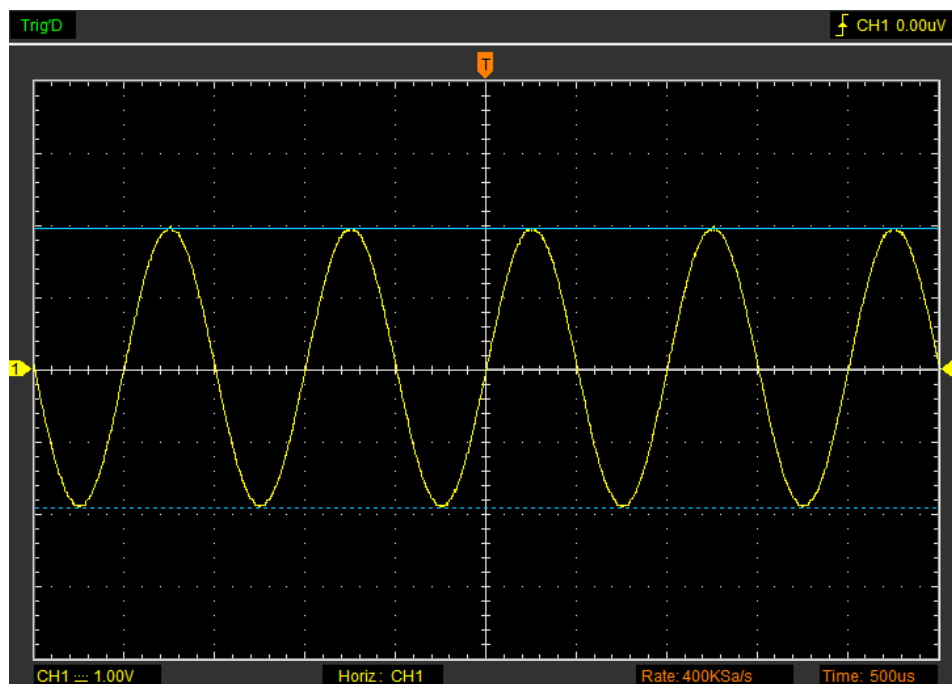
The **Horizontal** cursors appear as horizontal lines on the display and measure the horizontal parameters.

Do these steps:

1. Click "**Cursor**"-> "**Source**" select CH1 (select CH2 if you want measure CH2).
2. Click "**Cursor**"-> "**Type**", select **Horizontal**.
3. Push left mouse button, and the Horizontal lines appear.
4. Drag the mouse button to the point you want to measure.

5. Release the left mouse button, the voltage difference will be shown at the status bar.

The **Horizontal** cursor display window:



The **Horizontal** cursor measure result display on status bar: Volt: 3.88V

2.13.3 Automatic Measurement

Automatic Measurement: The oscilloscope performs all the calculations automatically in this mode. As this measurement uses the waveform record points, it is more precise than the graticule and cursor measurements. Automatic measurements show the measurement results by readouts which are periodically updated with the new data acquired by the oscilloscope.

Click **“Measure”** in main menu.

The oscilloscope provides 20 parametric auto measurements (12 voltage and 8 time measurements).

1. Source

The user can use the **“Source”** menu to select a measure source CH1 or CH2.

2. Vertical

Maximum: Voltage of the absolute maximum level, Measured over the entire waveform

Minimum: Voltage of the absolute minimum level, Measured over the entire waveform

Peak To Peak: Peak-to-peak = Max – Min, Measured over the entire waveform

Top : Voltage of the statistical maximum level, Measured over the entire waveform

Base: Voltage of the statistical minimum level, Measured over the entire waveform

Middle: Voltage of the 50% level from base to top

RMS: The Root Mean Square voltage over the entire waveform

Amplitude: Amp = Base - Top, Measured over the entire waveform

Mean: The arithmetic mean over the entire waveform

Cycle Mean: The arithmetic mean over the first cycle in the waveform

Positive OverShoot: Positive Overshoot = $(\text{Max} - \text{Top}) / \text{Amp} \times 100 \%$, Measured over the entire waveform

Negative OverShoot: Negative Overshoot = $(\text{Base} - \text{Min}) / \text{Amp} \times 100 \%$, Measured over the entire waveform.

3. Horizontal

Period: Time to take for the first signal cycle to complete in the waveform

Frequency: Reciprocal of the period of the first cycle to complete in the waveform

Rise Time: Time taken from lower threshold to upper threshold

Fall Time: Time taken from upper threshold to lower threshold

+Duty Cycle: Positive Duty Cycle = $(\text{Positive Pulse Width}) / \text{Period} \times 100\%$, Measured of the first cycle in waveform.

-Duty Cycle: Negative Duty Cycle = $(\text{Negative Pulse Width}) / \text{Period} \times 100\%$, Measured of the first waveform.

+Pulse Width: Measured of the first positive pulse in the waveform. The time between the 50% amplitude points

-Pulse Width: Measured of the first negative pulse in the waveform. The time between the 50% amplitude points

4. Clear Item

Clear all measure items on display screen.

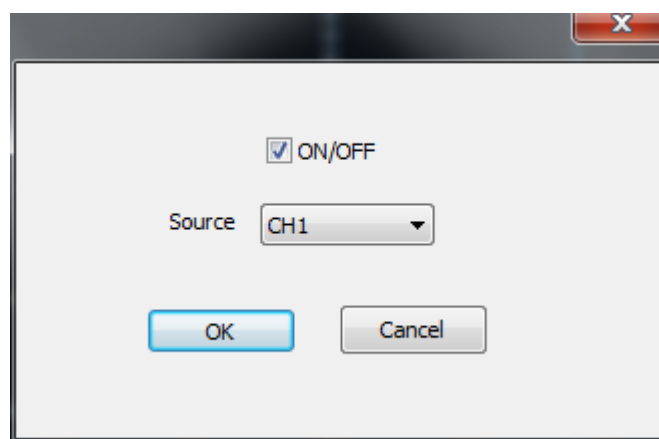
5. Edit Item

Click "**Measure->Edit Item**". User can select Maximum 8 measure options to measure.

Note: The results of the automatic measurements will be displayed on the bottom of the screen. Maximum 8 results could be displayed at the same time. When there is no room, the next new measurement result will make the previous results moving left, out of screen.

6. DVM

Click "**Measure->DVM**", the DVM dialog will be showed.

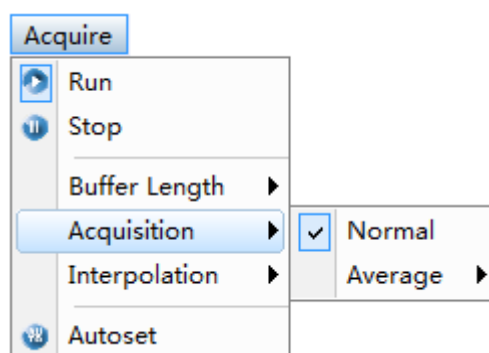


Turn on/off the DVM measure column, and it will be displayed on top right corner on the screen. Also you can drag it to any location.

DVM: Display RMS and frequency of measured waveform of CH1 or CH2.

2.14 Acquisition Modes

There are two acquisition modes: Normal, Average.



