Hantek®

Hantek Electronic co.,Ltd.
No.177 zhuzhou road(huite industry city),
QingDao,China

CE

DSO 3000SERIES

DIGITAL OSCILLOSCOPE

USER'S MANUAL

3064A

www.hantek.com.cn

DSO3000 SERIES
DIGITAL OSCILLOSCOPE

USER'S MANUAL

3064A

Content

General Safety Summary 3
CHAPTER 1: Getting Started..... 4
System Requirement.....5
Install Software.....6
Install Driver.....11
General Features.....16
General Check.....17
Probe Compensation.....17
Functional Check.....19
Seft Calibration.....21
Accessaries.....22
CHAPTER 2: Operating Basics.....23
The User's Inerface.....24
The Vertical System.....30
The Horizontal System.....32
The Trigger System.....33
Input Connectiors.....35
CHAPTER 3: Understanding Function.....36
Set Ocsilloscope.....37
Set Vertical System.....38
Set Horizontal System.....47
Set Trigger System.....55
Save/Load.....55
Utility Function.....57
Measure Signal.....65
The Display System.....73
Zoom In/Out and Drag waveforms.....78
Interpolation.....80
Acquisition.....83
Print.....84
CHAPTER 4: Application Example.....86
Simple Measurement.....87
Pass/Fail Test.....89
Capturing a Single Shot Signal.....93
The Application of The X-Y.....94
Taking Cursor Measurements.....97
Save Data.....101
CHAPTER 5:Wave Generator 103
CHAPTER 6: Appendix.....116
Appendix A.....117
Appendix B.....119

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**

■ **Use Proper Power Cord.** Use only the power cord specified for this product and certified for the country of use.

■ **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 informa tion before making connections to the product.

■ **Use Proper Probe.** To avoid shock hazard, use a properly rated probe for your meas urement.

■ **Avoid Circuit or Wire Exposure.** Do not touch exposed connections and compo nents 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.**

Chapter 1 Getting Start

The oscilloscope is small, lightweight, no external power required, portable oscilloscopes! The oscilloscopes 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.

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

- System Requirements
- Install your product
- General Features
- General Check
- Perform a probe check and compensate probes
- Match your probe attenuation factor
- Use the self calibration routine
- Accessories

System Requirement

To run the oscilloscope software, the needs of computer configuration are as follows:

Minimum System Requirements

Operating System
Window NT/2000/XP/VISTA/Win7

Processor
Upwards of 1.00G processor

Memory
256M byte

Disk Space
500M disk free space

Screen resolution
800 x 600

Recommended Configuration

Operating System
Windows XP SP3 Syst**em**

Processor
2.4G Processor

Memory
1G Byte Memory

Disk Space
80G Disk Space

Screen resolution
1024 x 768 or 1280 x 1024 resolution

DPI Setting
Normal Size (96DPI)

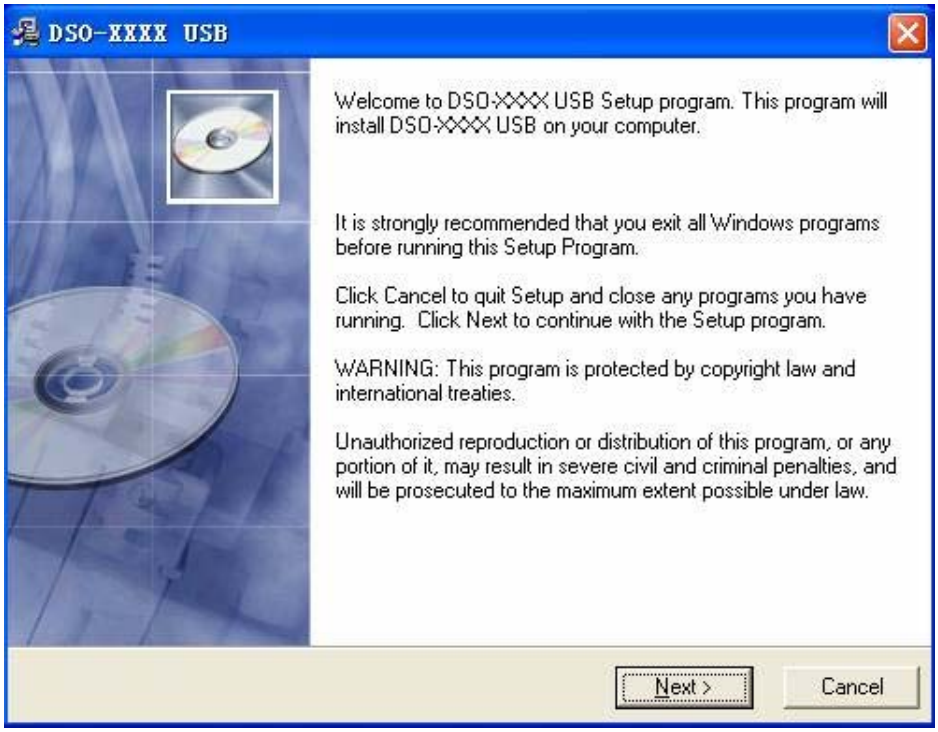
Install Software

Caution: You must install the software before using the oscilloscope.

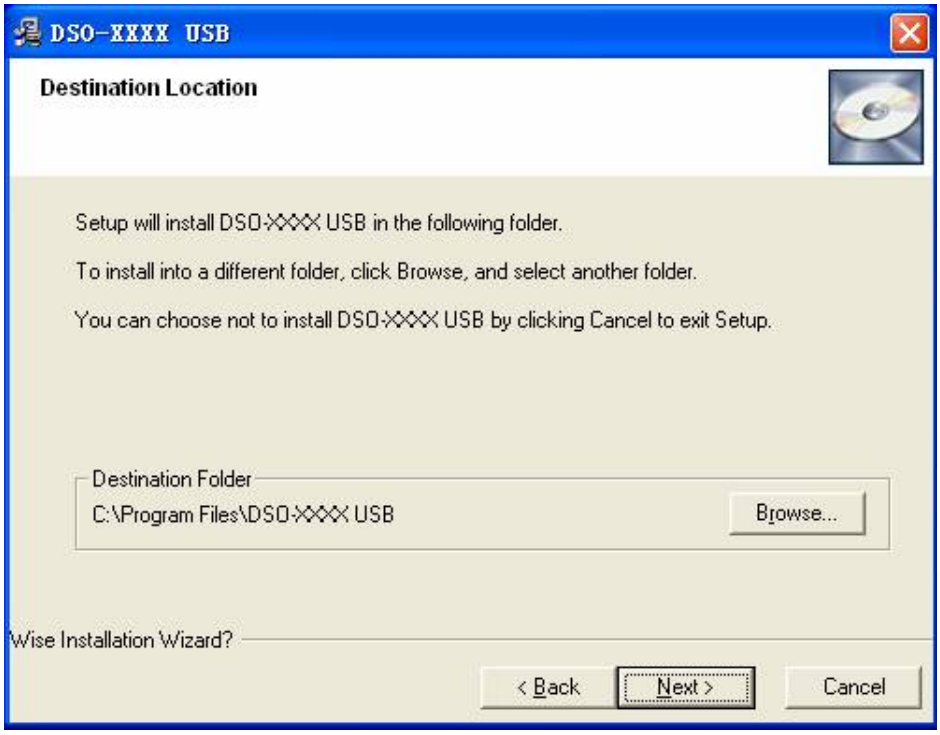
1. While in Windows, insert the installation CD into the CD-ROM drive.

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

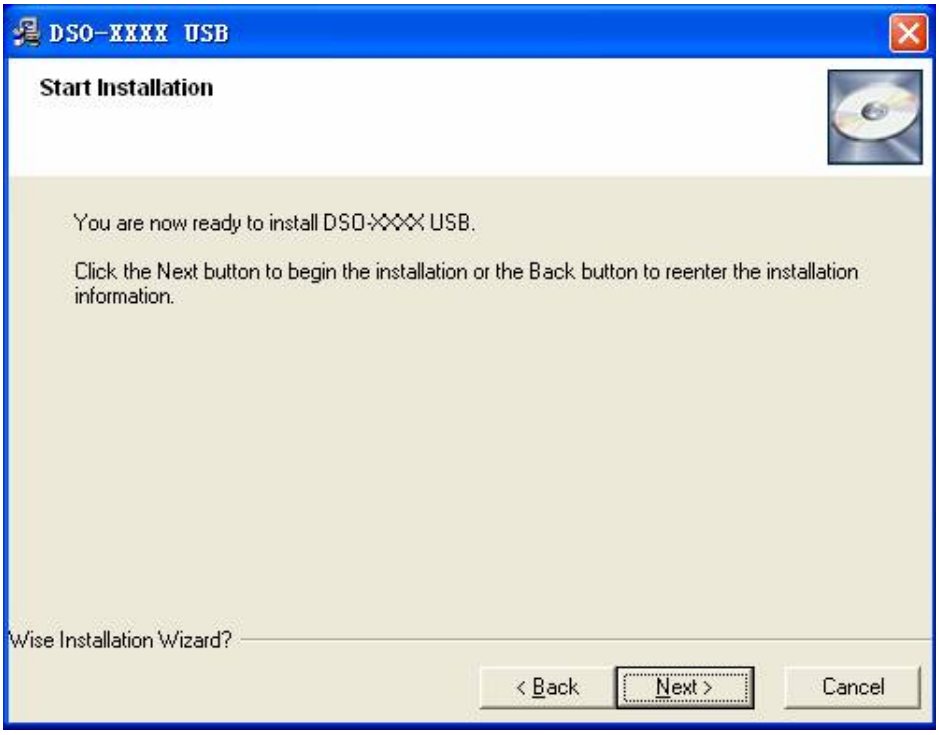
3. The software Installation is started. Click 'Next' to continue.



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

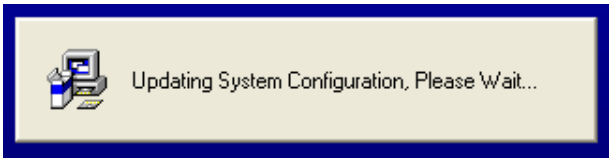


5. Check the setup information. Click Next to start copying of files.

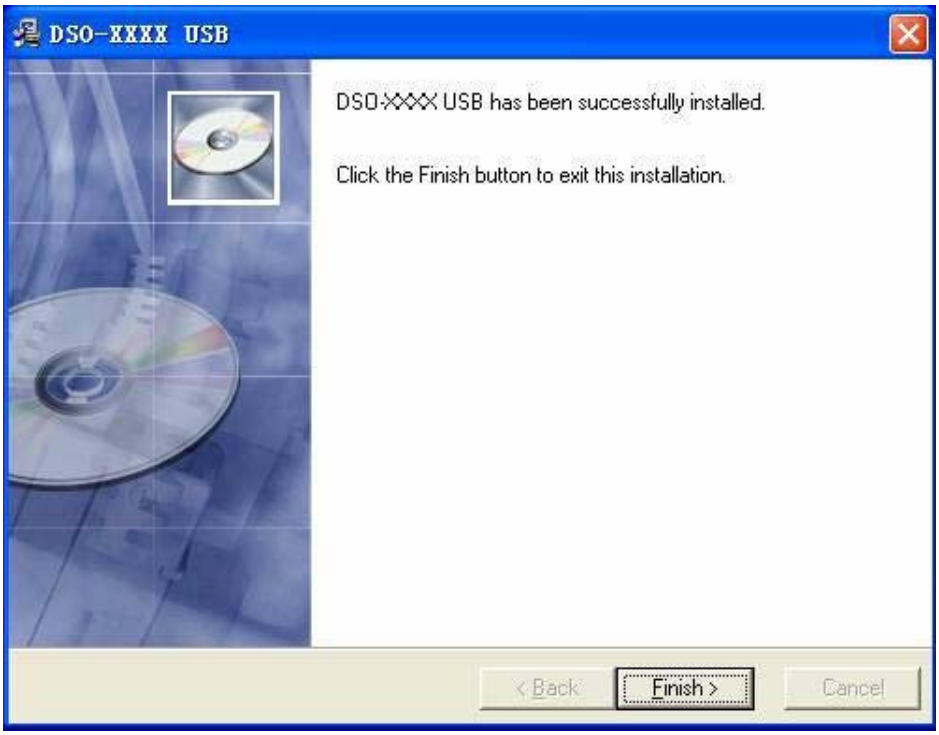


6. This Status dialog is displayed during copying of files.

7. Updating Your System Configuration.



8. The installation is complete.


DSO-XXXX USB
DSO-XXXX USB has been successfully installed.
Click the Finish button to exit this installation.
< Back
Finish >
Cancel

Install Driver

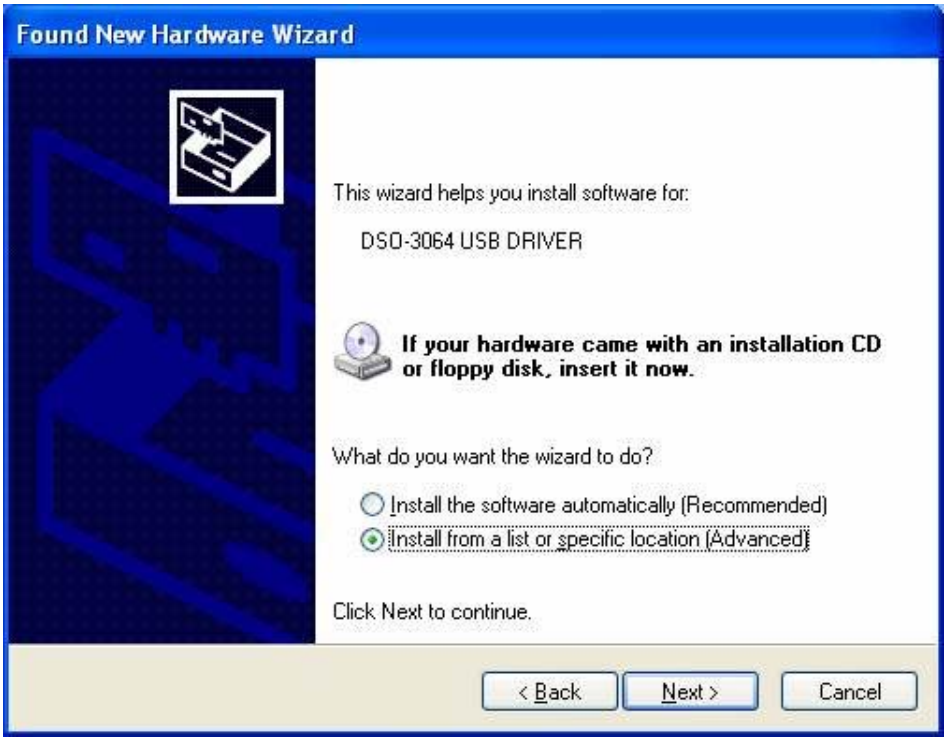
Example: DSO-3064 USB

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

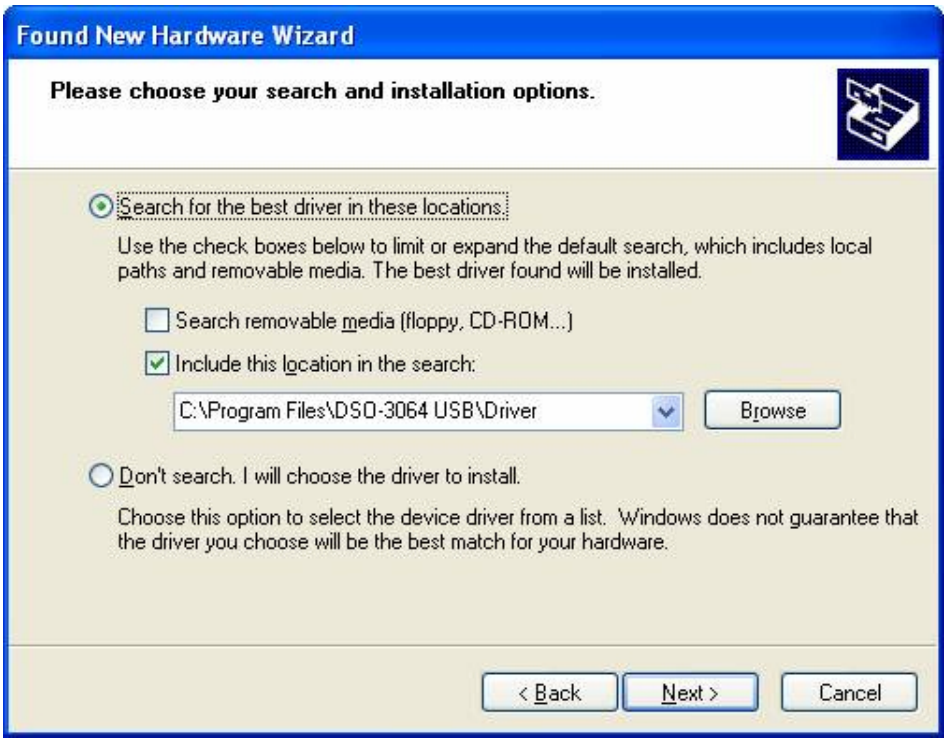
2. Connect the B-Type Plug of USB cable to DSO-3064 USB'S USB port.

3. New hardware is found.

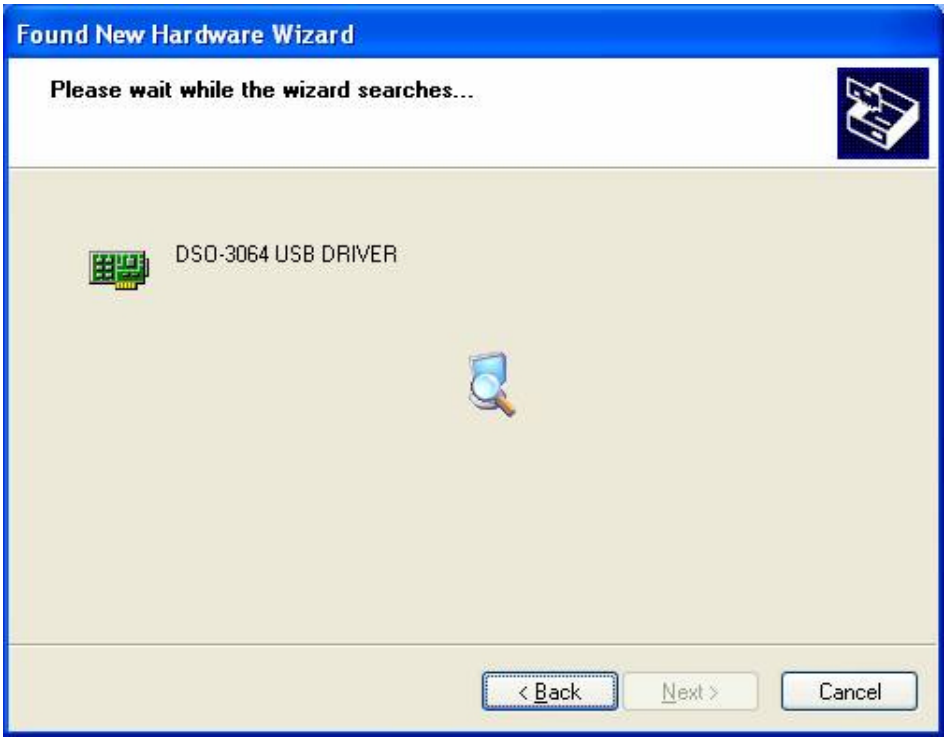
4. New hardware search wizard starts.



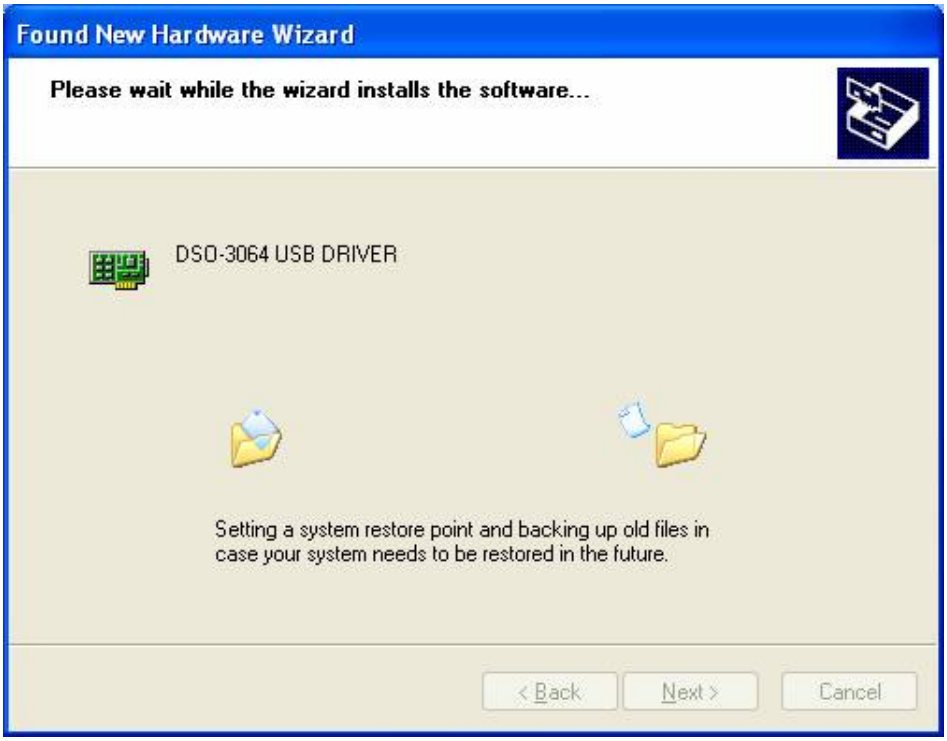
5. Select the specific location.