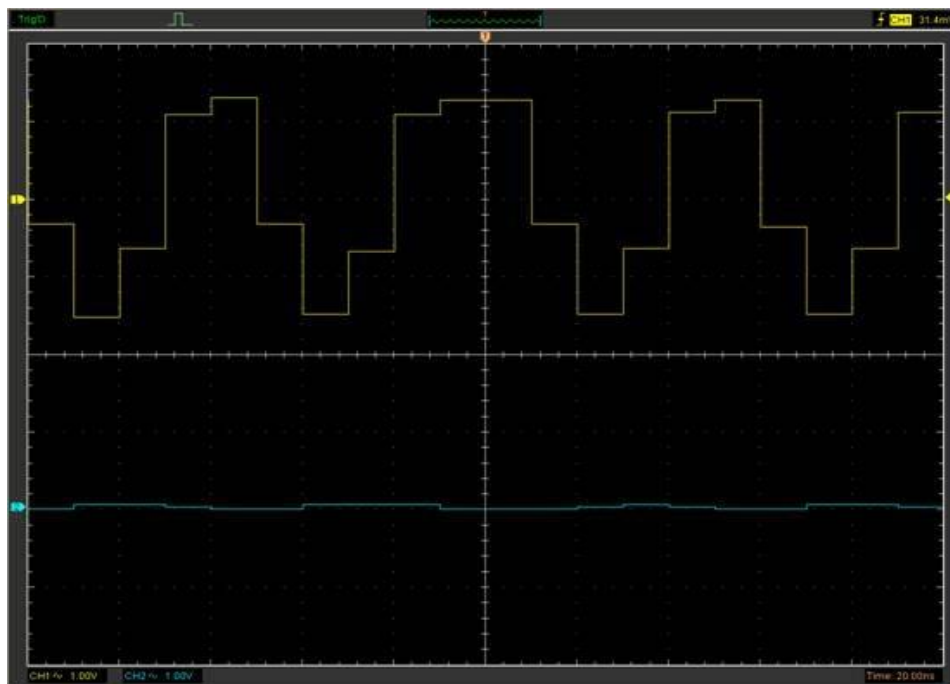
Normal: In this acquisition mode, the oscilloscope samples the signal in evenly spaced intervals to construct the waveform.

Average: In this acquisition mode, the oscilloscope acquires several waveforms, averages them, and displays the resulting waveform. You can use this mode to reduce random noise.

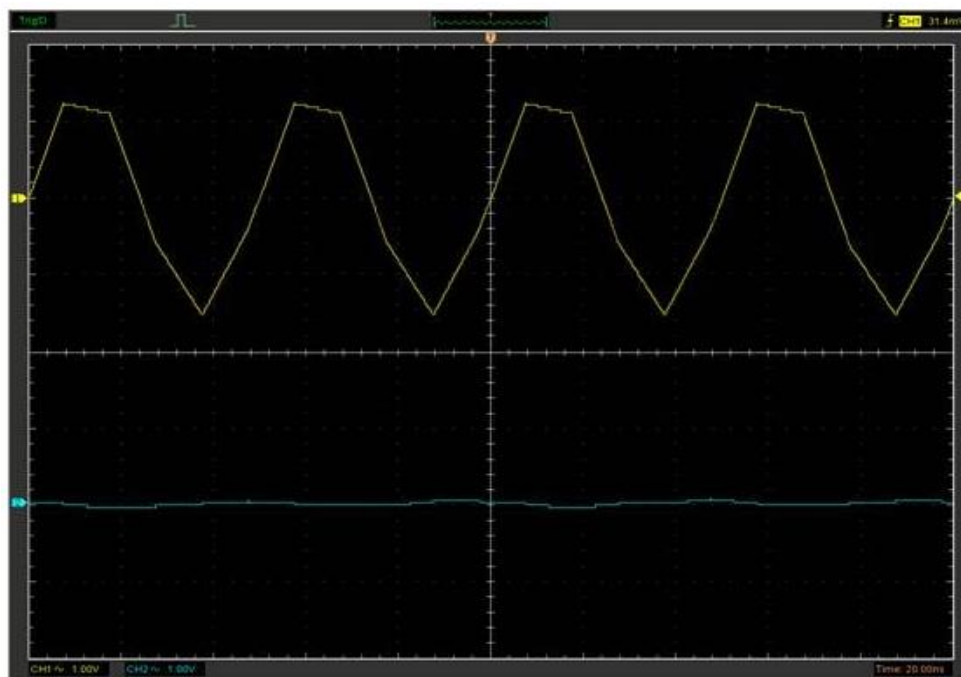
2.15 Interpolation

At the time base 50ns/div or faster, user can click “**Acquire->Interpolation**” to select 3 different interpolation modes to get waveforms of different smoothness.

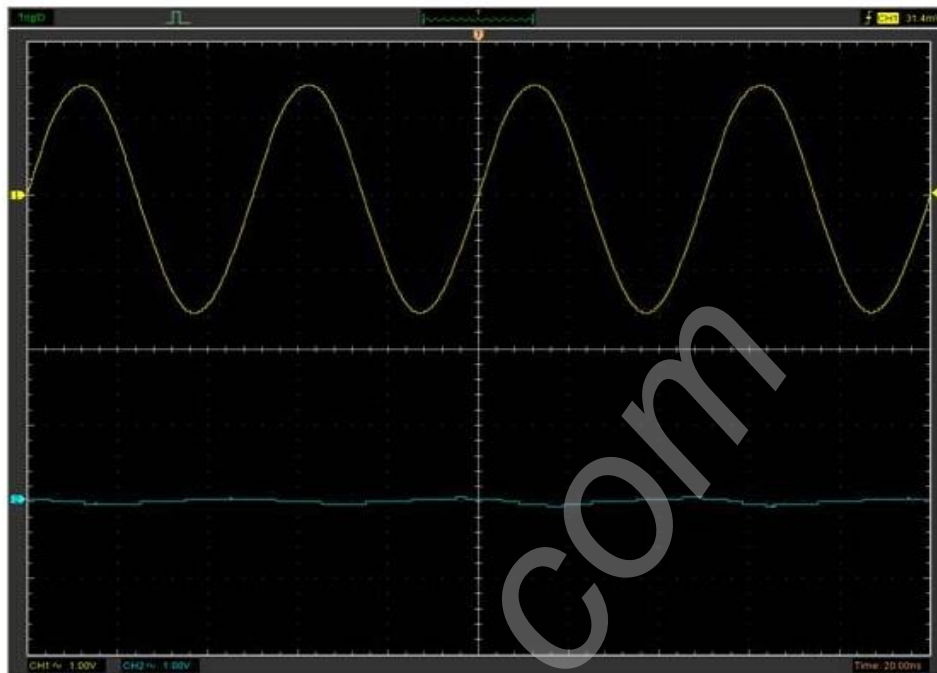
The **Step** Interpolation:



The **Linear** Interpolation:



The **Sin(x)/x** Interpolation:



Note: The default interpolation mode is $\text{Sin}(x)/x$.

Acquisition

When you acquire a signal, the oscilloscope converts it into a digital form and displays a waveform. The acquisition mode defines how the signal is digitized and the time base setting affects the time span and level of detail in the acquisition.

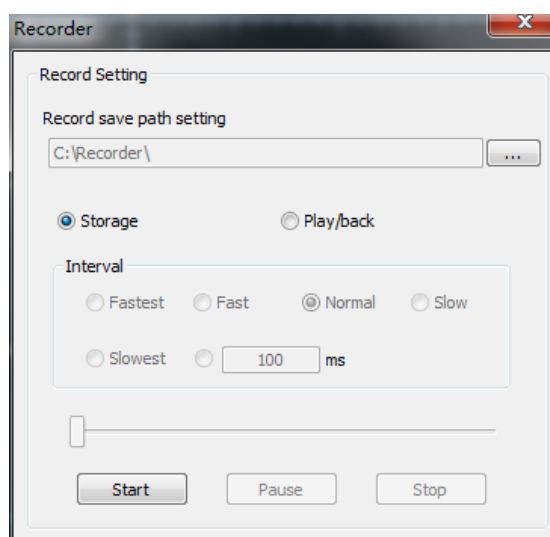
2.16 Utility Function

Click the “**Menu -> Utility**” to get into the “**Utility**” menu.

The “**Utility**” Functions:

- ◆ **Record**
- ◆ **Pass/Fail**
- ◆ **Calibrate**
- ◆ **Factory Setup**
- ◆ **Language**

2.16.1 Record



Before you select the save path every time, you need to manually create a new folder, and then select.

Storage

“Start” button: Start recording the waveform. And waveform files are saved in path.

“Stop” button: Stop record the waveform.

Play back

Interval: Set the time interval of play back. Also user can set time interval manually.

“Start” button: Start playing back the waveform.

“Stop” button: Stop play back the waveform.

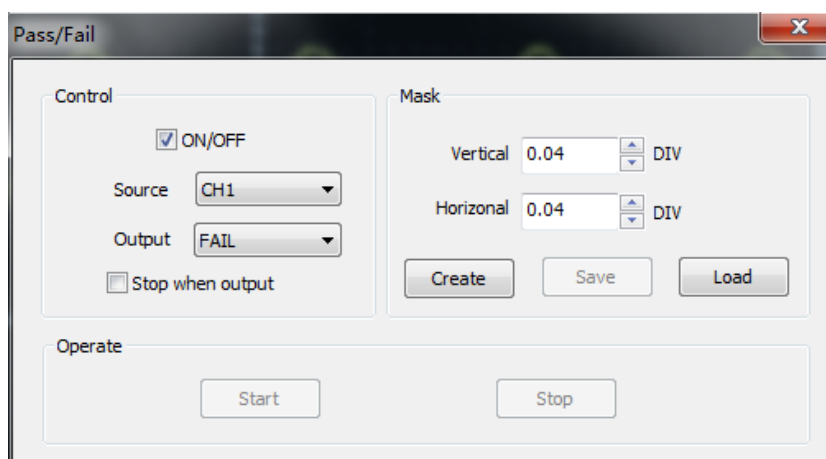
“Pause” button: Pause play back the waveform.

“Continue” button: Continue playing back the waveform.

2.16.2 Pass/Fail

Click **“Pass/Fail”** in **“Utility”** menu.

The **Pass/Fail** window appears:



The **Pass/Fail** function monitors changes of signals and outputs pass or fail signals by comparing the input signal with the pre-created mask.

Control Setting

ON/OFF: Turn on/off the **Pass/Fail**

Source: Select the **Pass/Fail** channel

Output: Select the **Pass/Fail** output condition

Stop When Output: If it was checked, the **Pass/Fail** will stop when output.

Mask Setting

Vertical: Set the vertical limit range

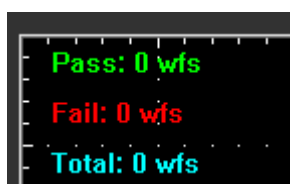
Horizontal: Set the horizontal limit range

“Create” button: Click this button to create **Pass/Fail** area according to the mask

“Save” button: Click this button to save the setups to file.

“Load” button: Click this button to load the saved setups file.

Pass/Fail information



Fail: Displays the fail waveform number

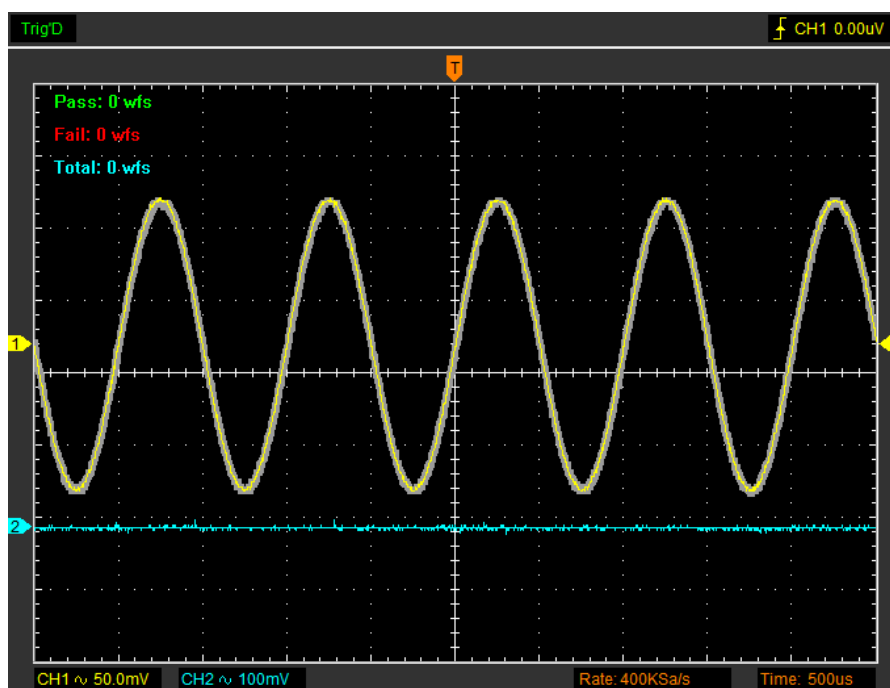
Pass: Displays the pass waveform number

Total: Displays the I total **Pass/Fail** waveform number

Operation

Click **“Start”** button to start the **Pass/Fail** test. Click **“Stop”** button to stop the **Pass/Fail** test.

The Pass/Fail function display:



NOTE: Pass/Fail function is unavailable in **X-Y** mode and **Roll** mode.

2.16.3 Self Calibration

Please refer to [Self calibration](#).

2.16.4 Factory Setup

Click **“Factory Setup”** in **“Utility”** menu to load default setups.

When you click the **Factory Setup** in **Utility** menu, the oscilloscope displays the CH1 and CH2 waveforms and removes all other waveforms.

The oscilloscope set up for normal operation when it is shipped from the factory and can be recalled at any time by user.

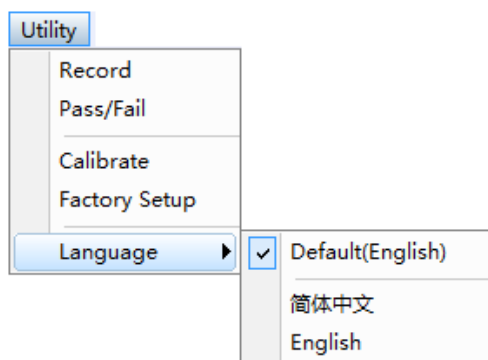
The Factory Setup function does not reset the following settings:

- ◆ Language option
- ◆ Date and time

2.16.5 Language

Click **“Language”** in **“Utility”** menu.

There are two languages in **“Language”** menu. The default language is English.



Note: The language of App depends on System language of iOS.

2.17 Print And Print Preview

Click **Print** in **File** menu to set the printer to print the current waveform.

1. Click **print** in **File** menu or Hotkeys "Ctrl+P" to set the current waveform.
2. Click the **Print Preview** in **File** menu to get into the Preview windows.

Click the **Print Preview** in **File** menu to get into the Preview window.

In **Print Preview** window, use the **Zoom In** button and the **Zoom Out** button to change the size of the waveform graph. Click the **Close** button to turn this window off and click the **Print** button to print the report.

Chapter 3 IOS Application

[User Interface](#)

[Menu System](#)

[Connect with Apple/Andriod](#)

[Set Horizontal System](#)

[Set Vertical System](#)

[Set Trigger System](#)

[Save and Load](#)

[Display System](#)

[Measure Signal](#)

[Acquire](#)

3.1 User Interface

The user's interface:



1. Channel information and Volt/Div



: CH1 is selected.



: CH2 is selected.

If the channel is select, you can see yellow background of CH1 or blue background of CH2.

2. Time/DIV: lead the setting of the time base parameters

Select the time setting for the beam to sweep one major division on the screen. By selecting different time/div settings it is possible to zoom in on the frozen waveform on the screen.

3. Battery level

4. Trigger State



A signal is triggerred and trigger source is CH1.



A signal is not triggerred and trigger source is CH1.