6. New hardware search wizard starts to search the driver.



7. New hardware wizard installs “DSO-3064 USB DRIVER ”.



8. The wizard has finished installing for “DSO-3064 USB DRIVER”.

General Features

Product features:

- **Four Channel, Bandwidth:**
  60MH
- **Maximum real-time sample rate:**
  200MSa/s
- **Memory depth:**
  10K-16M points
- **Automatic setup for ease of use (AUTOSET);**
- **Pass/Fail;**
- **Built-in Fast Fourier Transform function(FFT);**
- **20 Automatic measurements;**
- **Automatic cursor tracking measurements;**
- **Waveform storage, record and replay dynamic waveforms;**
- **User selectable fast offset calibration;**
- **Add, Subtract and Multiply Mathematic Functions;**
- **Selectable 20 MHz bandwidth limit;**
- **External trigger;**
- **Waveform average;**
- **Adjustable waveform intensity, more effective waveform view;**
- **User interface in several user-selectable languages;**

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.

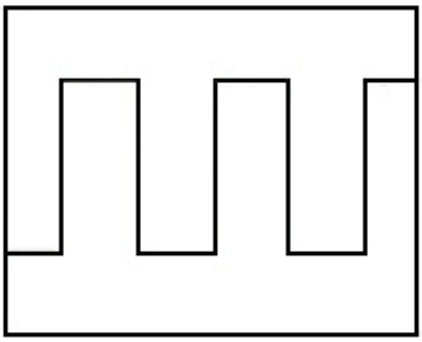
Probe Compensation

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.

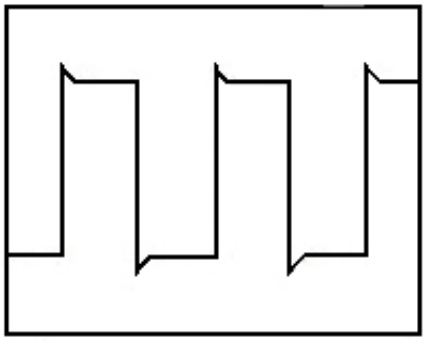
■ From the “Probe” menu, select attenuation to 1:10. Set the switch to “X10” on the probe and connect it to CH1 of the oscilloscope. When using the probe hook-tip, insert the tip onto the probe firmly to ensure a proper connection.

■ Attach the probe tip to the Probe Compensator and the reference lead to the ground connector, select CH1, and then press the “AUTOSET“ button into the menu or the toolbar.

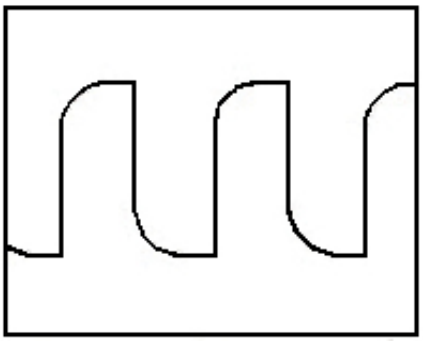
■ Check the shape of the displayed waveform.



Correctly Compensated



Over compensated



Under Compensated

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

■ Repeat if necessary.

WARNNING: 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.

Function Check

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

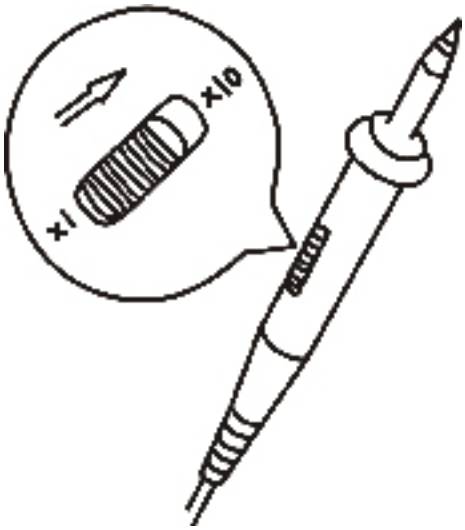
■ **Connect the oscilloscope**
You should connect the A-Type Plug of USB cable to your PC USB port and connect the B-Type Plug of USB cable to oscilloscope USB port.

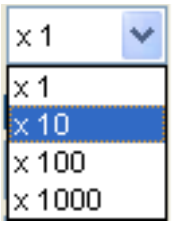
■ **Input a signal to a channel of the oscilloscope**
The oscilloscope is equipped with four channels plus external trigger.

Please input signal in the following steps:

1. Set the attenuation switch on the probe as 10X and connect the probe on the oscil loscope 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 button. A square wave will be displayed within a several seconds. (Approximately 1 kHz, 2V, peak- to- peak).

4. Inspect CH2 ,CH3 and CH4 with the same method. Repeat steps 2 and 3.

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 5v 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 -> Calibration**" option and follow the directions on the screen. The self calibration routine takes about several minutes.

Accessories

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

■ Probe × 4 (1.5m), 1:1, (10:1) Passive Probes

■ A User's Guide

■ An USB Cable

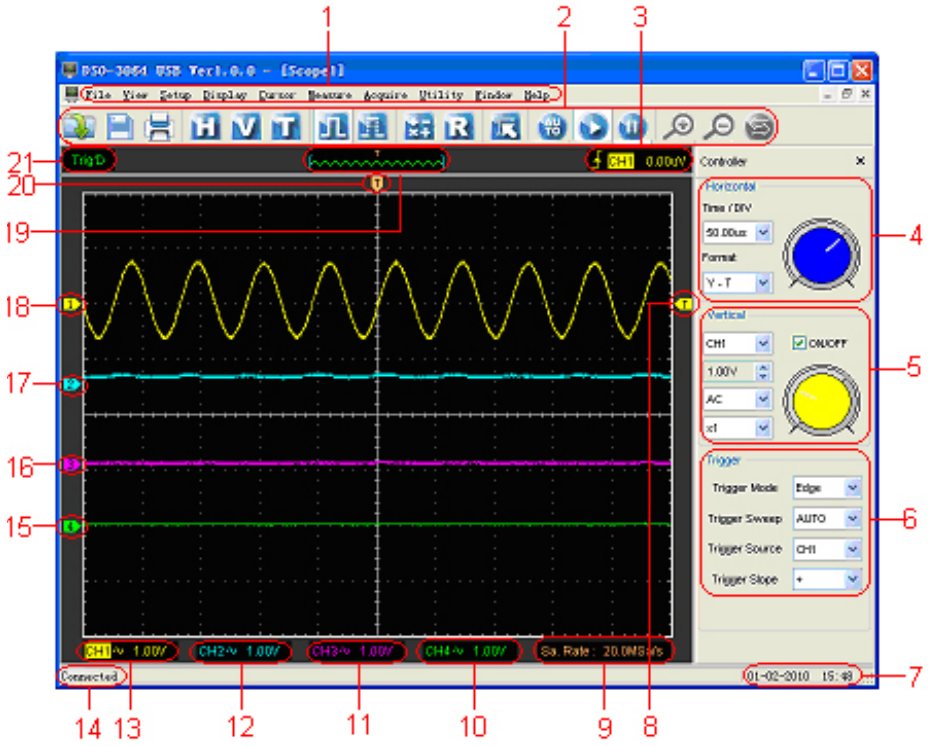
■ A PC software of the oscilloscope

Chapter 2 Operating Basics

- The User's Interface
- The Menu System
- The Vertical System
- The Horizontal System
- The Trigger System
- Input Connectors

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:



In addition to displaying waveforms, the display area is filled with many details about the waveform and the oscilloscope control settings.

1. **The Main Menu**
   All settings can be found in the main menu.

2. **The Toolbar**

3. **It shows the trigger information**
   It shows the edge trigger slope, source and level.

4. **The Horizontal Panel**

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

5. **The Vertical Panel**
The user can turn on/off the CH1/CH2/CH3/CH4. Also the user can change the CH1/CH2/CH3/CH4 volt/div, coupling and probe attenuation.

6. **The Trigger Panel**
In this panel, the user can change the trigger mode, sweep, source and slope.

7. **It shows the system time.**

8. **Marker shows Edge trigger level.**

9. **It shows the main time base setting.**

10. **It shows the CH4 information**
Readouts show the coupling of the channels.
Readouts show the vertical scale factors of the channels.
A “B” icon indicates that the channel is bandwidth limited.

11. **It shows the CH3 information**
Readouts show the coupling of the channels.
Readouts show the vertical scale factors of the channels.
A “B” icon indicates that the channel is bandwidth limited.

12. **It shows the CH2 information**
Readouts show the coupling of the channels.
Readouts show the vertical scale factors of the channels.
A “B” icon indicates that the channel is bandwidth limited.

13. **It shows the CH1 information**
Readouts show the coupling of the channels.
Readouts show the vertical scale factors of the channels.
A “B” icon indicates that the channel is bandwidth limited.

14. **It shows the software status.**

15. **The markers show the reference points of the displayed waveforms. If there is no marker, the channel is not displayed.**

16. **The same as up.**

17. **The same as up.**

18. **The same as up.**

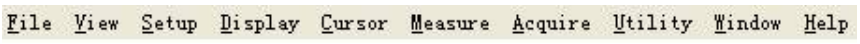
19. **A window that shows the display waveform in buffer position.**

20. **Marker shows horizontal trigger position.**

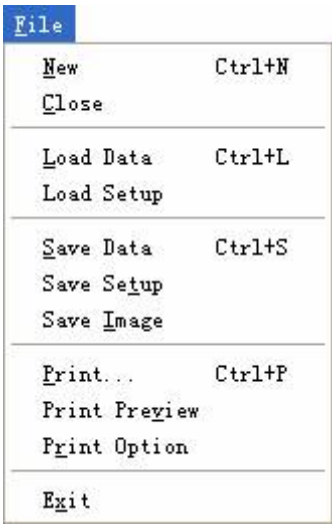
21. **Trigger status indicates the following:**
AUTO: The oscilloscope is in auto mode and is acquiring waveforms in the absence of triggers.
Trig'D: The oscilloscope has seen a trigger and is acquiring the post trigger data.
WAIT: All pretrigger data has been acquired and the oscilloscope is ready to accept a trigger.
STOP: The oscilloscope has stopped acquiring waveform data.
RUN: The oscilloscope is running.
PLAY: The oscilloscope is displaying the record waveforms.

The Menu System

The Main Menu

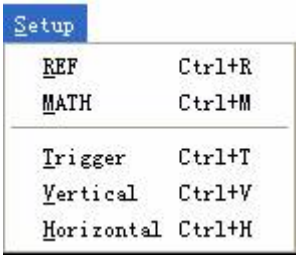


1. **File**: Load or Save data, setup



2. **View**: Change the user interface

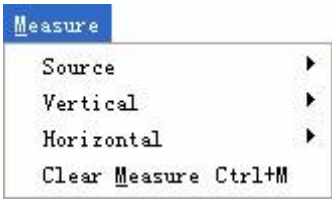
3. **Setup**: Setup setting



4. **Display**: Change wave display type

5. **Cursor**: Set Cursor measure type

6. **Measure**: Set measurement parameters



7.**Acquire**: Run ,Stop or other operation setting

8. **Utility**: Utility setting

9. **Window**: Window setting

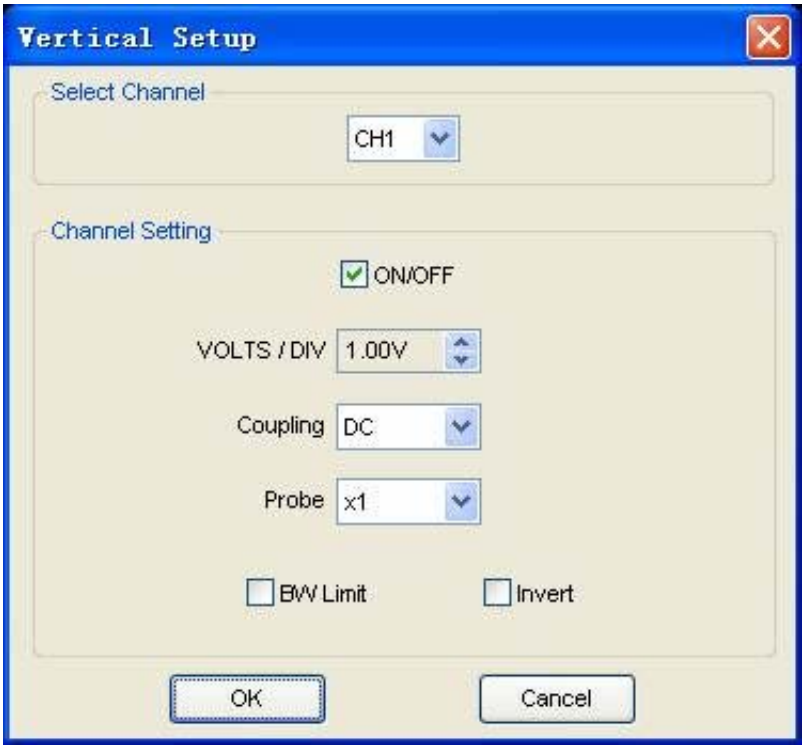
10. **Help**: Turn on help file

Help
Contents F1
About... F9

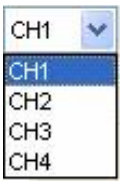
The Vertical System

Click “Setup”->” Vertical”

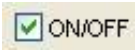
The following figure shows the vertical Setup window. It shows the vertical parameters setting.



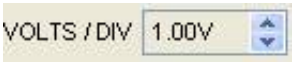
1. Select channel : User can select the channel by clicking the Combo box.



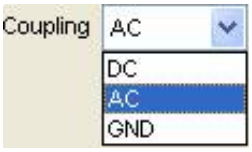
2. ON/OFF: Turn on or off the selected channel.



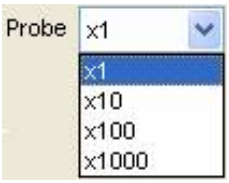
3. VOLTS/DIV: Set the selected channel voltage range.



4. Coupling: Set the selected channel to DC/AC.



5. Probe: Set the Select one according to the probe attenuation factor to ensure correct vertical scale reading



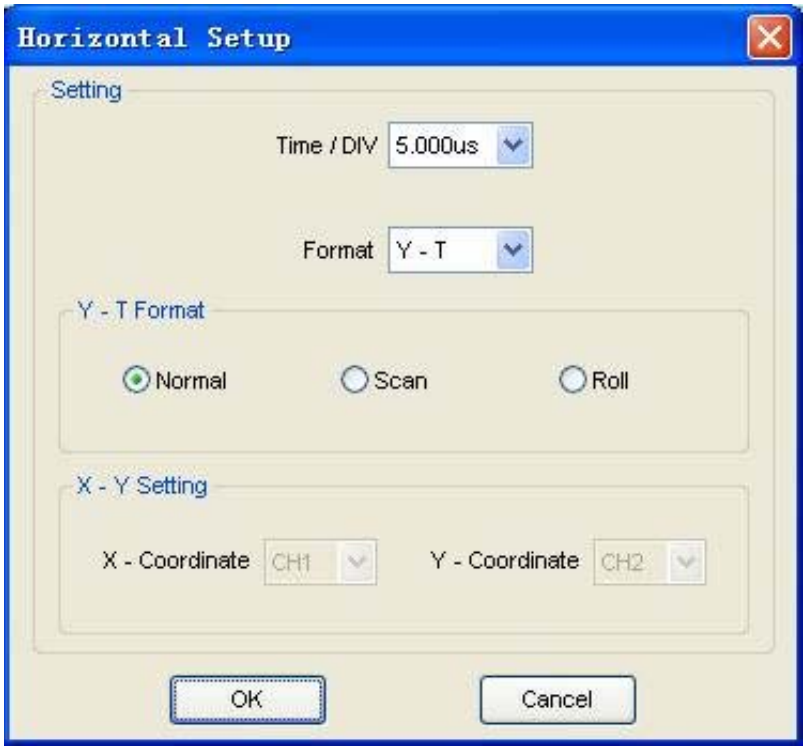
6. BW Limit: Reject the frequency component higher than 20MHz.

7. Invert: Invert the selected wave.

The Horizontal System

Click ”Setup”->“Horizontal”

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



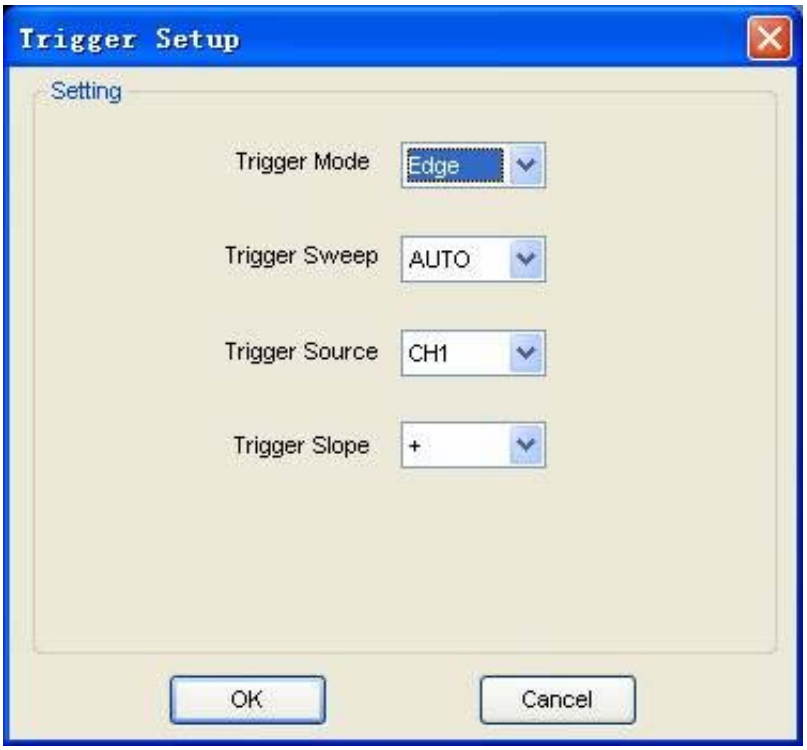
1. **Time/DIV**: leads the setting of the time base parameters

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

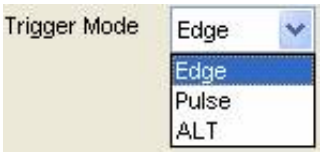
The Trigger System

Click “**Setup**”-> “**Trigger**”

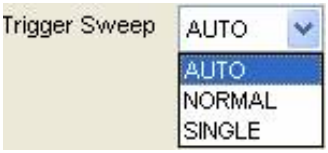
The following figure shows the trigger system control.



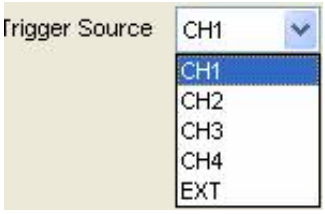
1. **Trigger Mode**: Sets the trigger mode



2. **Trigger Sweep**: Selects the trigger sweep mode to AUTO, NORMAL or SINGLE



3. **Tirgger Source**: Selects the trigger source to CH1, CH2, ALT, EXT or EXT/10



4. **Trigger Slope**: Selects the edge trigger slope to Positive or Negative slope

Input Connector

CH1/CH2/CH3/CH4: Input connectors for waveform display.

EXT.: Input connector for an external trigger source. Use the Trigger Menu to select the Ext. or Ext./10 trigger source.

Other Connector:

GND.: a ground terminal
USB PORT: Connect the B-Type Plug of USB cable to this port.
CAL.: Probe compensation output.

Chapter 3 Understanding Oscilloscope Functions

- Set Oscilloscope
- Set Vertical System
- Set Horizontal System
- Set Trigger System
- Save/Load
- Utility Function
- Measure Signal
- Zoom In/Out Waveforms
- Acquire Signal
- Print

Setup the Oscilloscope

Using "AUTOSET" to display a signal automatically
Auto setup functions one time each time you push the "**AUTOSET**" 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 the next time you run the software. You can use the "**Save Setup**" menu to permanently save up to several different setups.