A signal is triggerred and trigger source is CH2.



A signal is not triggerred and trigger source is CH2.


5. Function menu

RUN: Run the application. Tap RUN menu again to stop it.

AUTO: Automatically set the oscilloscope controls to generate a usable display of the input signals. Refer to the following table for relative content.

Menu: Main menu.

6. Trigger Level

Tap the trigger level icon  and move up or move down, the trigger level value will display on the screen.

7. Horizontal Trigger position

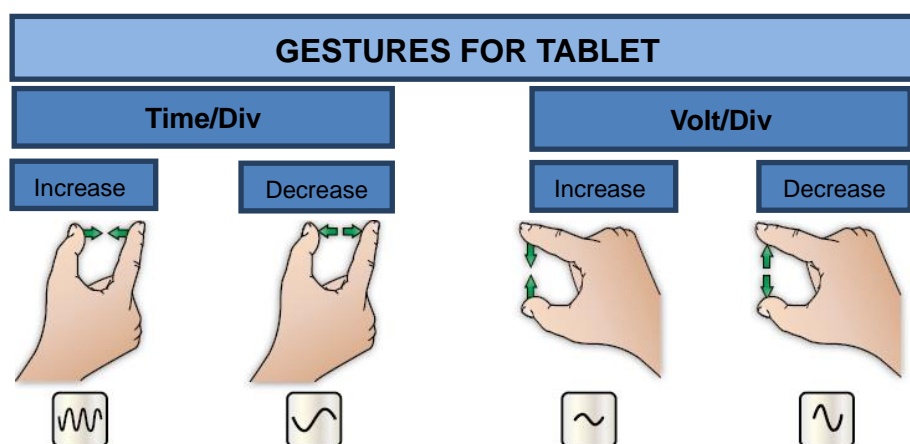
Tap the horizontal trigger position icon and move left or move right, the horizontal trigger position will display on the screen. Click “Trigger->Indicator” in main menu and slid the slider to enable the function.

8. CH2 Zero level

The marker shows the reference points of the displayed waveforms of CH1.

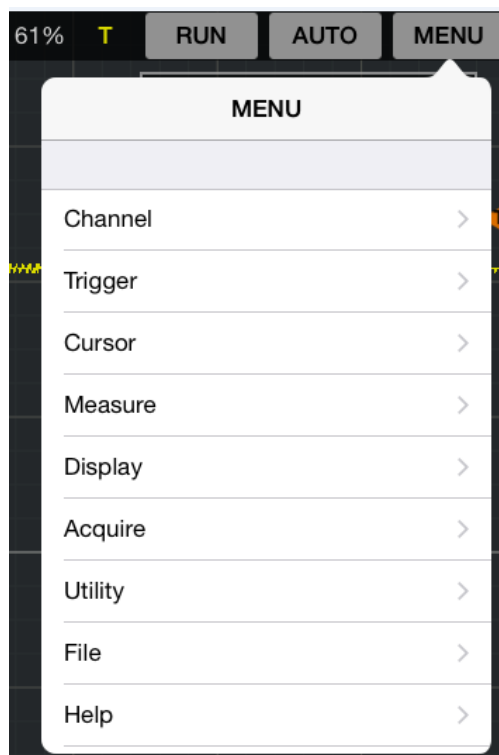
9. CH1 Zero level

The marker shows the reference points of the displayed waveforms of CH2.



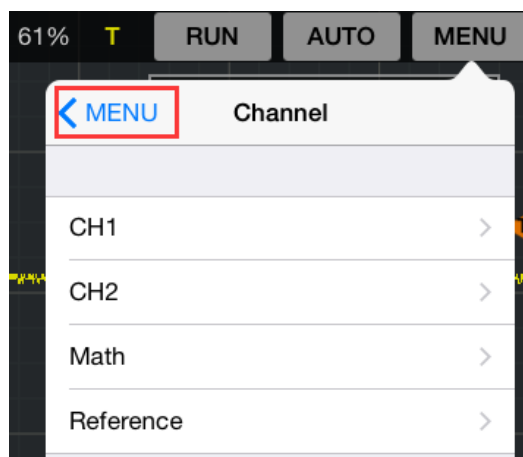
3.2 Menu System

The Main Menu:



1. **Channel:** CH1, CH2, Math and Reference setting
2. **Trigger:** Set trigger system
3. **Cursor:** Set cursor measure type
4. **Measure:** Set measurement parameters
5. **Display:** Change wave display type and color.
6. **Acquire:** Start, Stop, Autoset and other operation setting
7. **Utility:** Utility setting
8. **File:** Save and load waveform data and setup.
9. **Help:** Turn on help file

After enter submenu, please tap "<MENU" to return to main menu.



3.3 Connect IDSO With Apple

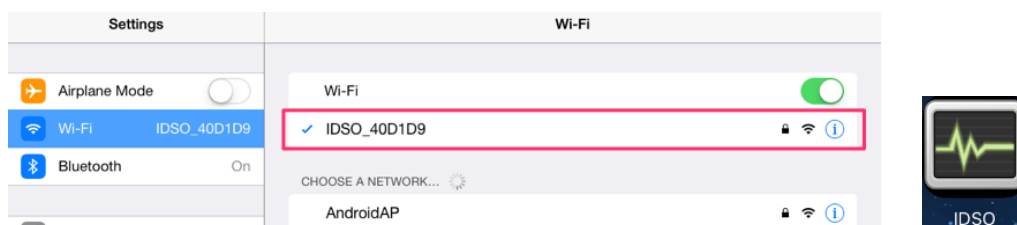
IDSO device work in direct mode

When in the direct mode, Apple/Android device is connect with the IDSO device via Wi-Fi hotspot which created by IDSO device. IDSO device works in this mode in default.

1. Power on the IDSO.

Keep pressing power button until power indicator lights up.

2. The indication of communication USB/Wi-Fi is slow blinking. Hotspot, creates own wireless local area network (WLAN).
3. Connect your Apple/Android device with Wi-Fi hotspot. Click “**Setting→Wi-Fi**” options, then select Network Name of the IDSO device. You can get Network Name from the label of IDSO’s shell.



4. Connect the IDSO device to a signal source.
5. Tap the IDSO app to open it. Then click “RUN” button. The scope is ready to use.
If you can’t connect it successfully, please do as the followings:
 - 1) Close the IDSO APP.
 - 2) Long press the Easylink button until the “USB/Wi-Fi” LED is off, release the Easylink button. IDSO device will reboot automatically. If the IDSO device doesn't reboot automatically, reboot it manually.
 - 3) After rebooted, the IDSO device will be in direct mode.

IDSO device work in Indirect Mode

When in the indirect mode, IDSO device and iOS device communicate via a third-path Wi-Fi router.

1. If the IDSO APP is running on background, please close it firstly.
2. Connect the iOS device to the Wi-Fi router. (Setting→Wi-Fi)
3. Open IDSO App, MENU->Utility->Indirect Mode, input the router's password, Press Apply button.
4. Long press the Easylink button until the “USB/Wi-Fi” LED off, release Easylink button. IDSO device will reboot automatically. If the IDSO device doesn't reboot automatically, reboot it manually.

After rebooted, IDSO will connect to the Wi-Fi router. IDSO App should pop up a tip.

3.4 Set Horizontal System

3.4.1 Horizontal Setting

1. **Time/DIV**: Use gesture to set the time base.
2. **Format**: leads the setting of the horizontal format parameters.

Format

Set the waveform display format (**Y-T**, **X-Y**). The default is YT mode.

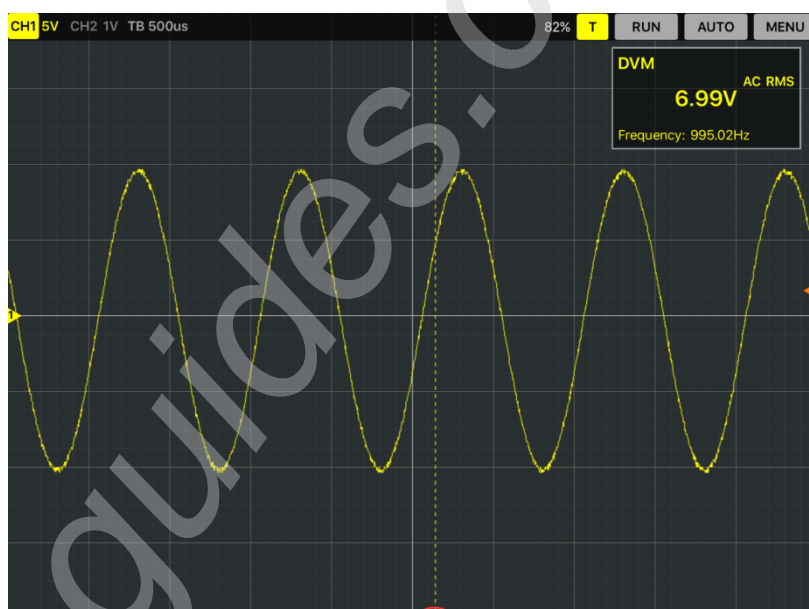
Y -T: Shows the relative relation between vertical voltage and horizontal time.

X -Y: Show CH1 value at X axis; CH2 value at Y axis. The XY format is used to analyzer phase difference, such as those represented by Lissajous patterns. Click "**Utility->XY Mode**" in main menu.

Horizontal Trigger Position

Tap the horizontal trigger position icon and move left or move right, the horizontal trigger position will display on the screen. Click "**Trigger->Indicator**" in main menu and slid the slider to enable the function.

Double click the icon to reset the trigger point back to the screen center.



3.4.2 The Application of the X-Y Operation

Select X-Y format in **Utility** menu. More information, please refer to [The Application of the X-Y Operation](#).

3.5 Set Vertical System

Click "**Channel**" in main menu.

Select Channel: User can select the channel CH1, CH2, Math and Reference.

Channel Setting:

ON/OFF: Tap CH1 or CH2 icon on the top left corner of the screen.



Volt/DIV: Use gestures to set volt/div.



Coupling: Select the channel coupling DC/AC/GND.

Probe: Select the attenuation factor for the probe. To check the probe attenuation setting, toggle the probe menu to match the attenuation factor of the probe.

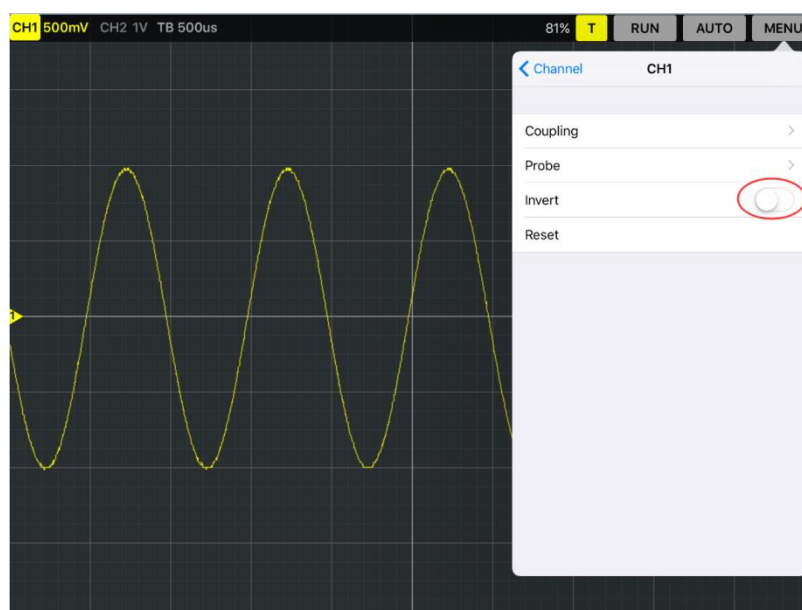
More information, please refer to [Set Vertical System](#).

Invert

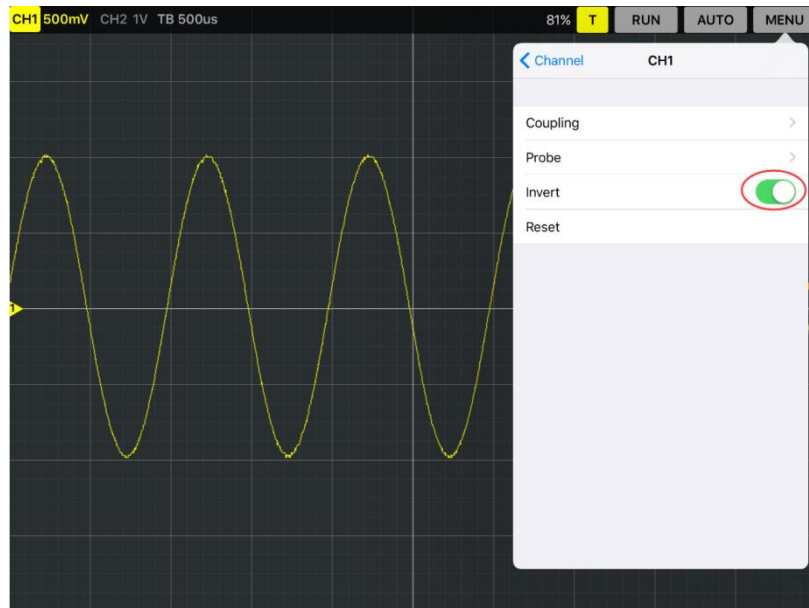
The invert function turns the displayed waveform 180 degrees, with respect to the ground level.

When the oscilloscope is triggered on the inverted signal, the trigger is also inverted.

The following picture shows the waveform before inversion:



The following picture shows the waveform of inversion:



Reset

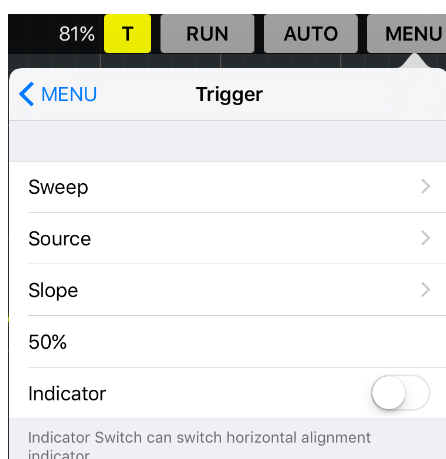
Double click the icon to reset the waveforms to the vertical center position on the screen.

Move the waveform

Drag Vertical Position icon   to move waveform up and down on the screen.

3.6 Set Trigger System

Click “Setup-> Trigger” menu.



Trigger Setting: Please refer to [set trigger system](#).

50%: The trigger level is set to the vertical midpoint between the peaks of the trigger signal.

Trigger Level

Trigger Level icon: User can also drag the trigger icon  on screen to adjust the trigger level.

Double click trigger icon to set trigger level to the vertical midpoint between the peaks of the trigger signal.

3.7 Save and Load

3.7.1 Save/Load File

Click **"File->Waveform Data"** in main menu to save waveform data as CSV file and load saved waveform data.

Export Data

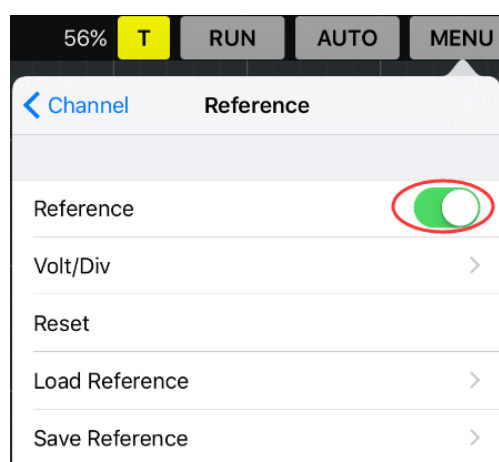
Click **"File->Waveform Data->Export Data"** in main menu to save the current waveform data as CSV file.