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 "**Load Setup**" menu to permanently recall a setup.

Factory Setup

The oscilloscope software is set up for normal operation when it is shipped from the factory. This is the factory setup. To recall this setup, push the "**Factory Setup**" menu.

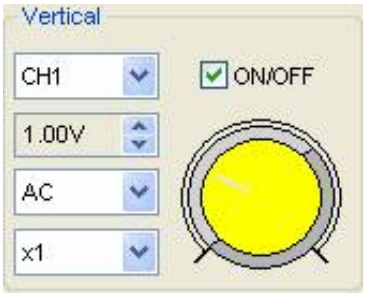
Set Vertical System

Set Channel

Click "**Vertical**" in "**Setup**" Menu.

The Channel Selection

The Channel Control Panel in sidebar



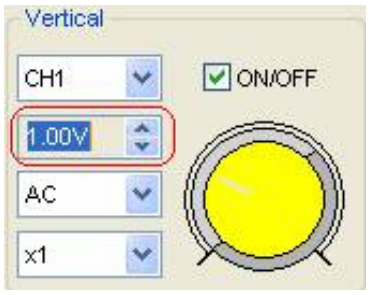
The Vertical function:

Turn ON/Off: Turn on/off the channel
Volt/DIV: Select the channel voltage/div
Coupling: Select the channel coupling
Probe: Select the channel probe attenuation
Filter: Select software filter
Reset: Set the channel vertical position to zero
Invert: Turn on/off the invert function.
BandWidth Limit: Limit the channel bandwidth to 20MHz to reduce noise.

Change Volt/DIV

You can click "Volt/Div" in "Vertical Setup" window to select the voltage

You can also change the selected channel voltage in sidebar

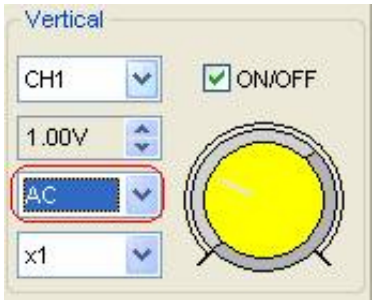


You can left click and drag the mouse on the knob to change the voltage.

Set Channel Coupling

Click "**Coupling**" in "**Vertical Setup**" window

In the sidebar, you can change the channel coupling too.



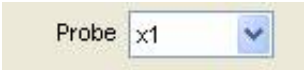
You can set the coupling to **DC**, **AC** or **GND**. If you set the coupling to **DC**, it blocks the **AC** component of the input signal.

Probe Attenuation Setting

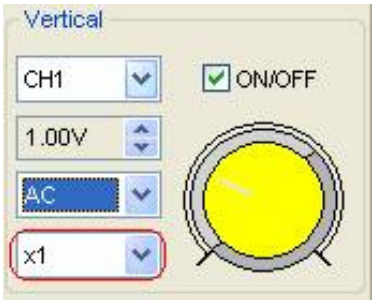
Select the attenuation factor for the probe. To check the probe attenuation setting, toggle the probe menu to match the attenuation factor of the probe.

This setting remains in effect before you changed again.

Click "**Probe**" in Vertical Setup window to select the probe attenuation



The probe setting window in the sidebar



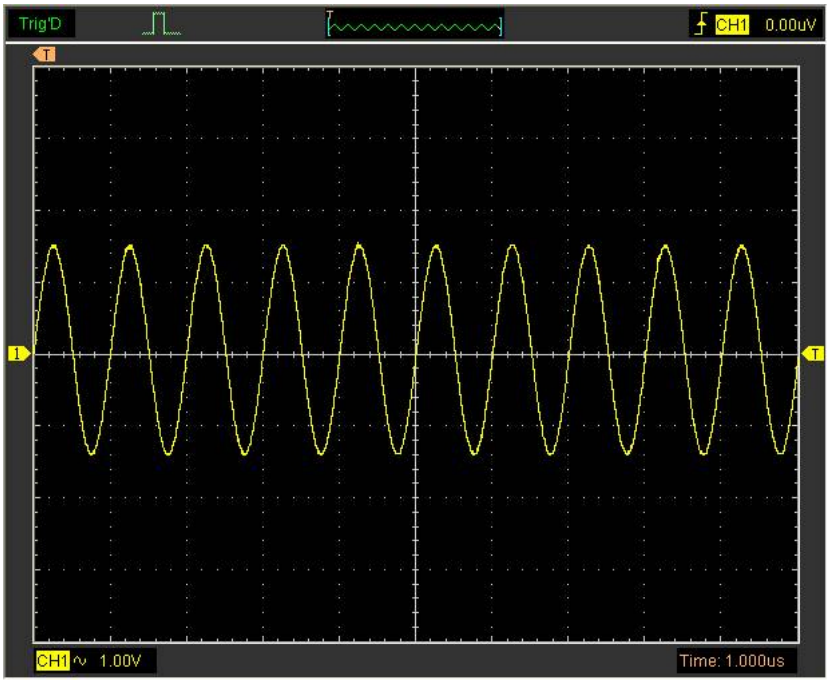
Note: The attenuation factor changes the vertical scale of the oscilloscope so that the measurement results reflect the actual voltage levels at the probe tip.

Invert

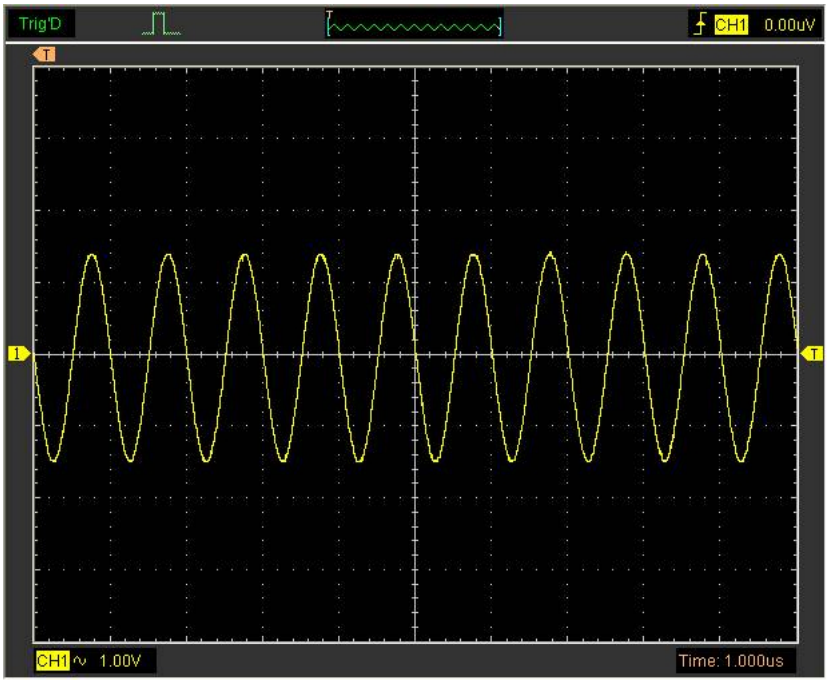
The invert function turns the displayed waveform 180 degrees, with respect to the ground level. When the oscilloscope is triggered on the inverted signal, the trigger is also inverted.

Click "**Invert**" in Vertical window

The following picture shows the waveform before inversion:



The following picture shows the waveform of inversion:



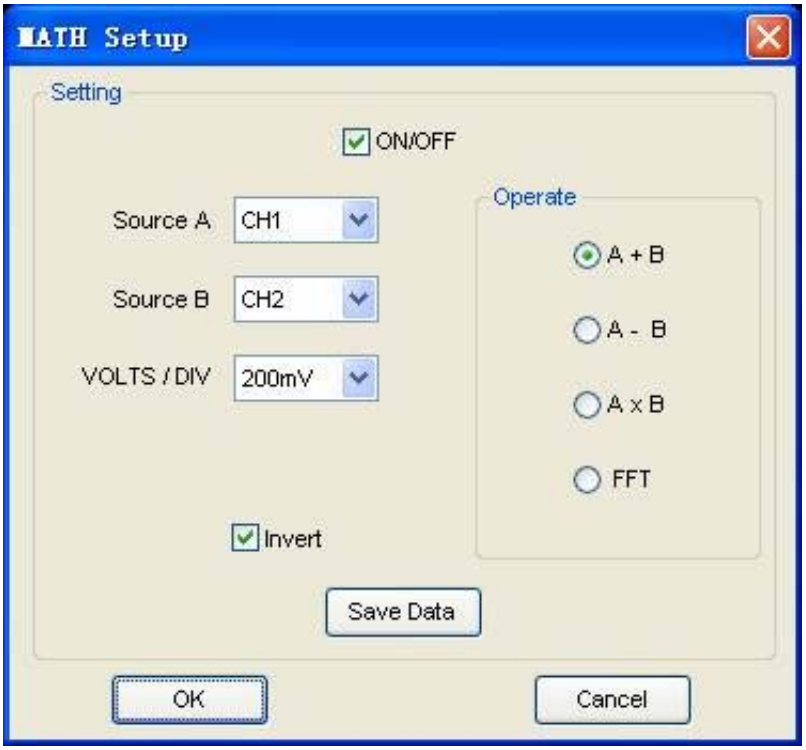
Set the Channel Bandwidth Limit

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.

When the "**BW Limit**" was turned on, a "B" sign will be displayed at the bottom of display screen.

Set Math

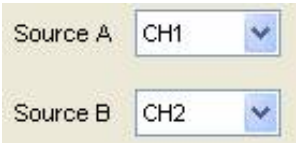
Click "**MATH**" in **Channel** menu to set **MATH** channel.
The **MATH Setup** window



ON/OFF: Turn On/Off the MATH Channel.
Source A/B: Set the sources of the math channel.
Operate: Set operates type of the math channel.
Volt/DIV: Set the resolution of the math channel.
Probe: Set the math channel probe attenuation.
Invert: Turn on/off the invert function
The mathematic functions include addition, subtract, multiply and FFT for CH1, H2,CH3 and CH4.

Source A/B

Source A and Source B Menu



Operate

Four Types:

A + B Add source A and source B

A - B Subtract source B from source A

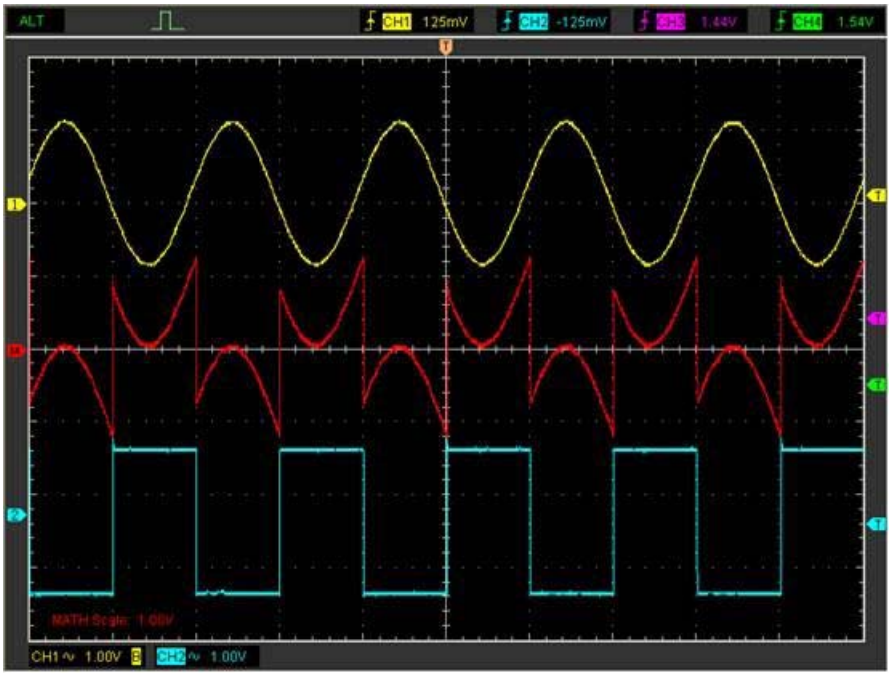
A × B Multiply source A by source B

FFT Convert a time-domain signal into its frequency components (spectrum).

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

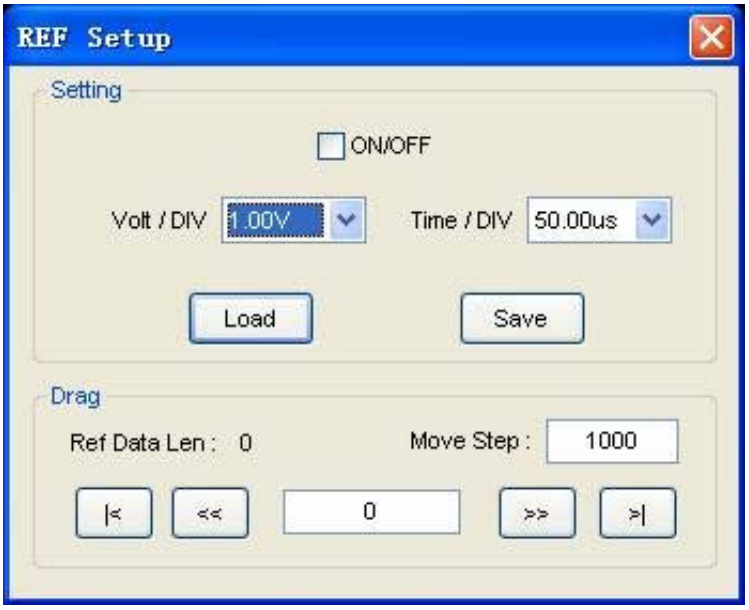
Select the operate type in the **Operate** menu. Select source A and B. Then adjust the vertical scale and offset to view the math channel clearly. The mathematic result can be measured by the measure and the cursor.

The Math Function Display



Set Reference

Click “**REF**” in “**Setup**” menu to set REF channel.



The Reference Channel Function:

On/Off: Turn on/off the reference channel.
Volt/DIV: Channel the resolution of the reference channel.
Load: Load the reference waveform from the “.rfc” file from your computer.
Save: Save the current reference waveform to your computer as “.rfc” format.
Save Reference: 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.

Load

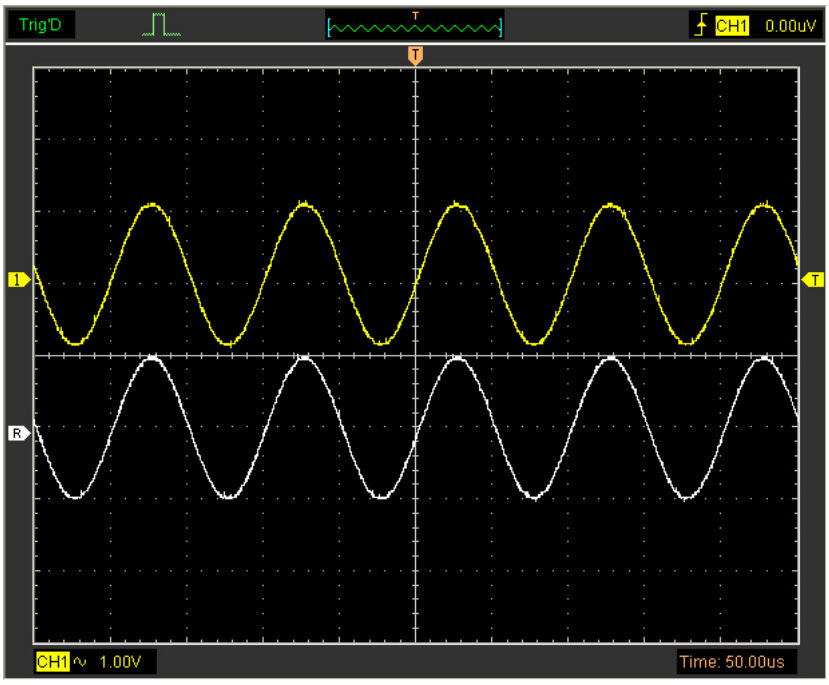
Click “**Load**” to load the *.rfc file that was selected. The load file window will appear.

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.

Setup Horizontal System

Change Time/Div

The "**Time/Div**"

Selects the horizontal **Time/DIV** (scale factor) for the main or the window time base

The Horizontal Panel

Click the blue knob can change **Time/Div**.

If the waveform acquisition is stopped, **Time/Div** control expands or compresses the waveform.

Change Format

Click "**Time/Div**" you can Set the Time base in Horizontal Setup window

In the "**Format**" item, set the waveform display format (**Y-T**, **X-Y** 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

Note: If the time/div bigger than 1s, the format will change to Roll mode automatically.

Change Horizontal Position

Double click the channel button to set the trigger point to the horizontal center of the screen.

Horizontal position changes the displayed waveform position, relative to the trigger point.

The user can drag on screen to change the horizontal position.

Set Trigger System

Set Edge Trigger

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 oscilloscope finds the trigger point on the rising or the falling edge of a signal. Select **Edge** trigger mode to trigger on **Rising** edge or **Falling** edge.

Mode: Select the trigger mode.

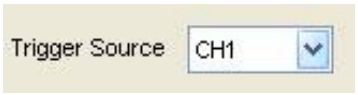
Sweep: Set the sweep mode to **Auto**, **Normal** or **Single**.

Auto: Acquire waveform even no trigger occurred

Normal: Acquire waveform when trigger occurred.

Single: Acquire waveform when trigger occurred then stop

Source: You can use the trigger source options to select the signal that the oscilloscope uses as a trigger. The source can be any signal connected to a channel BNC, or to the EXT. BNC.



CH1: Select CH1 as trigger signal

CH2: Select CH2 as trigger signal

CH3: Select CH3 as trigger signal

CH4: Select CH4 as trigger signal

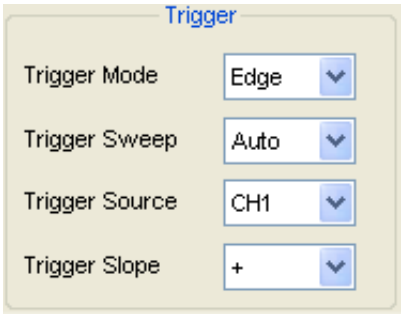
EXT: Select EXT as trigger signal

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

Rising: Trigger on rising edge

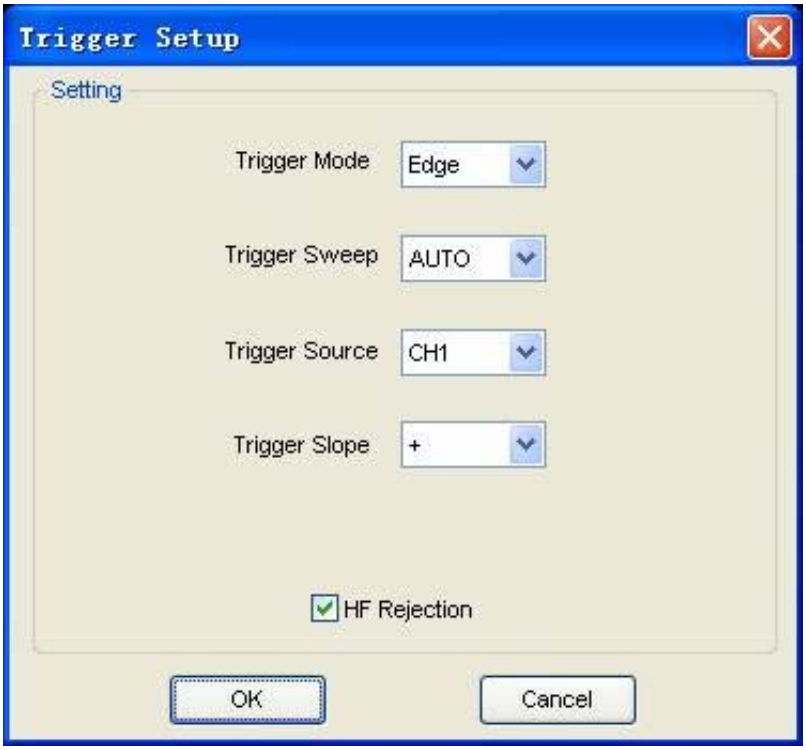
Falling: Trigger on falling edge

The user can also change the trigger setting on trigger panel in sidebar.



High Frequency Rejection

Select "**HF Rejection**" in "**Trigger Setup**" window

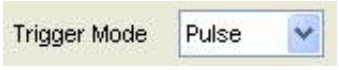


The user can turn on “HF Rejection” to eliminate trigger higher-frequency (20M above).

Set Pulse Trigger

Pulse trigger occurs according to the width of pulse. The abnormal signals can be detected through setting up the pulse width condition.

Mode: Select the trigger mode to **Pulse**.



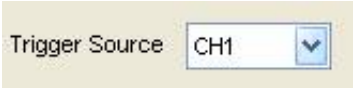
Sweep: Set the sweep mode to **AUTO**, **NORMAL** or **SINGLE**.