Import Data

Click **"File->Waveform Data->Import Data"** in main menu to load saved waveform data file.

3.7.2 Save/Load Reference

Click **"Channel->Reference"** in main menu.



The Reference Channel Function:

On/Off: Slid the slider to enable the function.

Volt/DIV: Channel the resolution of the reference channel.

Reset: Double click the icon to reset the waveforms to the vertical center position on the screen.

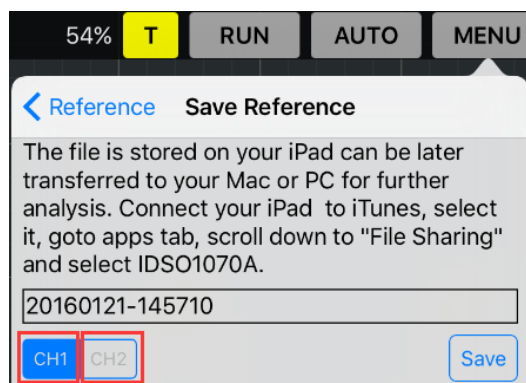
Load: Load the reference waveform from the “.rfc” file from your computer.

Save: Save the current reference waveform to your computer as “rfc” format.

You can change the vertical scale of a waveform. The waveform display will contract or expand relative to the reference level.


Save

Click **“Save Reference”** to save the waveform to *.rfc file. The saved source window appears.

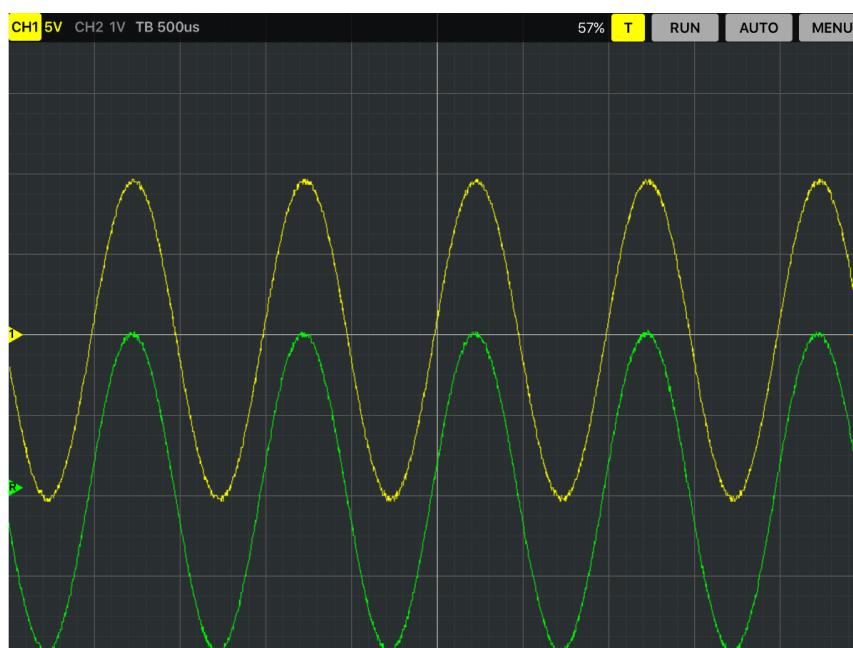


Selected the saved source, then click **“Save”** button to save reference.

Load

Click **“Load Reference”** to load the “*.rfc” file that was selected. Then click “Load” button to load selected file. You can tap  and drag the reference waveform up and down.

The Reference Waveform Display Window:



3.7.3 Save/Load Setup

Save Setup

The oscilloscope software saves the current setup before you close the oscilloscope software. The oscilloscope recalls this setup when you run the software next time. You can use the “**File->Setups->Save Setups**” in main menu to permanently save setting.

Load Setup

The oscilloscope can recall the last setup before the oscilloscope software was running, any saved setups, or the factory setup. You can use the “**File->Setups->Load Setups**” in main menu to permanently recall setup.

3.8 Display System

Please refer to [Display System](#).

3.9 Measure Signal

3.9.1 Cursor

Click “**Cursor**” in main menu. More information, please refer to [Cursor](#).

Please slid the slider to enable or unable the cursor function.

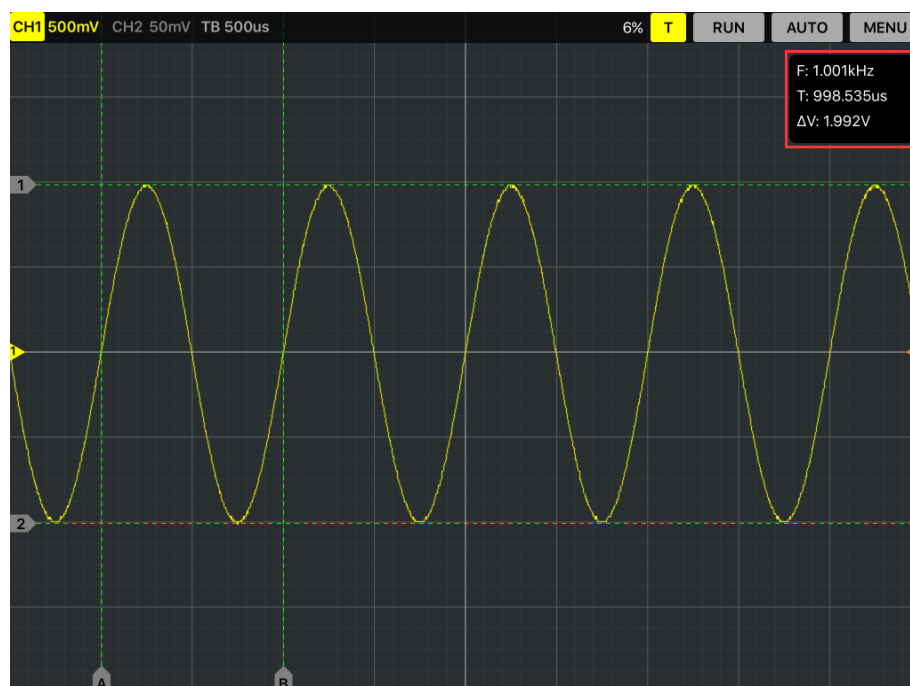
1. Source

The user can set the source to **CH1**, **CH2**, **MATH** and **FFT**.

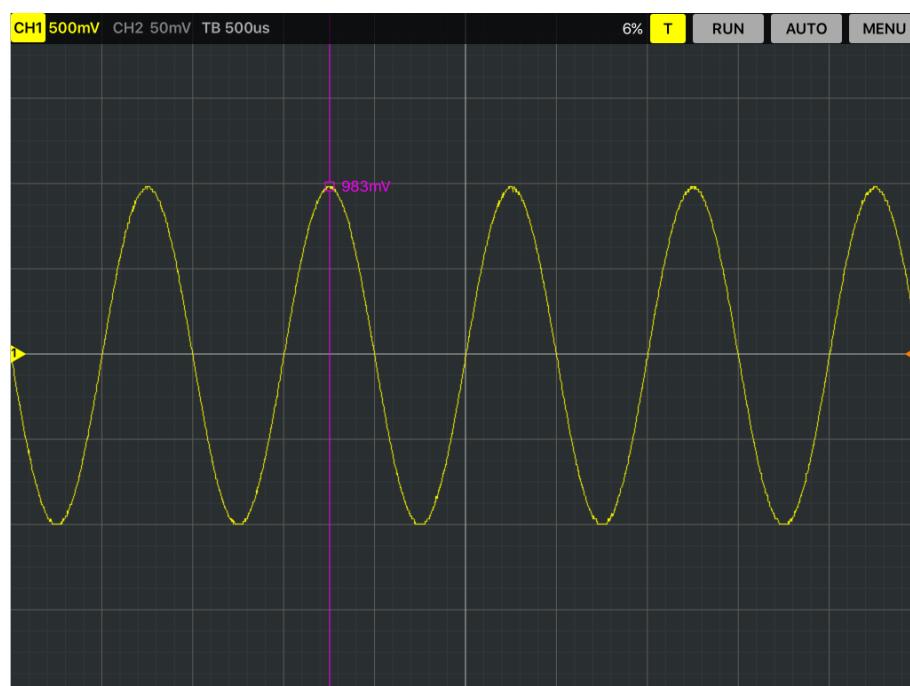
2. Type

There are two types of cursors: **Cross** and **Trace**.