AUTO: Acquire waveform even no trigger occurred

NORMAL: Acquire waveform when trigger occurred.

SINGLE: Acquire waveform when trigger occurred then stop

Source: You can use the trigger source options to select the signal that the oscilloscope uses as a trigger. The source can be any signal connected to a channel BNC, or to the EXT. BNC.



CH1: Select CH1 as trigger signal

CH2: Select CH2 as trigger signal

CH3: Select CH3 as trigger signal

CH4: Select CH4 as trigger signal

EXT: Select EXT as trigger signal

PW Condition: Set the PW Condition to the following condition.

+More: +Pulse width more than selecting pulse condition.

+Less: +Pulse width less than selecting pulse condition.

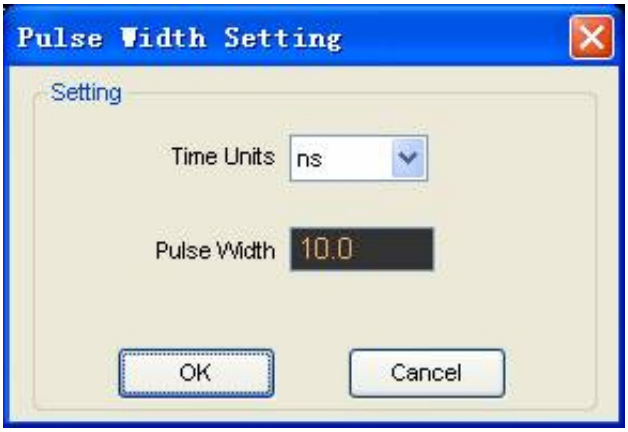
+Equal: +Pulse width equal to selecting pulse condition.

-More: -Pulse width more than selecting pulse condition.

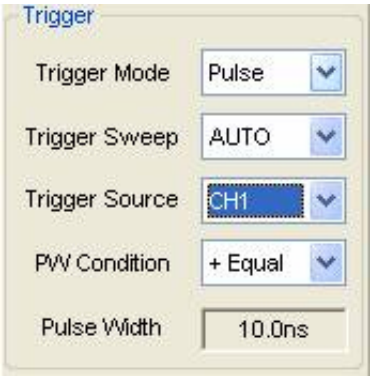
-Less: -Pulse width less than selecting pulse condition.

-Equal: -Pulse width equal to selecting pulse condition.

Pulse Width: The Pulse Width adjust range is 10ns~10s. When the condition is met ,it will trigger and acquire the waveform.



The user can also change the trigger setting on trigger panel in sidebar.



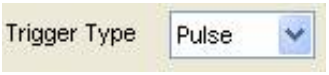
When alternative trigger is on, the trigger sources come from two vertical channels. This mode can be used to observe two non-related signals. You can choose two different trigger modes for the four vertical channels.

Set ALT System

Mode: Select the trigger mode.

Trigger Channel: Set the Trigger Channel to **CH1**,**CH2**,**CH3** or **CH4**.

Trigger Type: Set the Trigger Type to **Edge** or **Pulse**.



PW Condition: Set the PW Condition to the following condition.

+More: +Pulse width more than selecting pulse condition.

+Less: +Pulse width less than selecting pulse condition.

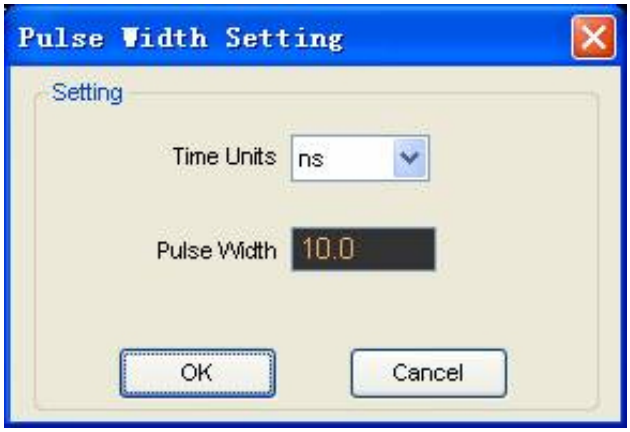
+Equal: +Pulse width equal to selecting pulse condition.

-More: -Pulse width more than selecting pulse condition.

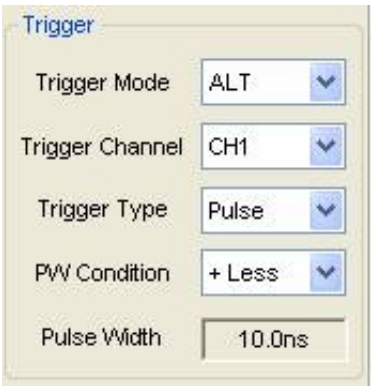
-Less: -Pulse width less than selecting pulse condition.

-Equal: -Pulse width equal to selecting pulse condition.

Pulse Width: The Pulse Width adjust range is 10ns~10s. When the condition is met, it will trigger and acquire the waveform.



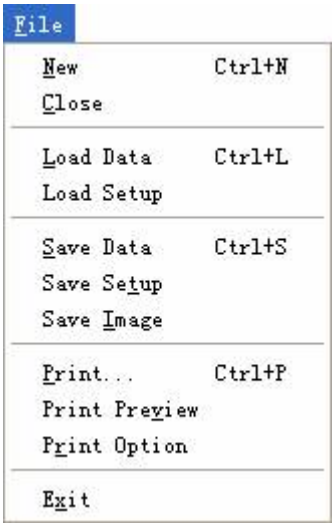
The user can also change the trigger setting on trigger panel in sidebar.



Save/Load

Save

Click “**File**” in main menu to save waveform, setups and screen



1. **Save Data**

Save waveform data as a type file.User can save CH1,CH2,CH3,CH4 or all channels data at one time .

2. **Save Setup**

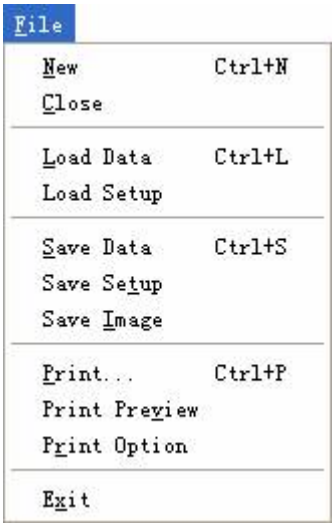
Save the current oscilloscope setup to file

3. **Save Image**

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

Load

Click "**File**" in main menu to recall saved waveform, setup



1. **Load Data**

Load the waveform that had saved as a type file

2. **Load Setup**

Load the instrument that had saved

Utility/Function

Record and Play Back

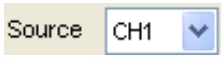
Click “**Record**” in “**Utility**” menu.

The **Record** window will display. The following picture shows the **Record** Interface.

This function can record input waveform form CH1,CH2,CH3, or CH4. The maximum record length is **1000** frames.

Record Setup window

Source:



Select record source channel. (CH1, CH2, CH3 or CH4)

End Frame:

Set the number of record times. The max frames are 1000.

Record:

Record counter, it shows the record frames.

"**Start**" button:

Start to record frames. After you start to record waveforms, this button changes to "**Stop**" button. It stops recording waveforms.

Play back setup window

Start Frame:

Set the start frame of play back.

End Frame:

Set the end frame of play back.

Current Frame:

It shows the current frame of play back. You can also change this number to watch the waveform one by one.

"**Play**" button:

Click this button to start playing back waveform. It can stop playing back if you started playing back.

"**Load**" button:

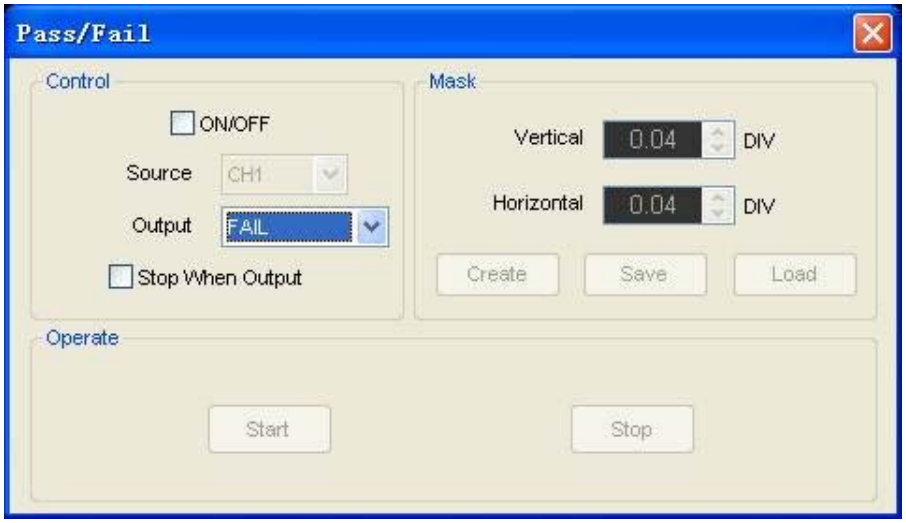
Click this button to load a record setup.

Note: When it plays back waveform, the other channel will be turned off.

Pass/Fail

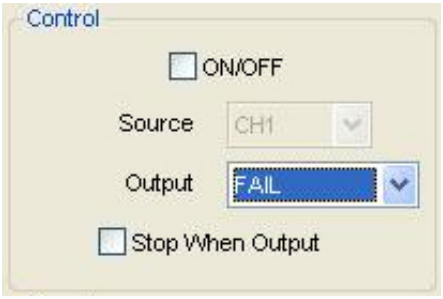
Click "**Pass/Fail**" in "**Utility**" menu

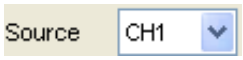
The **Pass/Fail** window appears:



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



Source: 

Select the **Pass/Fail** channel

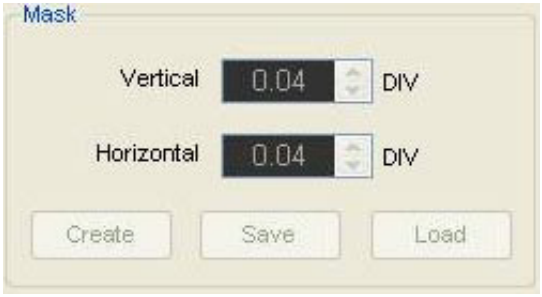
Output:

Select the **Pass/Fail** output condition

Stop When Output:

If it was checked, the Pass/Fail will stop when output.

Mask Setting

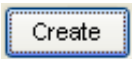


Vertical:

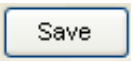
Set the vertical limit range

Horizontal:

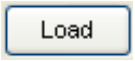
Set the horizontal limit range

“Create” button: 

Click this button to create Pass/Fail area according to the mask

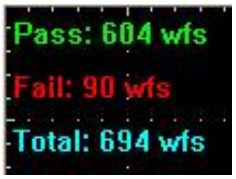
"Save" button: 

Click this button to save the setups to file.

"Load" button: 

Click this button to load the saved setups file.

Information Display

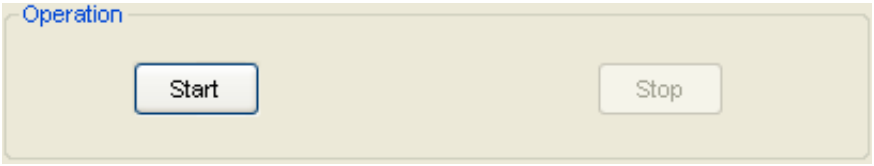


Fail:
It shows the fail waveform number

Pass:
It shows the pass waveform number

Total:
It shows the I total **Pass/Fail** waveform number

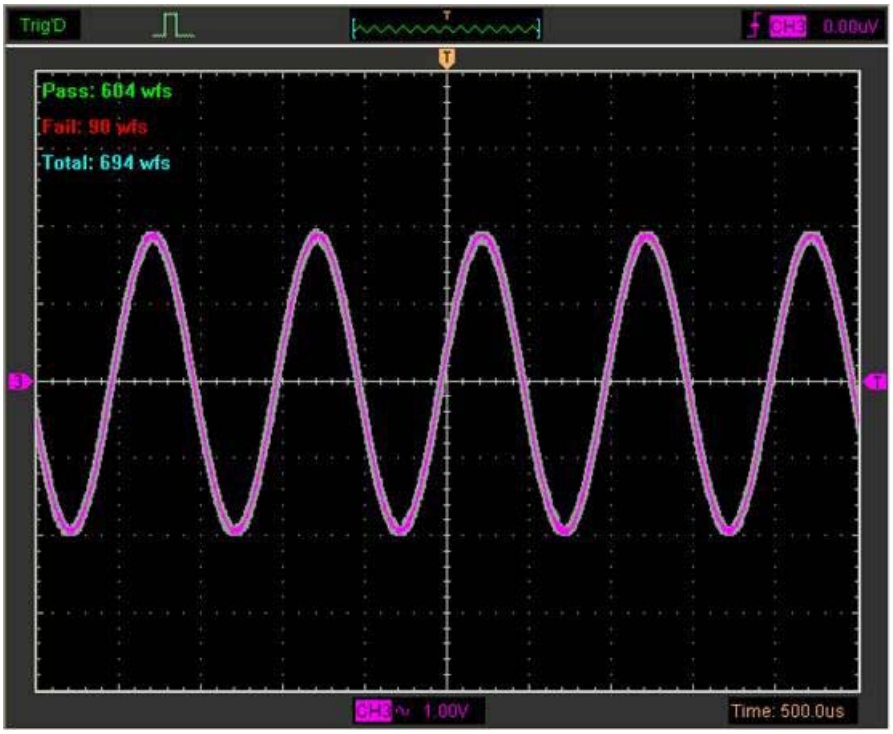
Operation



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.

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 anytime by user.

The Factory Setup function does not reset the following settings:

- Language option
- Date and time

Language

Click "**Language**" in "**Utility**" menu

There are four languages in "**Language**" menu. The default language is English.

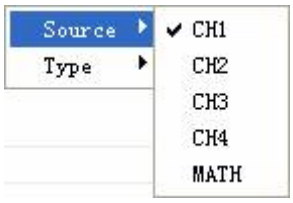
Measure Signal

Cursor Menu

Click "**Cursor**" in main menu

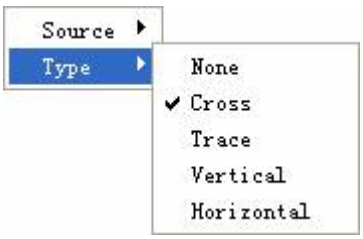
This method allows you to take measurements by moving the cursors

1. **Source**



The user can set the source to **CH1**, **CH2**, **CH3,CH4** 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**

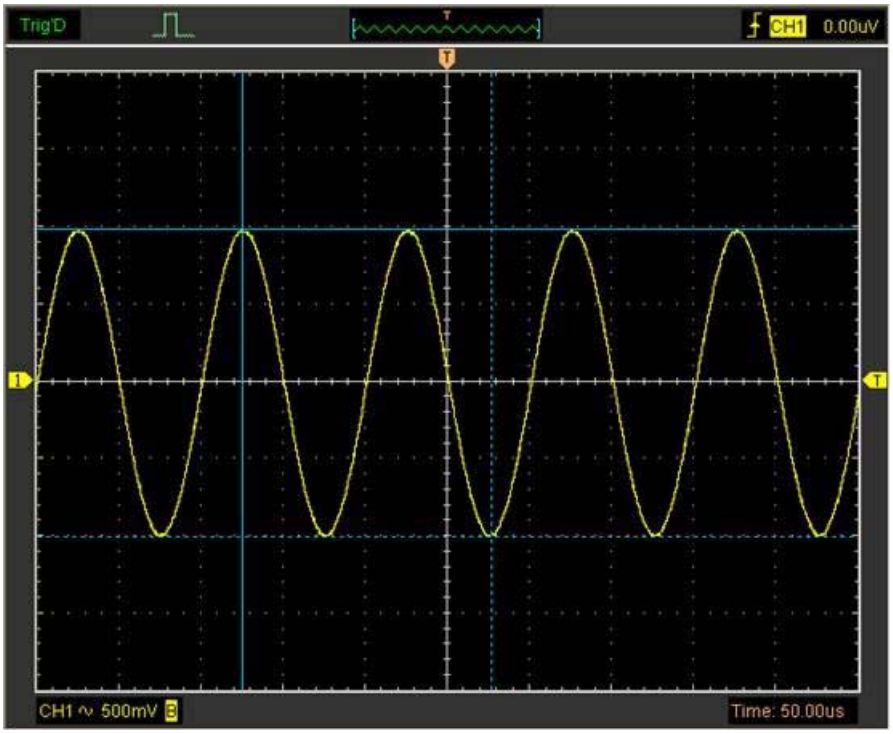


There are four types of cursors: **Cross**, **Trace**, **Vertical** and **Horizontal**

1) **Cross**

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

horizontal parameters.
The **Cross** cursor display window

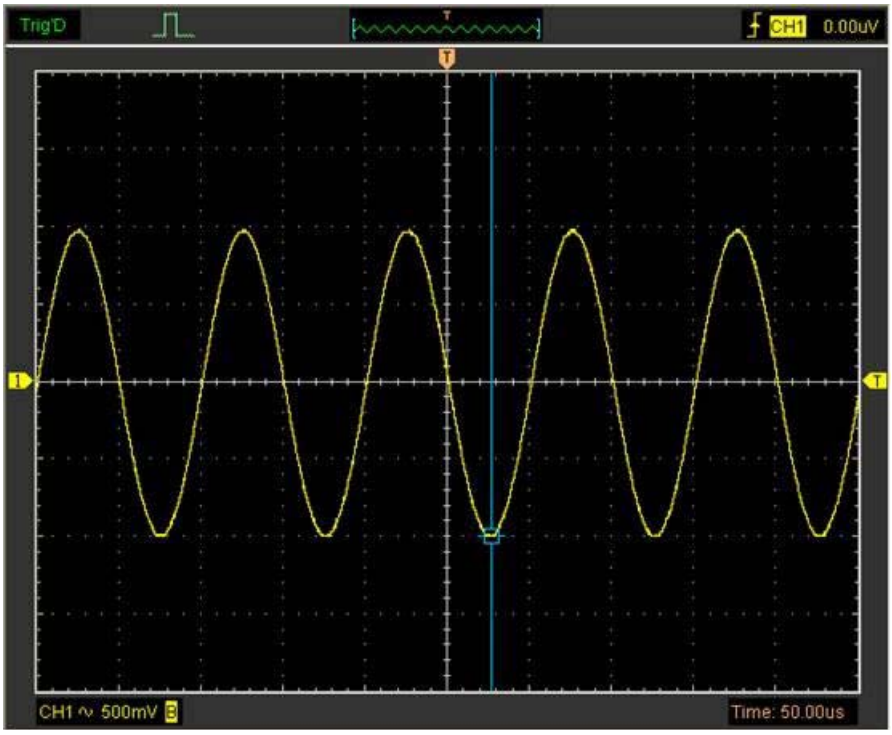


The **Cross** measure result displays on status bar

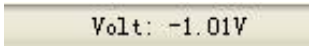
2) **Trace**

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

The **Trace** cursor display window

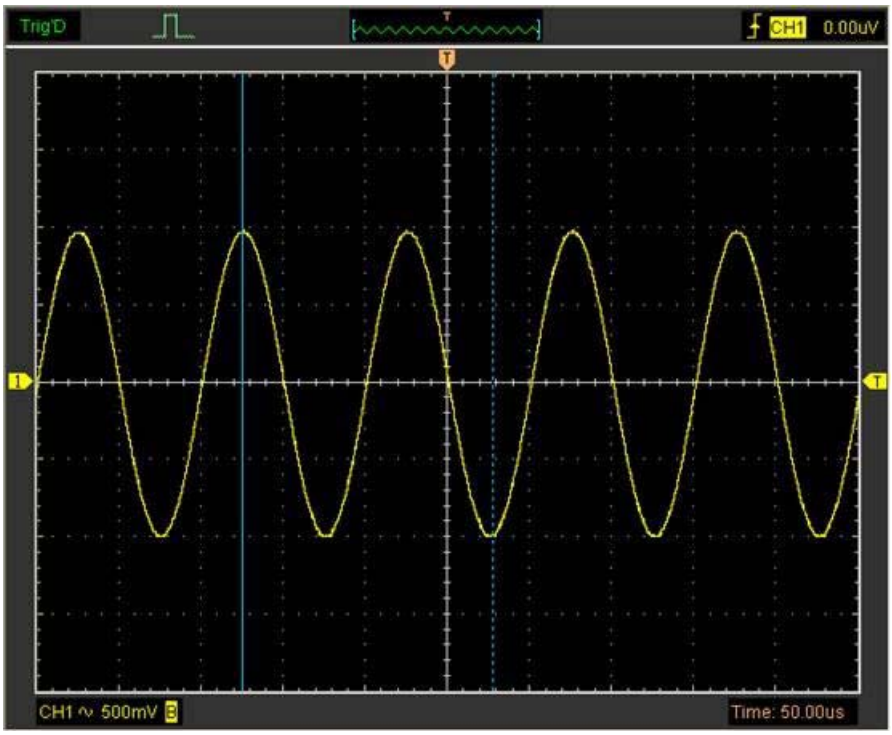


The **Trace** cursor measure result display on status bar



3) **Vertical**

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

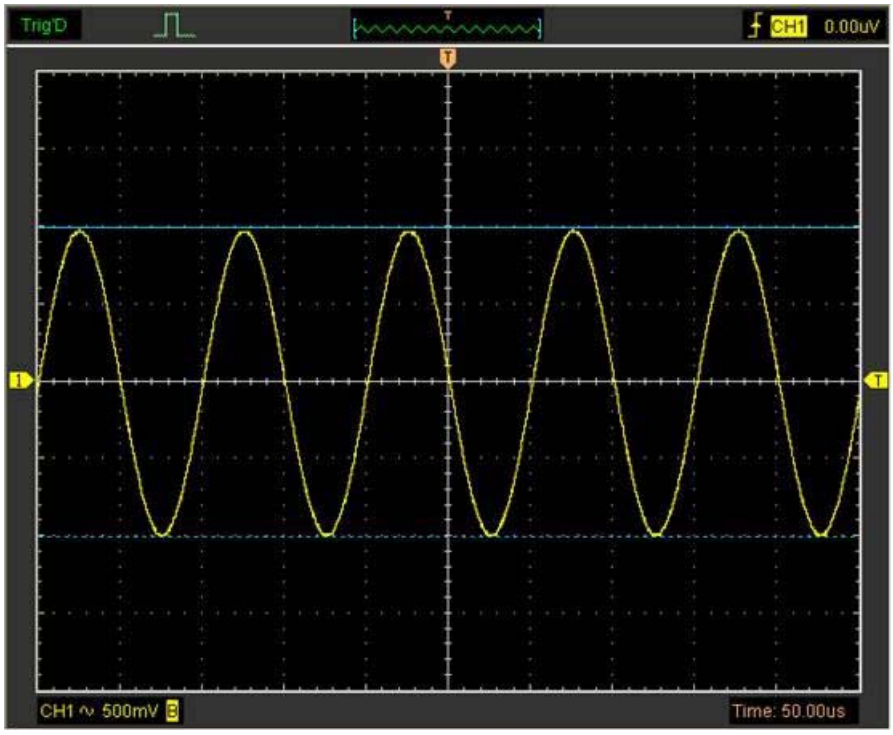


The **Vertical** cursor measure result display on status bar

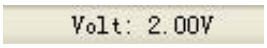
4) **Horizontal**

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

The **Horizontal** cursor display window



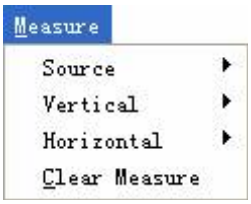
The **Horizontal** cursor measure result display on status bar



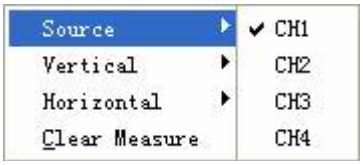
Measure Menu

Click "**Measure**" in main menu

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

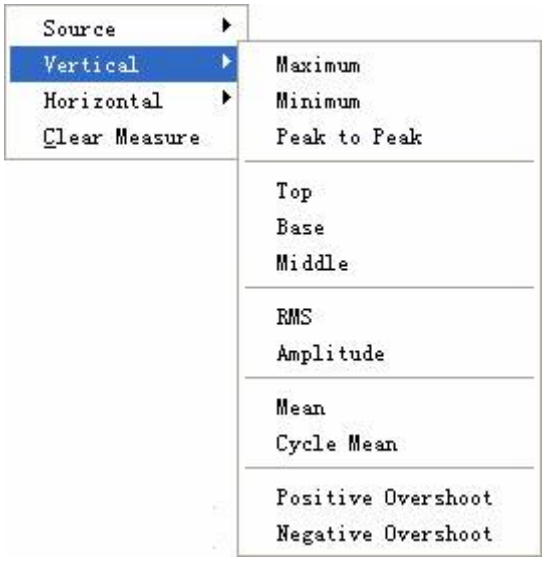


1. **Source**



The user can use the "**Source**" menu to select a measure source.

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

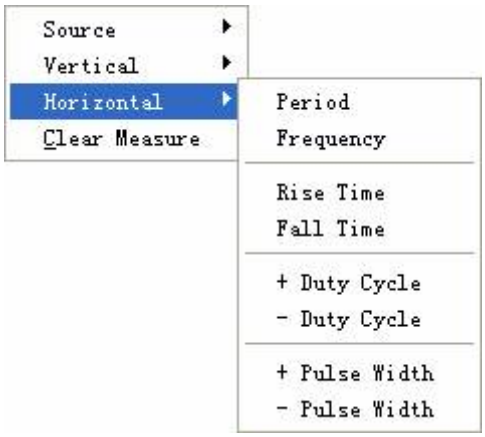
Mean: The arithmetic mean over the entire waveform

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

Preshoot: Positive Overshoot = (Max - Top)/Amp x 100 %, Measured over the entire waveform

Overshoot: Negative Overshoot = (Base - Min)/Amp x 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 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 = (Positive Pulse Width)/Period x 100%, Measured of the first cycle in waveform

-Duty Cycle: Negative Duty Cycle = (Negative Pulse Width)/Period x 100%, Measured of the first cycle in waveform

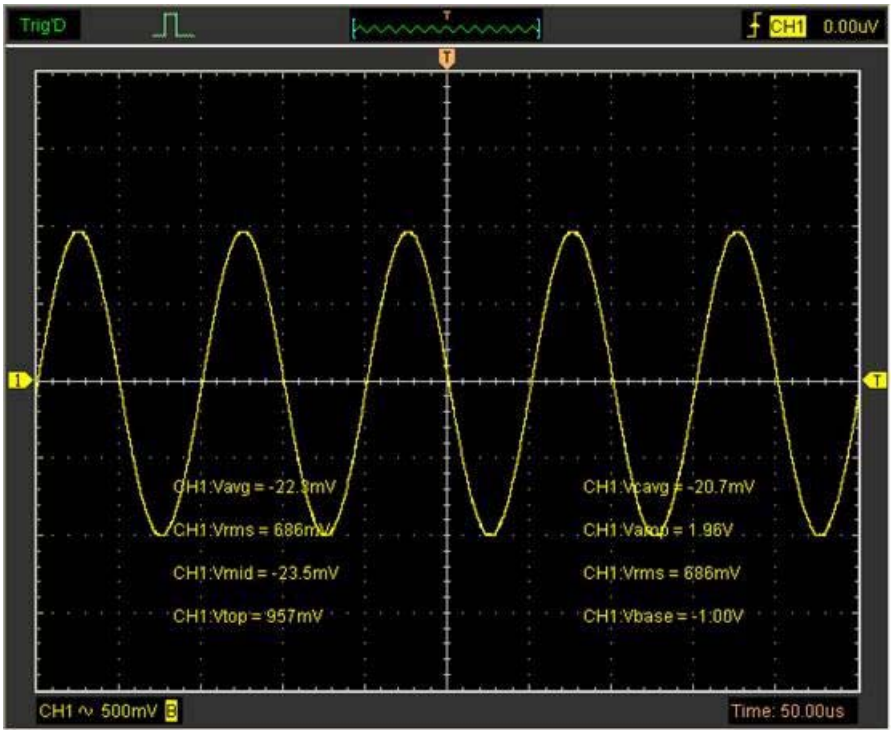
+Pulse Width: Measured of the first positive pulse in the waveform. The time between the 50% amplitude points

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

4. **Clear Measure**

Clear all measure items on display screen.

The **Measure** Display Window



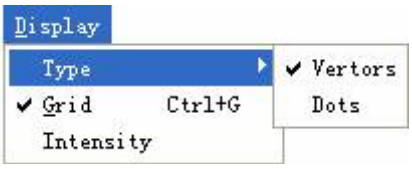
Note: The results of the automatic measurements will be displayed on the bottom of the screen. Maximum 8 results could be displayed at the same time. When there is no room, the next new measurement result will make the previous results moving left, out of screen.

The Display System

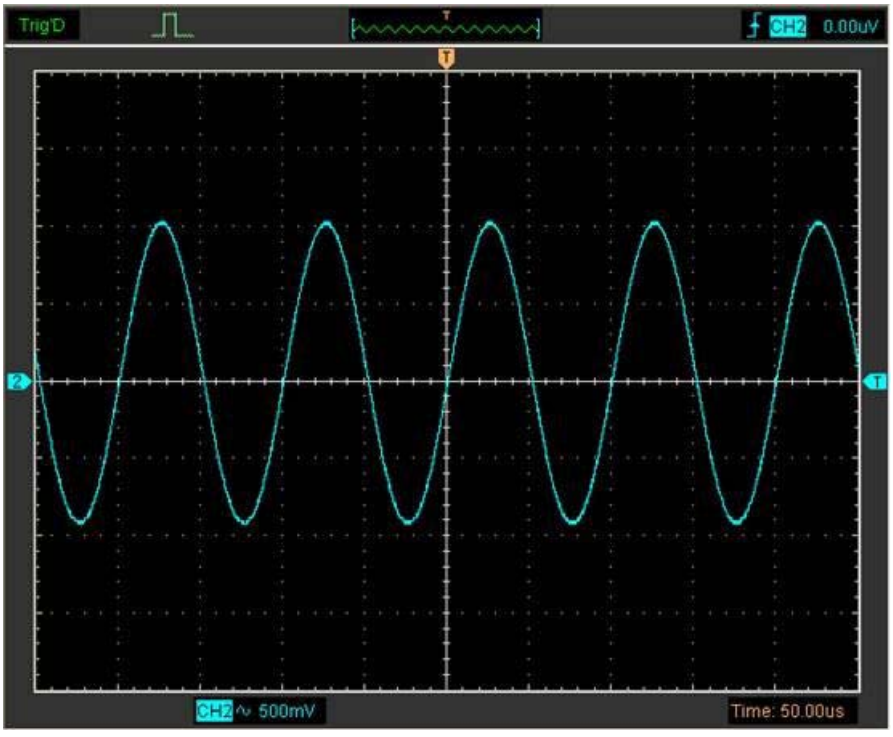
Display Type

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

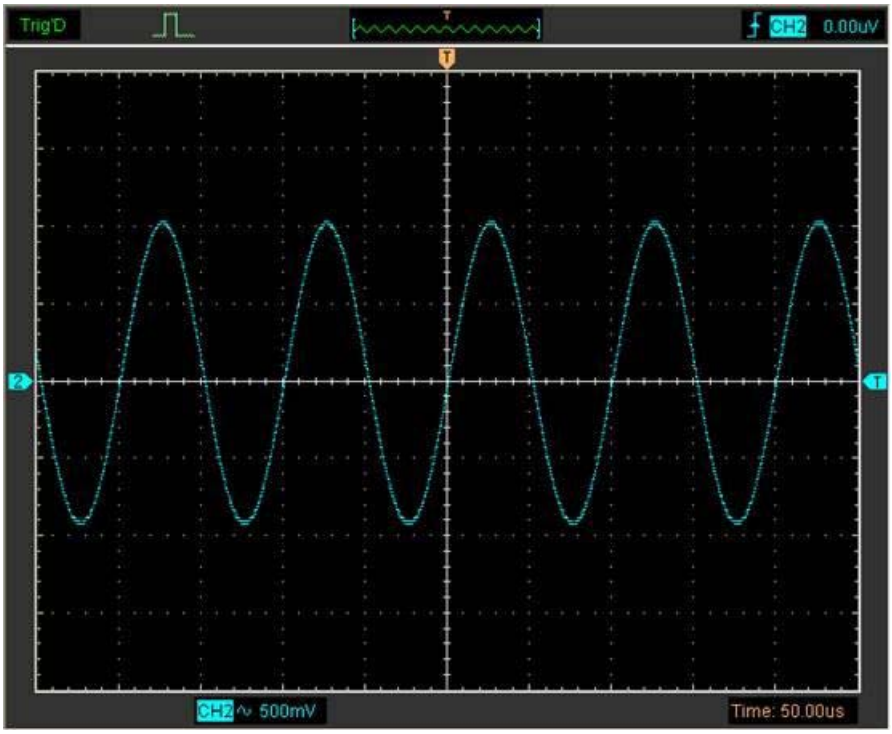
The following figure shows the type parameters setting.



If the **Vectors** type mode is selected, the waveform will be displayed as following figure.



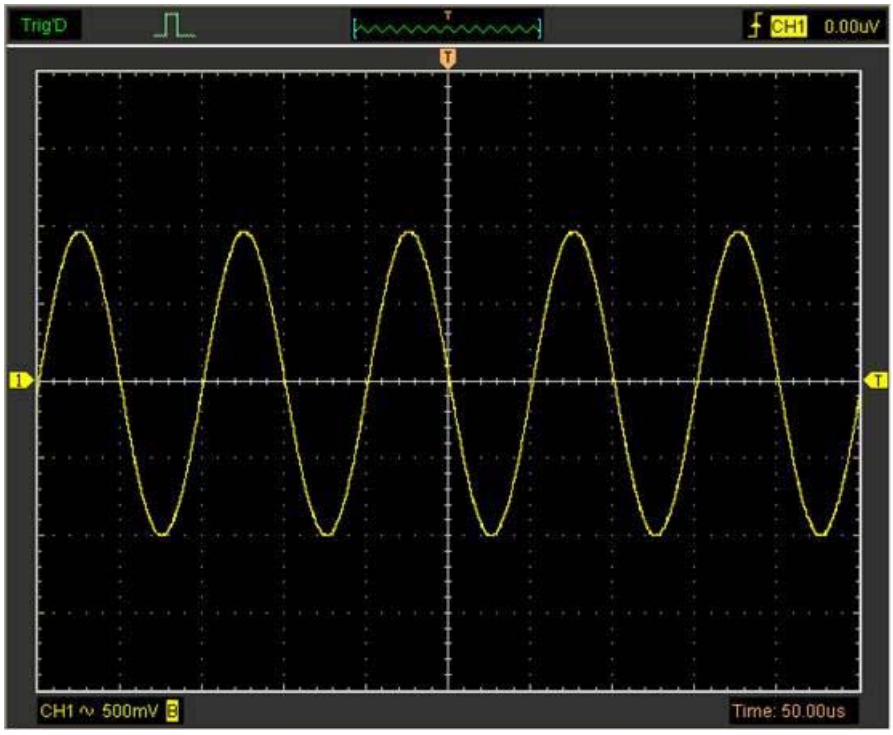
If the **Dots** type mode is selected, the waveform will be displayed as following figure.



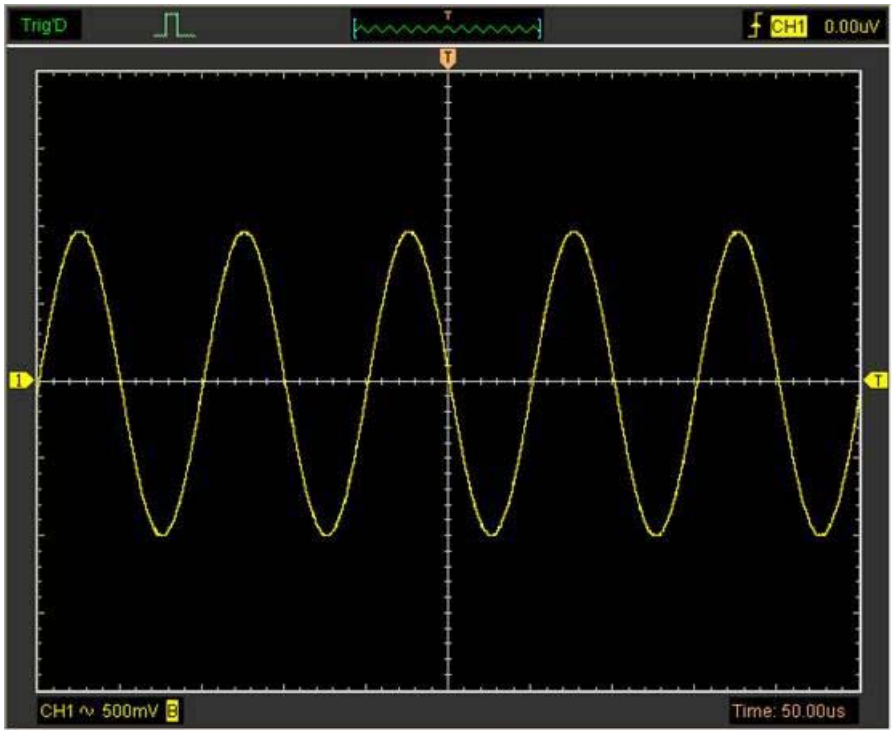
Display Grid

Click "**Display**" in main menu

The grid shows:



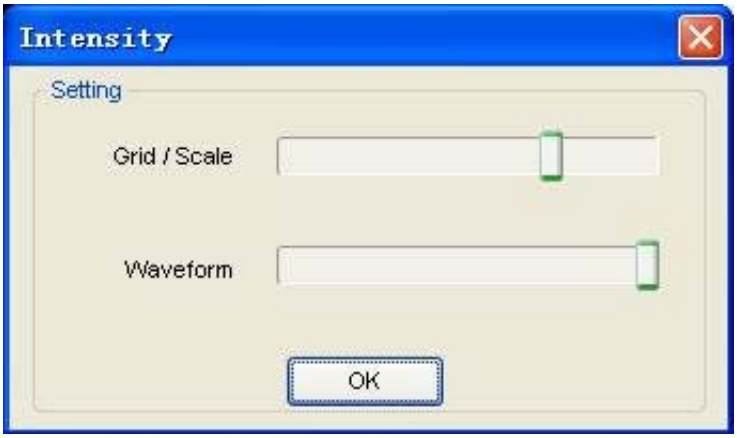
The grid not shows:



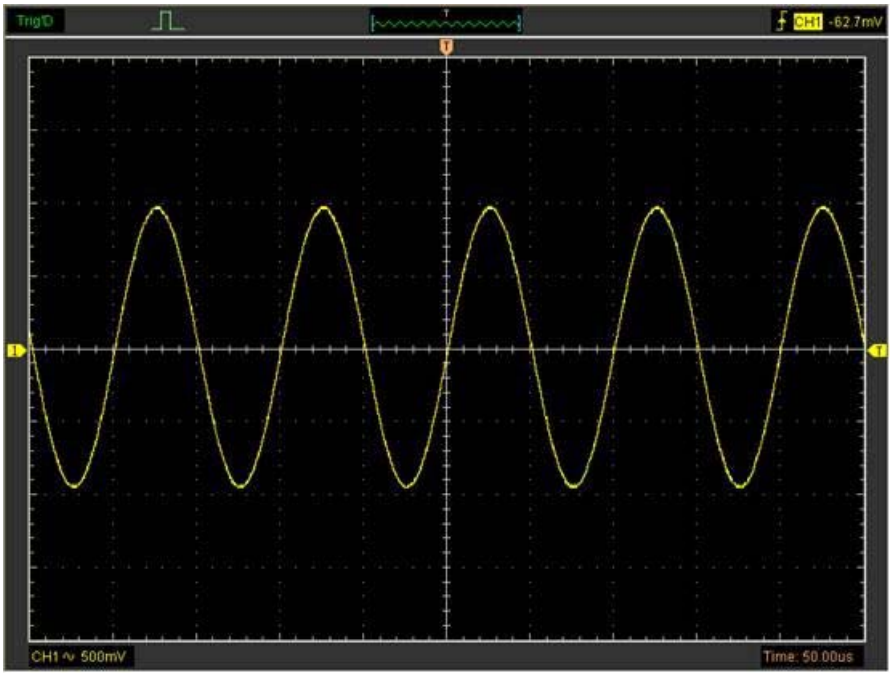
Intensity and Persistence

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.



Zoom In/Out and Drag Waveforms

The software will stop updating waveform after the user clicked "**Stop**" button,
The user can change the waveform display by adjusting the scale and position. When you change the scale, the waveform display will increase or decrease in size. When you change the position, the waveform will move up, down, right, or left.