The **Cross** cursor display window:



The **Trace** cursor display window:



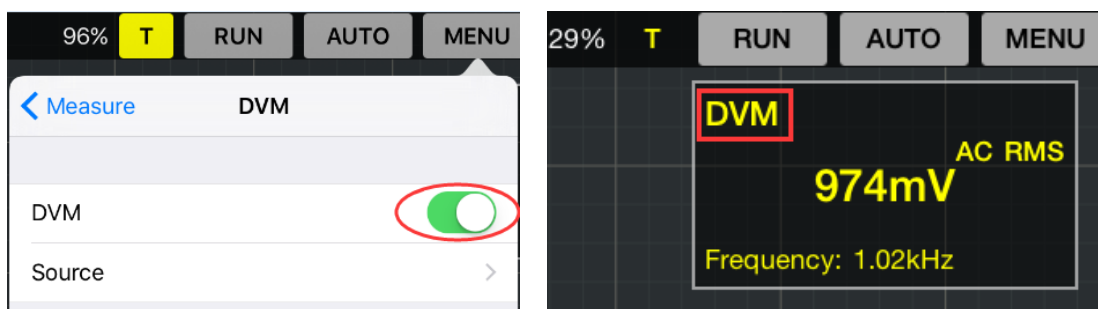
3.9.2 Measure

Click “**Measure**” in main menu.

DVM

DVM: Display RMS and frequency of measured waveform of CH1 or CH2.

Slid the slider to enable the DVM function. You can see it on top right corner on the screen. Also you can tap the DVM column and drag it to any location.



More information, please refer to [Measure](#).

3.10 Acquire

3.10.1 Capture Mode

There are three capture modes **Normal**, **Roll** and **Scan** mode.

Please refer to [Set Horizontal System Y-T Format](#).

3.10.2 Interpolation

Please refer to [Interpolation](#).

Chapter 4 Android Application

[User Interface](#)

[Menu System](#)

[Connect with Andriod](#)

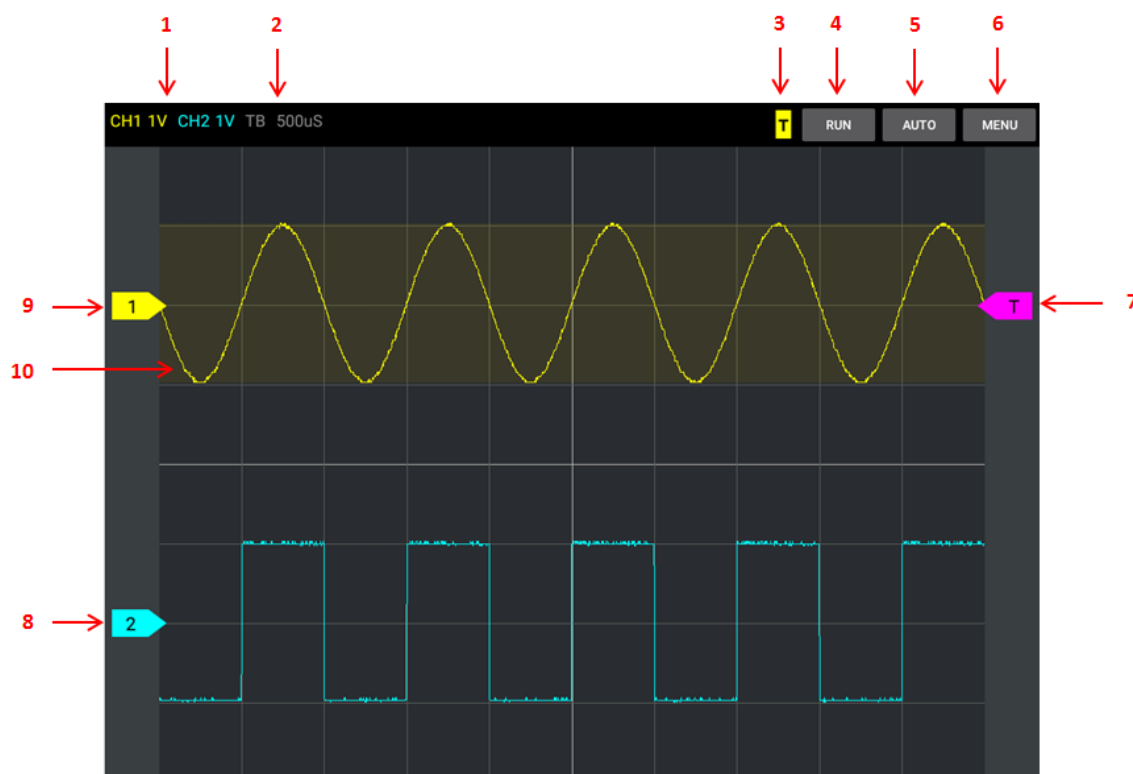
[Set Vertical System](#)

[Set Trigger System](#)

[Measure Signal](#)

4.1 User Interface

The user's interface:



1. Channel information and Volt/Div

2. Time/DIV: lead the setting of the time base parameters

Select the time setting for the beam to sweep one major division on the screen..

3. Trigger State



A signal is triggered.



A signal is not triggered.

4. Function menu


RUN: Run the application. Tap RUN menu again to stop it.

5. AUTO

Automatically set the oscilloscope controls to generate a usable display of the input signals.

6. Menu

7. Trigger position

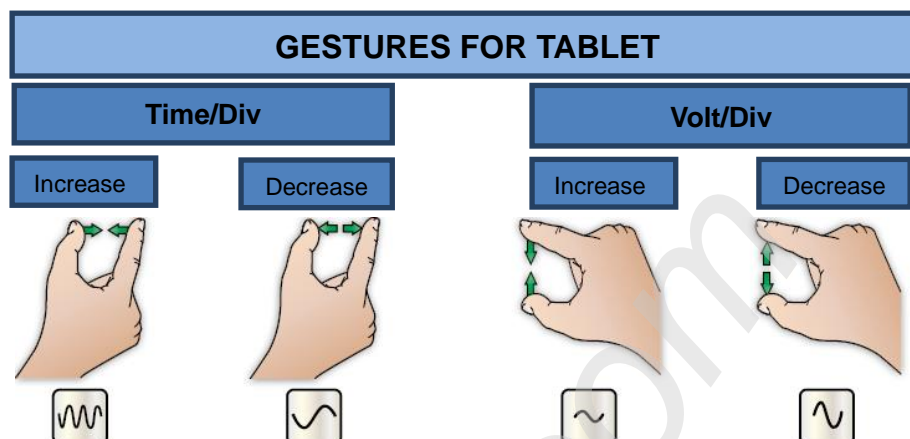
Tap the trigger level icon  and move up or move down, the trigger level value will display on the screen.

8. CH2 Zero level

The marker shows the reference points of the displayed waveforms of CH1.