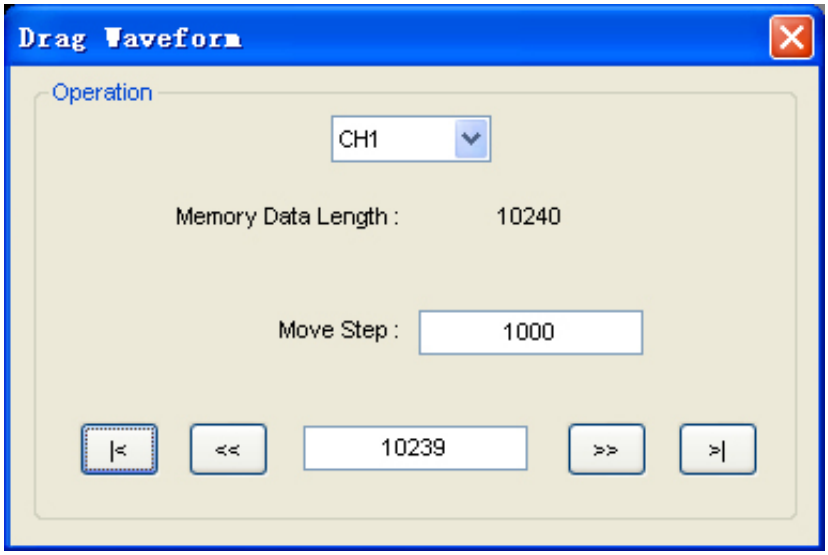
The channel reference indicator identifies each waveform on the display. The indicator points to the reference level of the waveform record.

Zoom In/Out

The user can click "**Zoom In/Out**" in "**Acquire**" menu, then left or right click the mouse button on display screen to **zoom in/out** the waveform. Also the user can change **Time/Div** in **Horizontal** menu or in **Horizontal** panel to zoom in/out the waveform.

Drag

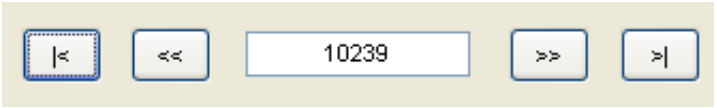
The user can modify the waveform position after clicked "**Drag**" in "**Acquire**" menu following the following steps.



1. Select Channel:

2. Set the Move Step:

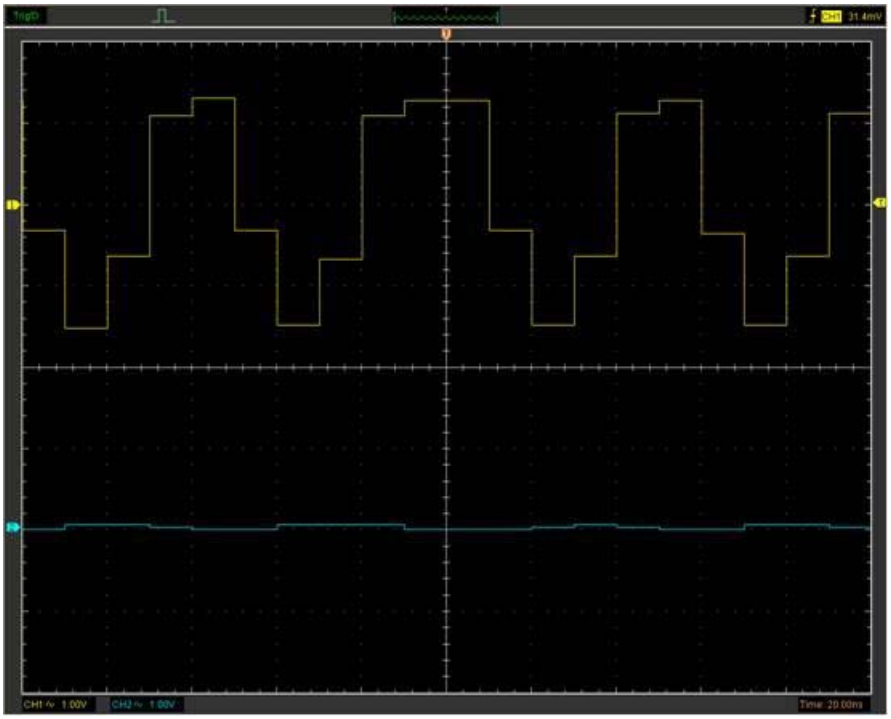
3. Change the waveform position:



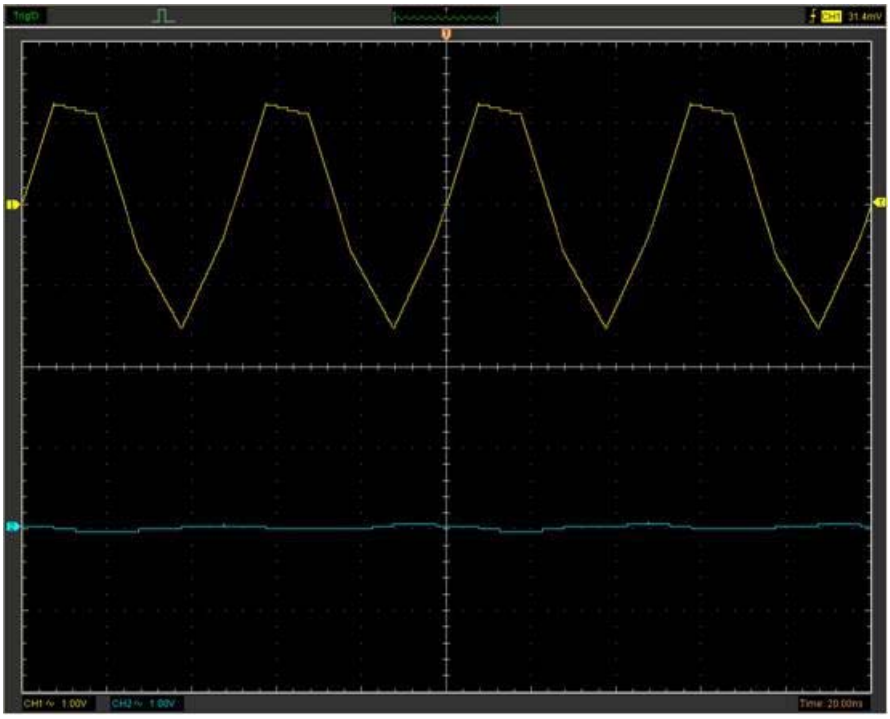
Interpolation

At the time base 40ns/div or faster, user can use the 3 different interpolation mode to get waveforms of different smoothness.

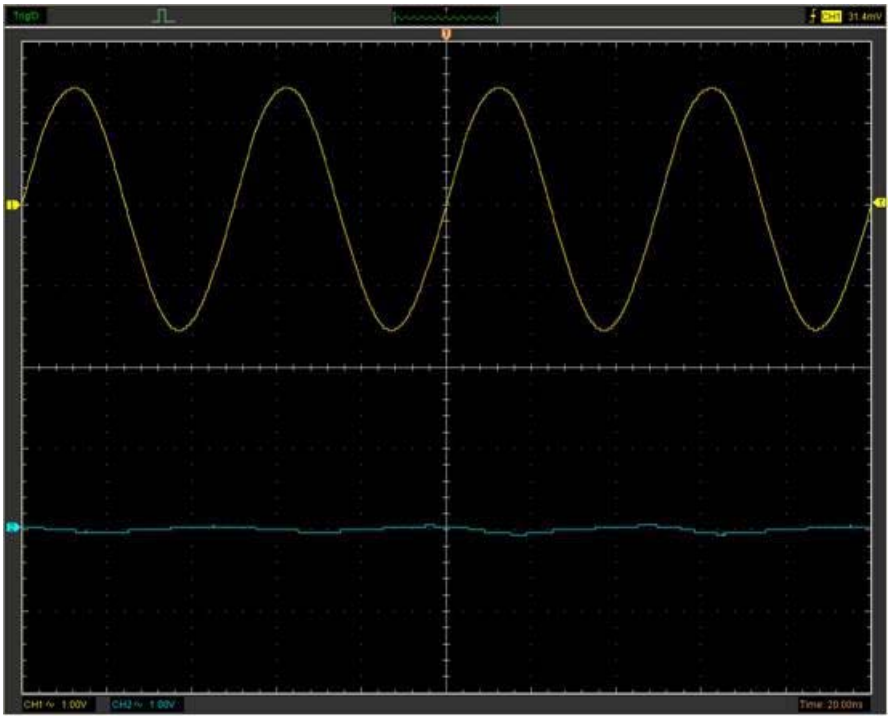
The **Step** Interpolation



The **Linear** Interpolation



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



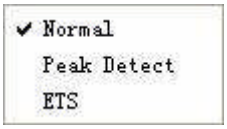
Note: The default interpolation mode is Sin(x)/x.

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.

Acquisition Modes

There are two acquisition modes: Normal and Average.

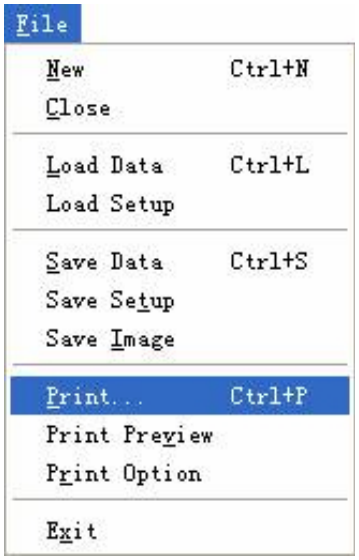


Normal: In this acquisition mode, the oscilloscope samples the signal in evenly spaced intervals to construct the waveform.

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.

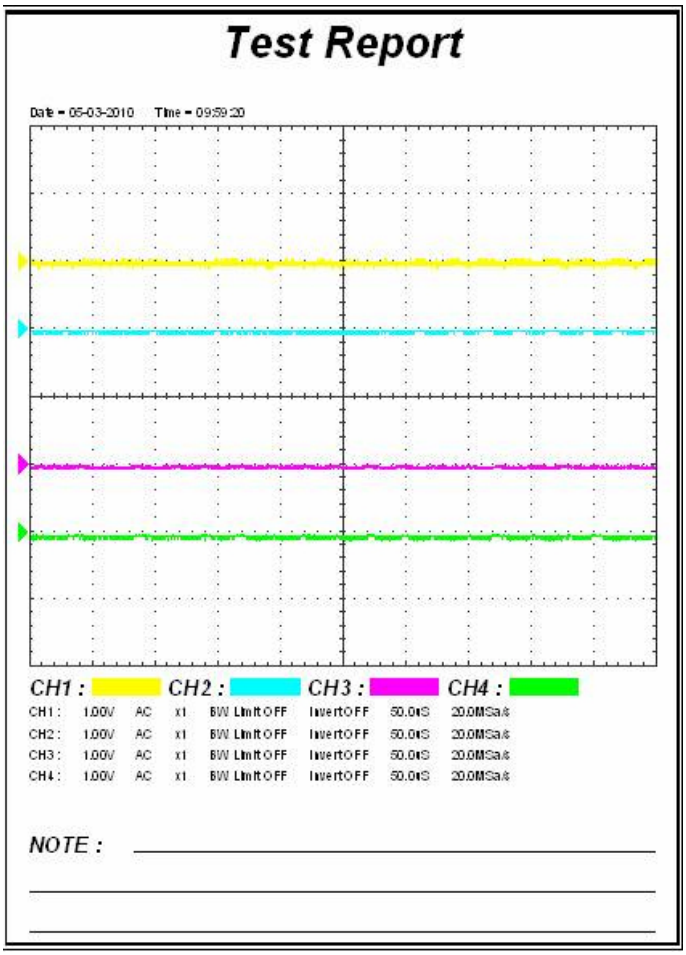
ETS: In this acquisition mode, you can test high frequency cycle signal.

Print And Print Preview



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

The Print report



2. Click the “**PrintPreview**” in “**File**” menu to get into the Preview window.

In”**PrintPreview**” 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 4 Application Example

- **Sample Measurement**
- **Pass/Fail Test**
- **Capturing a Single-Shot Signal**
- **The Application of the X-Y**
- **Taking Cursor Measurement**

Simple Measurement

To acquire and display a signal, please do the steps as follows:

1. Connect signal to **CH1** by using probe

2. Click the button on toolbar or "**Acquire -> Auto Setup**" on menu.

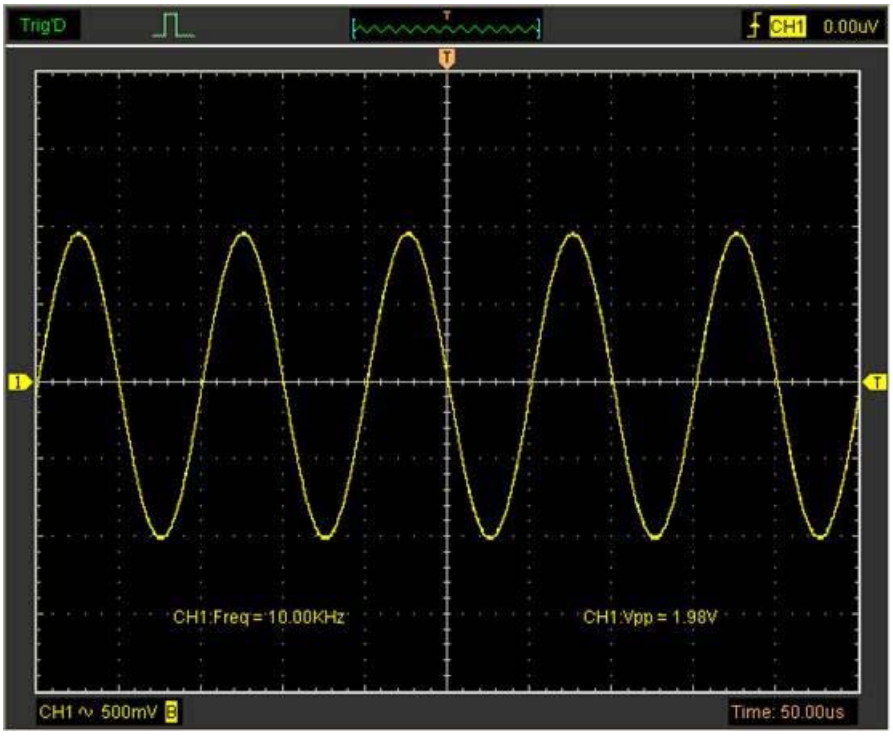
The DSO set the vertical, horizontal, and triggers controls at the best status automatically. Also, you can adjust the controls to meet your measurement to optimize the waveform display.

To measure the frequency and "**Vpp**", you can do these steps as follows:

1. Click the "**Measure->Horizontal->Frequency**" button, the frequency of the signal display on the bottom of the waveform interface.

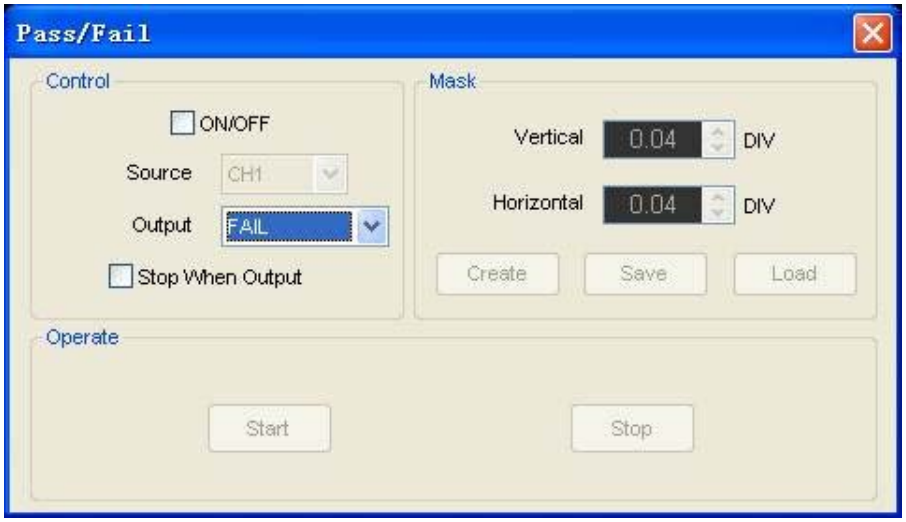
2. Click the "**Measure->Vertical->Peak-to-Peak**" button, the "**Vpp**" of the signal will also display on the bottom of the waveform interface.

3. To clear the measurement on the waveform interface, click the "**Measure->Clear Measure**" button.

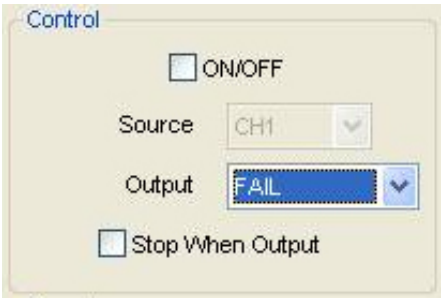

Trig'D
CH1 0.00uV
CH1:Freq = 10.00KHz
CH1:Vpp = 1.98V
CH1 ∿ 500mV
Time: 50.00us

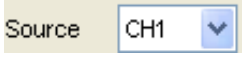
Pass/Fail Test

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

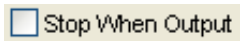


Source:


Select the **Pass/Fail** channel

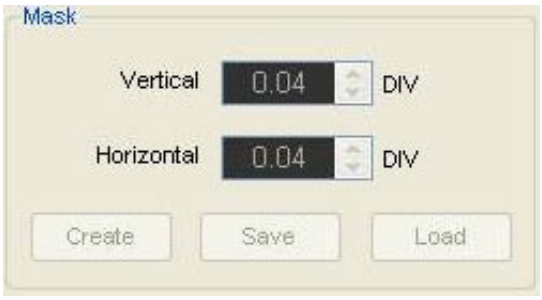
Output:

Select the **Pass/Fail** output condition

Stop When Output: 

If it was checked, the Pass/Fail will stop when output.

Mask Setting

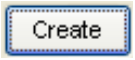


Vertical:

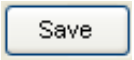
Set the vertical limit range

Horizontal:

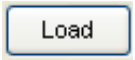
Set the horizontal limit range

"Create" button: 

Click this button to create Pass/Fail area according to the mask

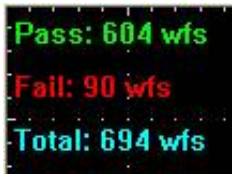
"Save" button: 

Click this button to save the setups to file.

"Load" button: 

Click this button to load the saved setups file.

Information Display



Fail:
It shows the fail waveform number

Pass:
It shows the pass waveform number

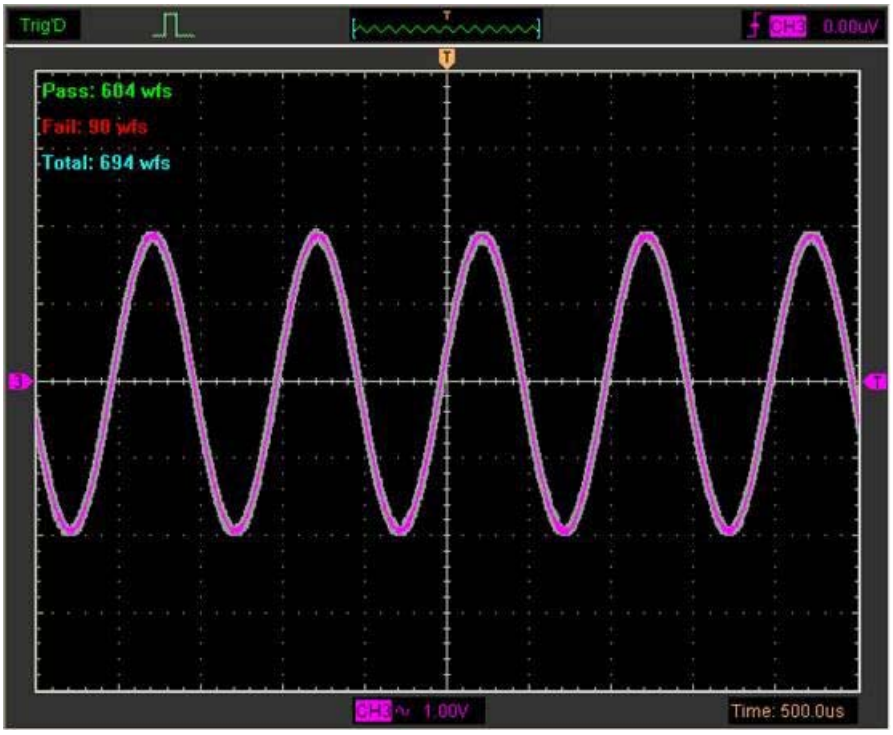
Total:
It shows the I total **Pass/Fail** waveform number

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.

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 X 10.

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 Volts/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’s usually 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.

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 button.

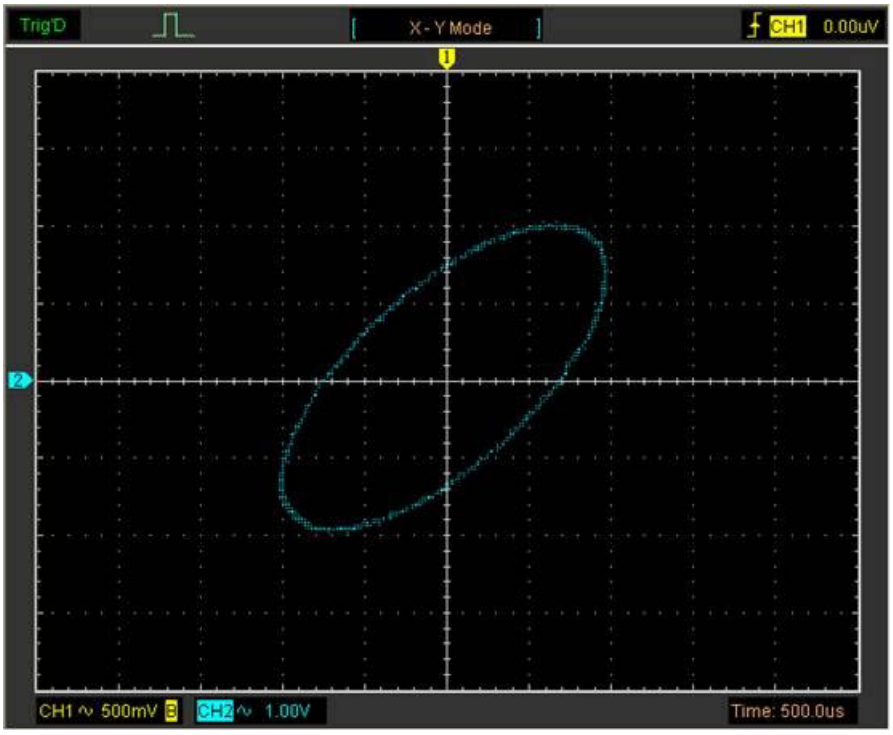
4. Adjust the vertical scale and offset to display approximately the same amplitude signals on each channel.

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.

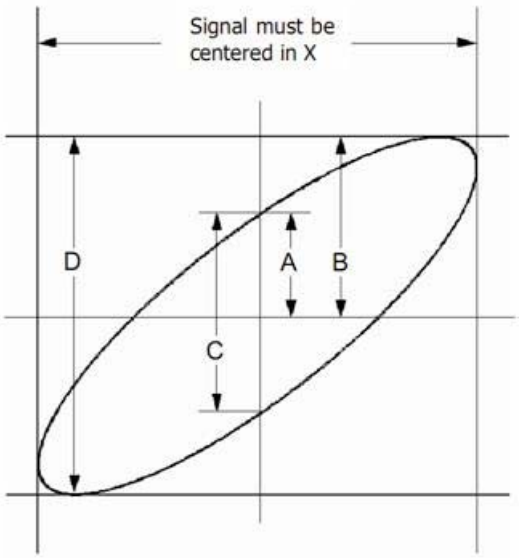
6. Adjust the scale and offset of the horizontal and vertical to a desirable waveform display. The following picture shows a typical example.

7. Apply the Ellipse Method to observe the phase difference between the two channels.

Signal in X-Y Format:



Instruction of the Ellipse Method



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

From the formula above:

θ = ±arcsine (A/B) or ±arcsine (C/D)

θ must be in the range of (0~π/2) or (3π/2~2π) 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 (π/2~π) or (π~3π/2).

Taking Cursor Measurements

Use cursors to make time and amplitude measurements on a waveform quickly.

Measure the Peak Frequency or Time of the First Sine Waveform

Do these steps:

1. Click "**Cursor->Source**", select CH1 (select CH2 if you want measure CH2).

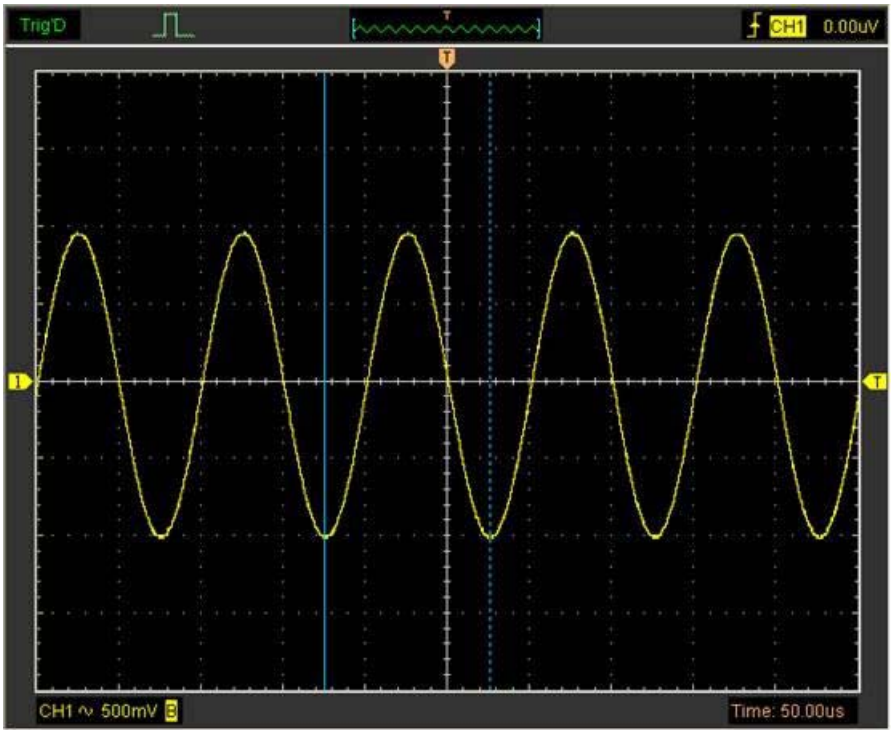
2. Click "**Cursor->Type**", select Vertical.

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.

Measure the Frequency and Time:



Read the details showing in the status bar.

Measure the Amplitude of the First Waveform Peak of the Waveform

Do these steps:

1. Click "**Cursor->Source**", select CH1 (select CH2 if you want measure CH2).

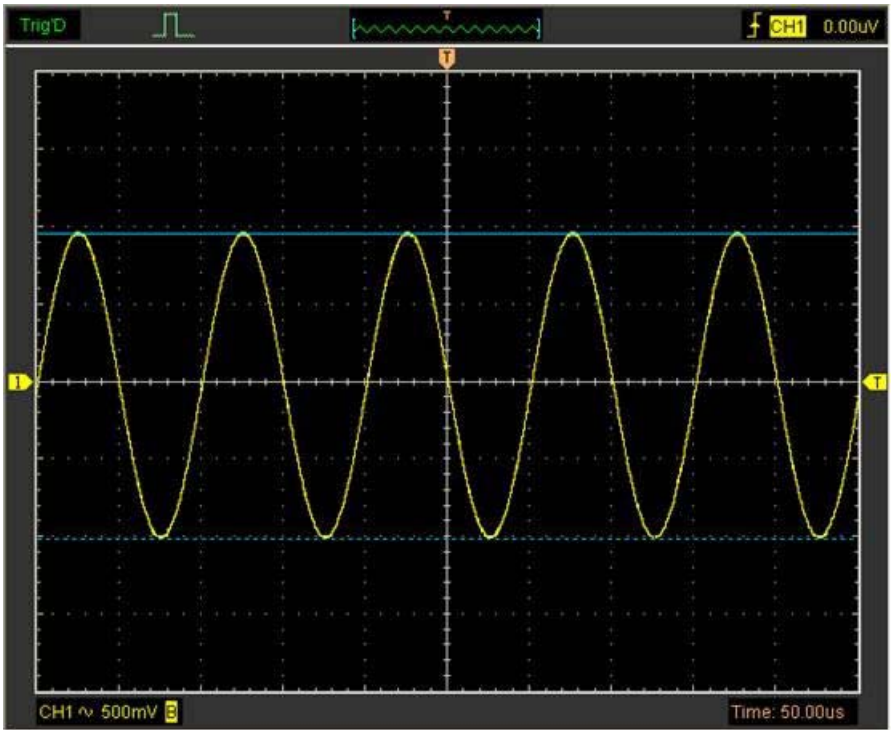
2. Click "**Cursor->Type**", select Horizontal.

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.

Measure the Amplitude:



Read the details showing in the status bar.

Volt: 1.98V

Trace the Amplitude of a fixed position on X-axis in a Waveform

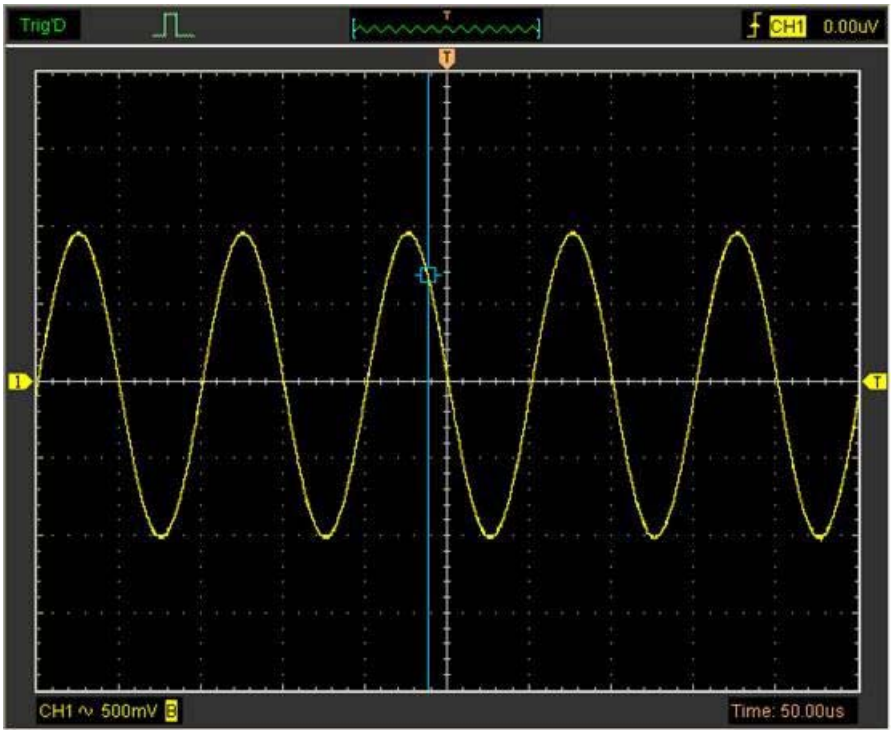
Do these steps:

1. Click "**Cursor->Source**", select CH1 (select CH2 if you want trace CH2).

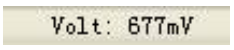
2. Click "**Cursor->Type**", select Trace.

3. Click the cursor at the position that you want traced of the wave in the waveform window.

Trace the Amplitude:



Read the details showing in the status bar.



Note: Click "**Cursor->Type**", select "**Cross**", you can measure time and amplitude at one time.

Save Data

User can use DSO3000 series osilloscope sotfware to save the waveform data as some type file.

The DSO3000 series oscilloscope have four channels which display the real time test data .To save the data user should do the following steps:

1. Click File->Save Data in main menu.

Then the Save Window is pushed up.

2.There are CH1,CH2,CH3,CH4 and ALL items in the down list.
CH1:Save CH1 data.
CH2:Save CH2 data.
CH3:Save CH3 data.
CH4:Save CH4 data.
ALL:Save CH1,CH2,CH3,CH4 DATA.

3.User can select the channel they want to save. For example, CH1 is selected.Then cick the button "OK". Then the "Save Data As..window" is pushed up.User Can select one of the file type from(*.txt,*.csv,*.rfc,*.xls,*.doc)and select the save path and file name.
4.Click the button "Save". The channel is saved.

Chapter 5 Wave Generator

Generate the Sine wave

To output a Sine Wave ,please do the following Steps:

1.Press the check box "On/Off" to open the wave ouput funtion.

2.Select the Wave Type "Sine Wave".

3.Set the Wave Parameter:

Frequency:Set the output wave frequency.

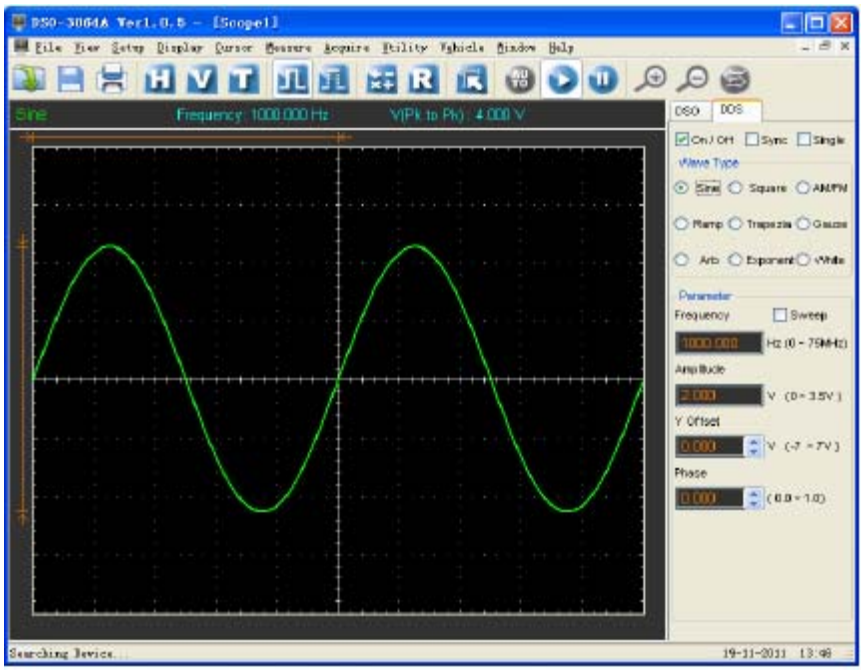
Sweep:Set the output wave to sweep.

Amplitude:Set the output wave amplitude.

Y Offset:Set the output wave vertical level offset.

Phase:Set the output wave phase.

The Sine wave window as the following:



Generate the Square wave

To output a Square Wave ,please do the following Step:

1.Press the check box"On/Off" to open the wave output function.

2.Select the Wave Type "Square".

3.Set the Wave Parameters:

Frequency:Set the output wave frequency.

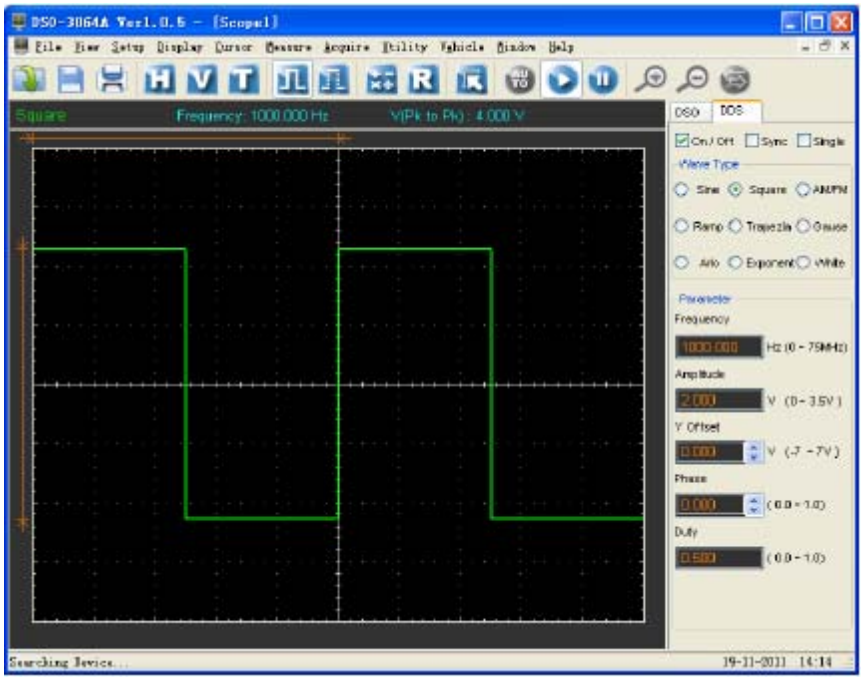
Amplitude:Set the output wave amplitude.

Y Offset:Set the output wave vertical level offset.

Phase:Set the output wave phase.

Duty:Set the duty of the output wave.

The Square wave window as the following:



Generate the AM/FM wave

To output an AM/FM Wave ,please do the following Steps:

1.Press the check box"On/Off" to open the wave output function.

2.Select the Wave Type"AM/FM".

3.Set the Wave Parameters:

Frequency:Set the output wave frequency.

Amplitude:Set the output wave amplitude.

Y Offset:Set the output wave vertical level offset.

Phase:Set the output wave phase.

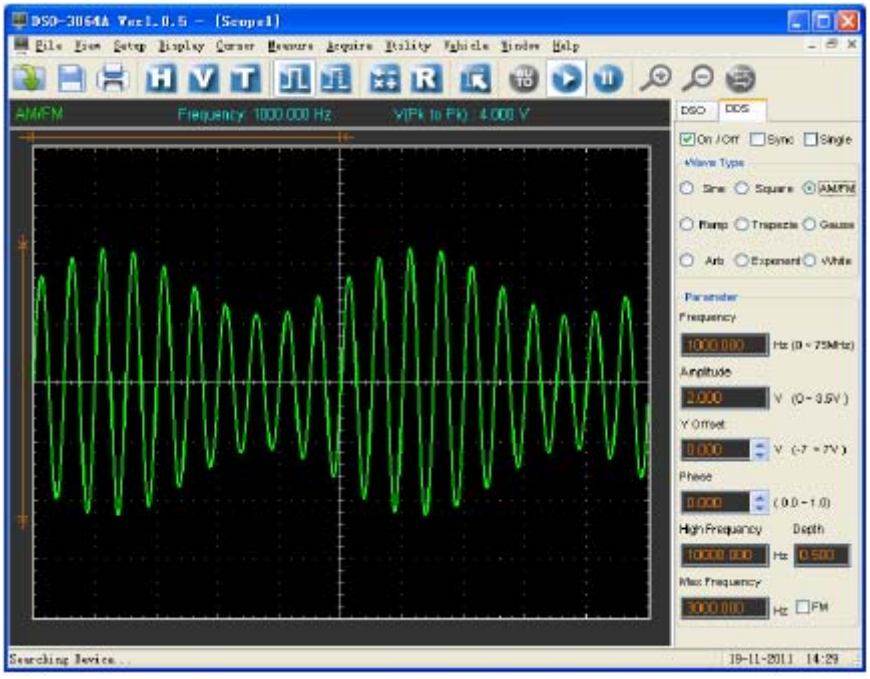
High Frequency:Set the output wave Hight Frequency.

Depth:Set the output wave Depth.

Max Frequency:Set the output wave Max Frequency.

FM: Change the output wave "AM" into "FM".

The AM/FM wave window as following:



Generate the Ramp wave

To output a Ramp Wave,please do the following steps:

1.Press the check box"On/Off" to open the wave output function.

2.Select the Wave Type "Ramp".

3.Set the Wave Parameters:

Frequency:Set the output wave frequency.

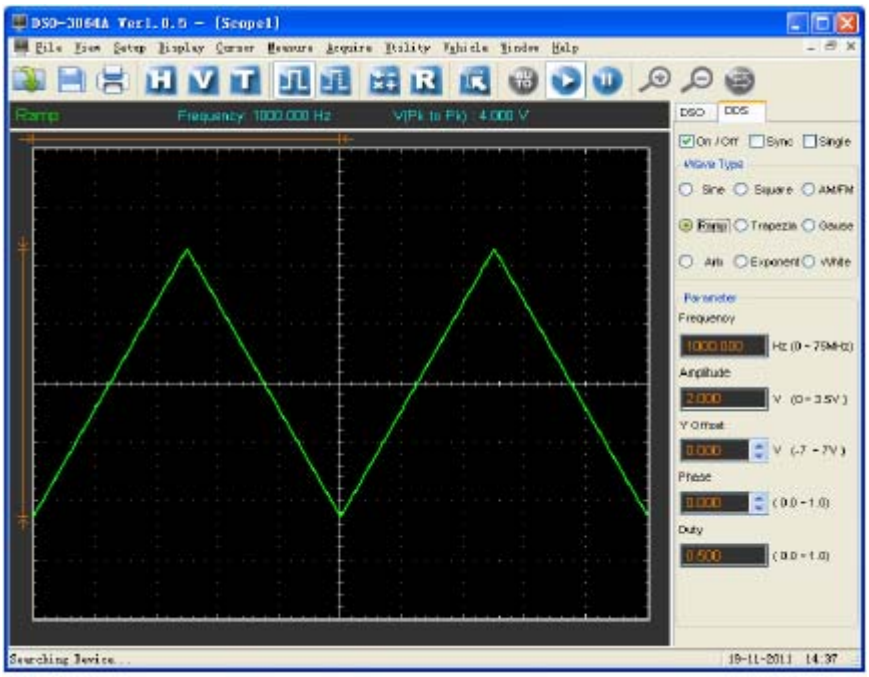
Amplitude:Set the output wave amplitude.