9. CH1 Zero level

The marker shows the reference points of the displayed waveforms of CH2.

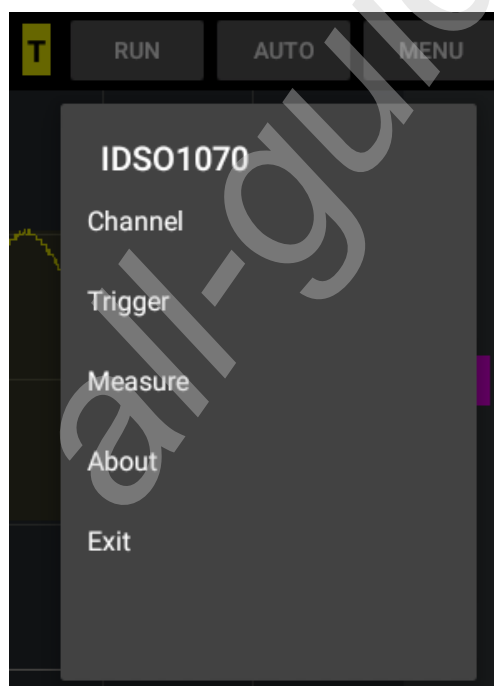


10. CH1 waveform

Click on the waveform to select the channel. If the channel is select, you can see yellow background of CH1 waveform or blue background of CH2 waveform.

4.2 Menu System

The Main Menu:

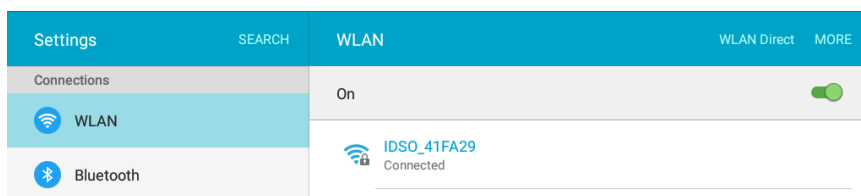


1. **Channel:** CH1, CH2 settings
2. **Trigger:** Set trigger system
3. **Measure:** Set measurement parameters
4. **About:** Software information
5. **Exit:** Exit the software.

4.3 Connect IDSO with Android

When in the direct mode, Android device is connect with the IDSO device via Wi-Fi hotspot which created by IDSO device. IDSO device works in this mode in default.

1. Power on the IDSO.
Keep pressing power button until power indicator lights up.
2. The indication of communication USB/WI-FI is slow blinking. Hotspot, creates own wireless local area network (WLAN).
3. Connect your Android device with Wi-Fi hotspot. Click “**Setting→WLAN**” options, and then select Network Name of the IDSO device. You can get Network Name from the label of IDSO’s shell.



4. Connect the IDSO device to a signal source.
5. Tap the IDSO app to open it. Then click “RUN” button. The scope is ready to use.
If you can’t connect it successfully, please do as the followings:
 - 1) Close the IDSO APP.
 - 2) Long press the Easylink button until the “USB/WI-FI” LED is off, release the Easylink button. IDSO device will reboot automatically. If the IDSO device doesn't reboot automatically, reboot it manually.
 - 3) After rebooted, the IDSO device will be in direct mode.

4.4 Set Vertical System

Click “**Channel**” in main menu.

Select Channel: User can select the channel CH1 and CH2.

Channel Setting:

ON/OFF: Tap CH1 or CH2 icon on the top left corner of the screen.



Volt/DIV: Use gestures to set volt/div.



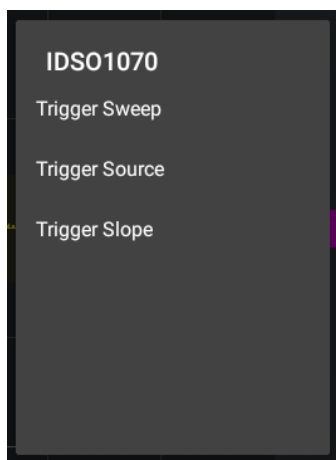
Coupling: Select the channel coupling DC/AC/GND.

Move the waveform up and down

Drag Vertical Position icon   to move waveform up and down on the screen.


4.5 Set Trigger System

Click “**Setup-> Trigger**” menu.



Trigger Setting: Please refer to [set trigger system](#).

Trigger Level

Trigger Level icon: User can also drag the trigger icon  on screen to adjust the trigger level.

4.6 Measure Signal

Click “**Measure**” in main menu.

More information, please refer to [Measure](#).

Chapter 5 Appendix

- ◆ [Appendix A: Specifications](#)
- ◆ [Appendix B: General Maintenance](#)
- ◆ [Appendix C: Accessories](#)

Appendix A: Specifications

Specifications Table:

Acquisition	
Channels	2
Bandwidth	70 MHz analog bandwidth
Sample rate	250MSa/s(Single channel), 125MSa/s(Dual channel)
Mode	Normal, Peak Detect, Average(PC Software)
Input	
Input Coupling	AC, DC,GND
Input Impedance	1M Ω , 25pF
Probe Attenuation Factors	1X, 10X, 100X, 1000X
Max. Input Voltage	35Vpk (DC + peak AC < 10 kHz, without external attenuation)
Horizontal	
Scanning Speed Range(Sec/div)	5ns/div ~ 500s/div(1-2-5 sequence)
Delay Time Accuracy	± 50 ppm(any interval ≥ 1 ms)
Waveform Interpolation	Step, Linear, Sin(x)/x
Memory Depth	Max. 6K
Vertical	
Vertical resolution	8 bits/channel
Gain range	10mV ~5V/div @ x1 probe(1,2,5 sequence) 100mV ~ 50V/div @ x10 probe 1V ~ 500V/div @ x100 probe 10V ~ 5KV/div @ x1000 probe
Position Range	± 4 division
BandWidth	20MHz
Lower Frequency Response(-3dB)	≤ 10 Hz(at input BNC)
Rise Time at BNC	≤ 5 ns
DC accuracy	$\pm 3\%$
Math	Addition, Subtraction, Multiplication, Division
FFT	Rectangular, Hanning, Hamming, Blackman Window
Trigger	
Trigger Type	Edge trigger
Trigger Mode	Auto, Normal and Single
Source	CH1, CH2, EXT

Sensitivity	0.02 div increments
Trigger level	± 4 divisions

Measurement		
Cursor	Amplitude difference between cursors (ΔV)	
	Time difference between cursors (Δt)	
	Reciprocal of Δt in Hertz ($1/\Delta t$)	
	(Cross, Trace, Horizontal, Vertical)	
Auto Measure	Voltage Measurement	Vp-p, Vmax, Vmin, Vmean, Vamp, Vtop, Vbase, Vmid, Vrms, Vcrms, Preshoot, Overshoot
	Time Measurement	Frequency, Period, Rise Time(10%~90%), Fall Time(10%~90%), Positive Width, Negative Width, Duty Cycle

Appendix B: General Maintenance

General Care

Do not store or leave the oscilloscope where the device will be exposed to direct sunlight for long periods of time.

Caution

To avoid damages to the device or probes, do not expose them to sprays, liquids or solvents.

To avoid damages to the surface of the device or probes not use any abrasive or chemical cleaning agents.

Cleaning

In order to keep the instrument clean ,you need to check often whether the channels are dusty or not .Please clean the instrument surface and the channel connecting point as per the following cautions(Make sure the device disconnect form all power sources):

1. Remove loose dust on the outside of the oscilloscope and probes with a lint-free cloth.
2. Use a soft cloth dampened with water to clean the device.

Appendix C: Accessories

All the accessories listed below are standard accessories for the oscilloscope:

- ◆ Probex2 (1.5m), 1:1, (10:1) Passive Probes
- ◆ A USB Cable
- ◆ A Power Adapter
- ◆ A CD