Y Offset:Set the output wave vertical level offset.

Phase:Set the output wave phase.

Duty:The the duty of the output wave.

The Ramp wave window as the following:



Generate the Trapezia wave

To output a Trapezia wave ,please do the following steps:

1.Press the check box"On/Off" to open the wave output function.

2.Select the Wave Type "Trapezia".

3.Set the Wave Parameters:

Frequency:Set the output wave frequency.

Amplitude:Set the output wave amplitude.

Y Offset:Set output wave vertical level offset.

Phase:Set the output wave phase.

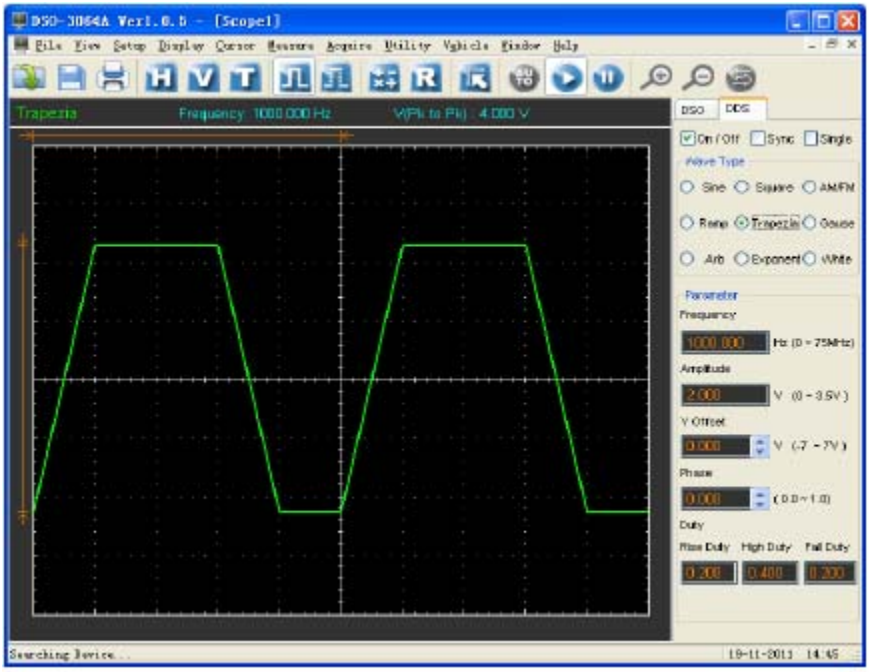
Duty:Set the output wave duty.

Rise Duty:Set the output wave rise duty.

High Duty:Set the output wave high duty.

Fall Duty:Set the output wave fall duty.

The Trapezia wave window as the following:



Generate the Gause wave

To output a Gause Wave,please do the following steps:

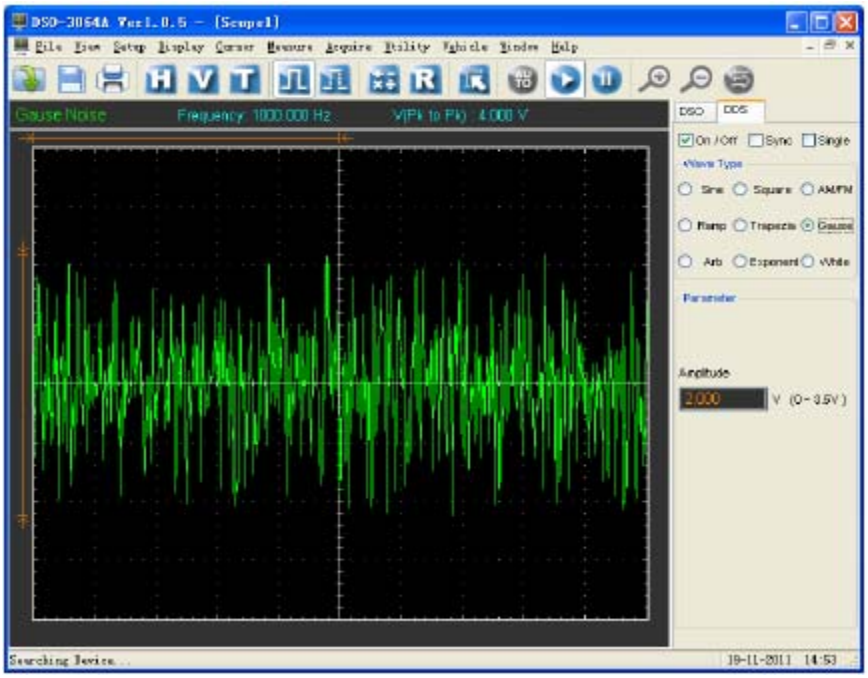
1.Press the check box "On/Off" to open the wave output function.

2.Select the Wave Type "Gause".

3.Set the Wave Paramters:

Amplitude: Set the output wave amplitude.

The Gause wave window as the following:



Generate the Arb wave

To output an Arb Wave,please do the following steps:

1.Press the check box "On/Off" to open the wave output function.

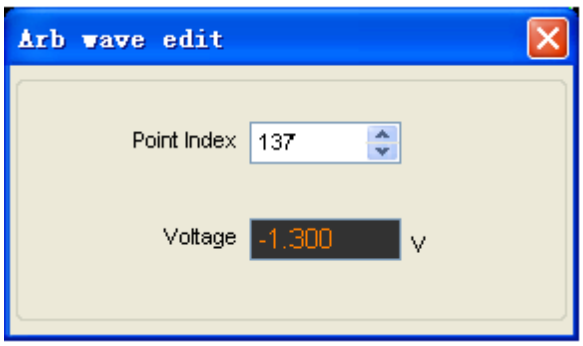
2.Select the Wave Type "Arb".

3.Set the Wave Parameters:

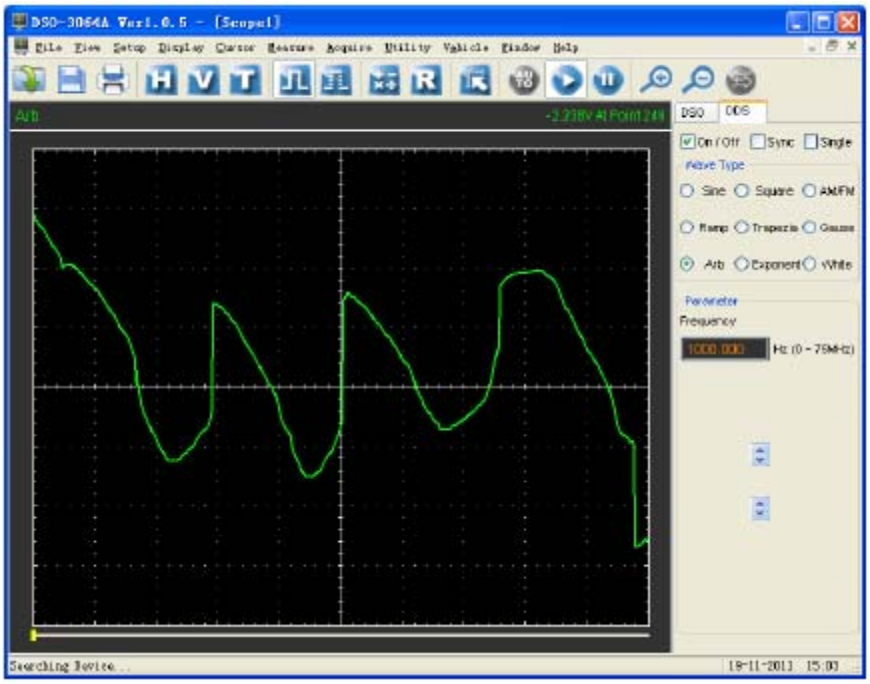
Frequency:Set the output wave frequency.

4.Set the output wave.

User can set the output wave by moving the mouse .Also,user can double click the mouse left key to set level of the wanted point index.



The Arb wave window as the following:



Generate the Exponent wave

To output an Exponent Wave,please do the following steps:

1.Press the check box "On/Off" to open the wave output function.

2.Select the Wave Type "Exponent".

3.Set the Wave Parameters:

Frequency:Set the output wave frequency.

Amplitude:Set the output wave amplitude.

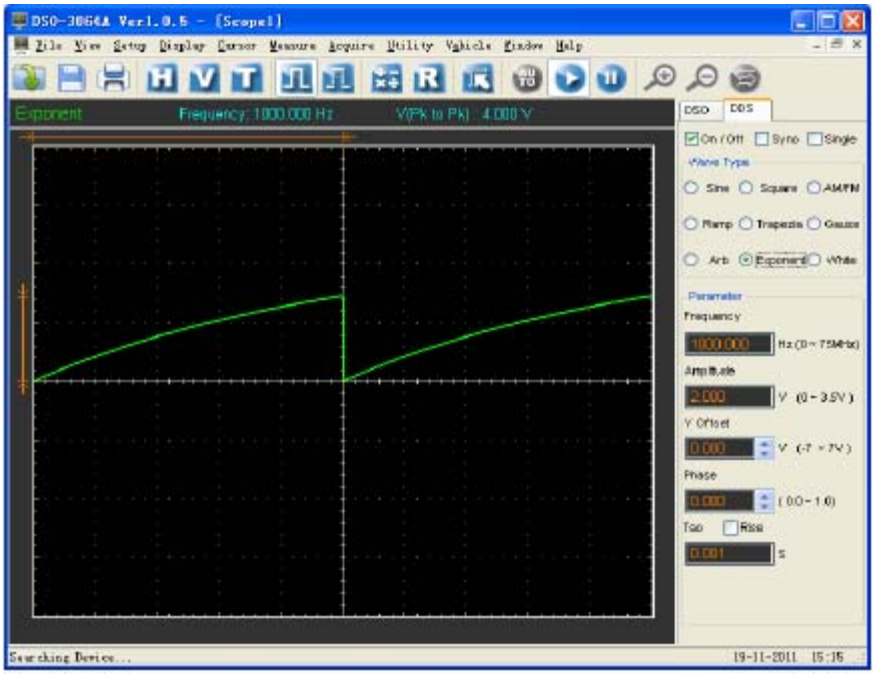
Y Offset:Set the output wave vertical level offset.

Phase:Set the output wave phase.

Tao:Set the ouput wave Tao param .

Rise:Set the output wave slope.

The Exponent wave window as the following:



Generate the White noise wave

To Generate a White noise Wave ,please do the following steps:

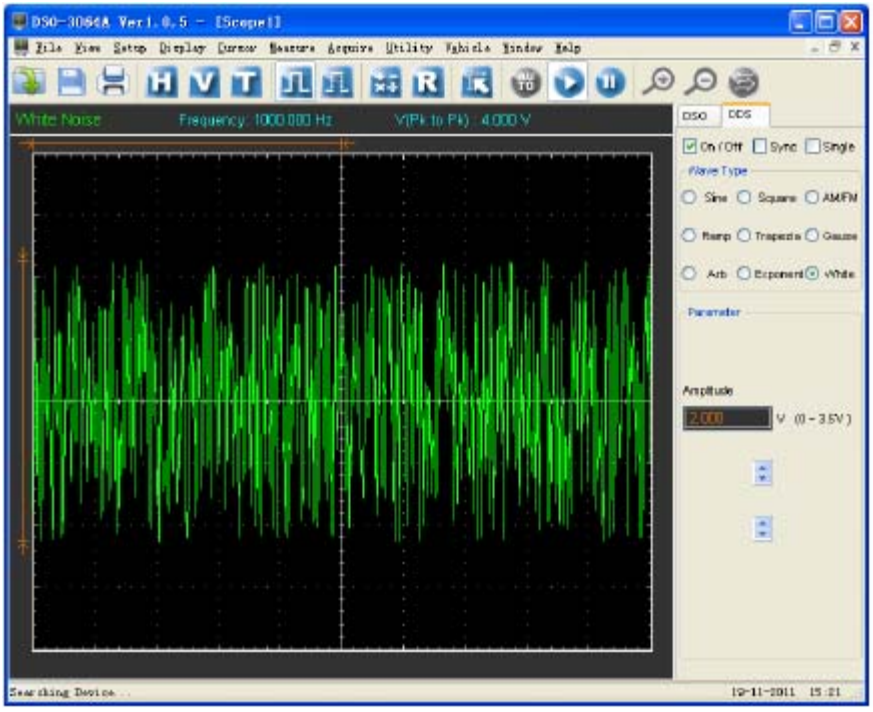
1.Press the check box"On/Off" to open the wave output function.

2.Select the Wave Type "White".

3.Set the Wave Parameters:

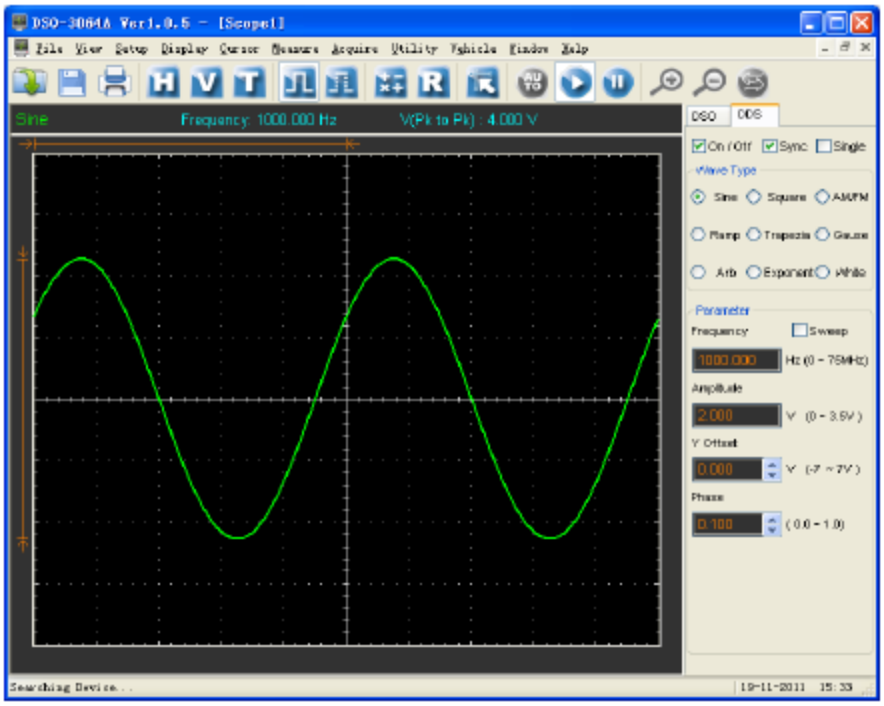
Amplitude:Set the output wave amplitude.

The White noise wave window as the following:

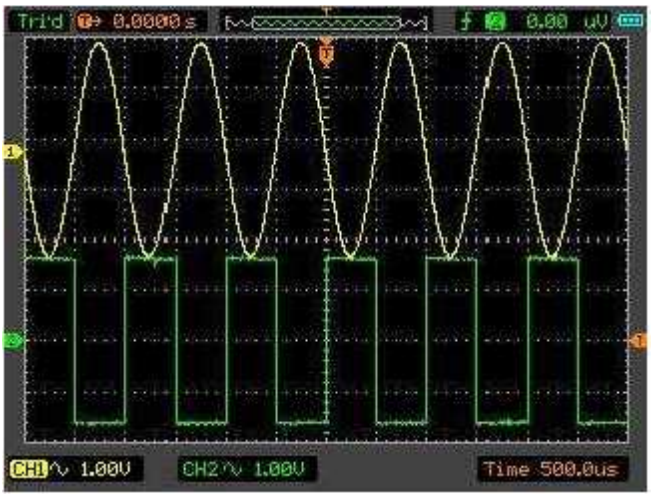


Synchronized output

If you generate a waveform by software, there is a synchronized signal output from SYNC terminal. The signal is square waveform, the frequency of which is equal to the waveform you generated. For example, if you generated an 1 KHz, sine waveform, you will also generate an 1 KHz, square waveform simultaneously. The software setting is following figure.



The output ware of the Oscilloscope is the following figure.

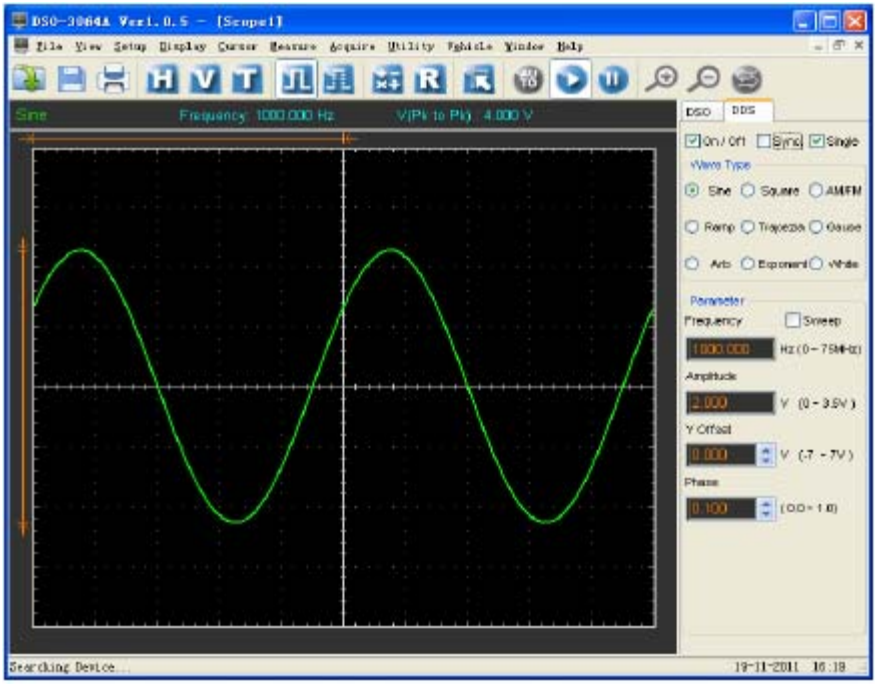


The waveform on CH1 is the sine waveform you generated from the OUTPUT terminal, and that on CH2 is the synchronized signal from SYNC OUT terminal.□

Single

Select the function to output waveform only once.

The Setting as the following figure:



Chapter 6 Appendix

■ **Appendix A: Specifications**

■ **Appendix B: General Maintenance**

Appendix A: Specifications

Specifications Table:

Input	
Max. sample rate	200MS/s (Single Channel) 100MS/s (Dual Channels)
Channels	4 Channels
Bandwidth	60MHz (-3dB)
Vertical resolution	8 bits/channel
Gain range	10mV ~ 5V/div @ x1 probe(10mV, 20mV, 50mV, 100mV, 200mV, 500mV, 1V, 2V, 5V/div1,2,5 sequence) 100mV ~ 50V/div @ x10 probe 1V ~ 500V/div @ x100 probe 10V ~ 5KV/div @ x1000 probe
Range	8 divisions
Offset level	+/-4 divisions
Coupling	AC, DC,GND
Offset increments	0.02 div
Impedance	1M ohm
DC accuracy	+/-3%
Input protection	35Vpk (DC + peak AC < 10 kHz, without external attenuation)
Display Mode	Y-T, X-Y
Timebase	
Timebase range	5ns/div ~ 1000s/div(1-2-5 sequence)
Acquisition mode	Realtime sampling: 5ns /div ~ 200ms/div. Roll mode: 500ms/div ~ 1000s/div
Range	10 divisions
Buffer Size	10K ~ 16M points (Single Channel)
Trigger	
Type	Edge trigger, Pulse trigger
Mode	Auto, Normal and Single
Autoset	Yes
Range	10 divisions

Trigger level	+/-4 divisions
Settabillity	0.02 div increments
Math	
Measurements	Vpp, Vmax, Vmin, Vmean, Vrms, Vamp, Vtop, Vbase, Vmid,positive overshoot, negative overshoot, cycle mean, cycle RMS, period, frequency, positive pulse width, negative pulse width, rise time (10%~90%), fall time (10%~90%), positive duty cycle, negative duty cycle
Math	Addition, Subtraction, Multiplication, FFT
FFT	Rectangular, Hanning, Hamming, Blackman Window
Physical	
Interface	USB2.0 (USB1.1 compatible)
Power	External power source required.(8.5v dc)
Dimensions	224 x168 x 37(mm)

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.

Cleaning
Inspect the device and probes as often as operating conditions require. Make sure the device disconnect form all power sources.

To clean the exterior surface, perform the following steps:

1. Remove loose dust on the outside of the oscilloscope and probes with a lint-free cloth. Use care to avoid scratching the clear glass display filter.

2. Use a soft cloth dampened with water to clean the device.

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