



**DPO6000,MPO6000 Series**

**Digital Phosphor Oscilloscope**

**V 1.1**



## **Copyright Statement**

### **Copyright**

Qingdao Hantek Electronic Co.,Ltd.

### **Statement**

Qingdao Hantek Electronic Co., Ltd. reserves the right to amend this document without prior notice. Qingdao Hantek Electronic Co., Ltd. promises that the information provided is correct and reliable, but does not guarantee that this document is infallible. Please make sure that the specifications of relevant technical documents used are the latest and valid version by yourself before using this product. If you require the cooperation of products, patents or works of a third party when your company use the documents or products of Qingdao Hantek Electronic Co., Ltd., your company shall be responsible for obtaining the consent and authorization of the third party. The aforesaid consent and authorization is not the responsibility of our company to guarantee.

## Technical Support

If you have any question or ambiguity in the process of using products of Qingdao Hantek Electronic Co., Ltd., you can get the service and support through the following ways:

A: Please contact the local distributor of Qingdao Hantek Electronic Co., Ltd.;

B: Please contact the local office directly under Qingdao Hantek Electronic Co., Ltd.

C: Please contact the headquarters of Qingdao Hantek Electronic Co., Ltd.

Contact Method of our company:

Qingdao Hantek Electronic Co., Ltd.

<http://www.Hantek.com>

Address:35# Building No.780 Baoyuan Road,High-tech District,Qingdao China

Zip Code:266114

Telephone: 0532-88705792

Fax: 0532-88705691

Email: [service@Hantek.com](mailto:service@Hantek.com)

Technical Support:

Telephone:0532-88703687

Email: [support@Hantek.com](mailto:support@Hantek.com)

## Summary of general safety matters

### General Safety Summary

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

#### **Avoid fire and personal injury.**

Only authorized personnel can perform maintenance.

#### **Use Proper Power Cord.**

Only the exclusive power cord designed for the instrument and authorized for use within the local country could be used.

#### **Connect and disconnect correctly.**

Connect the probe to the oscilloscope before connecting it to the circuit being measured; Disconnect the probe and the circuit measured before disconnecting the probe from the oscilloscope.

#### **Ground the Instrument.**

To avoid electric shock, the instrument is grounded through the Earth lead of the power cord. The earth lead must be connected to the ground. This product must be grounded before connecting any input or output terminals.

#### **Connect the probe correctly.**

If a probe is used, the probe ground lead must be connected to earth ground. Do not connect the ground lead to high voltage. Improper way of connection could result in dangerous voltages being present on the connectors, controls or other surfaces of the oscilloscope and probes, which will cause potential hazards for operators.

#### **Observe All Terminal Ratings.**

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

#### **Use Proper Overvoltage Protection.**

Ensure that no overvoltage (such as that caused by a bolt of lightning) can reach the product. Otherwise, the operator might be exposed to the danger of an electric shock.

#### **Do Not Operate Without Covers.**

Do not operate the instrument with covers or panels removed.

#### **Provide Adequate Ventilation.**

Inadequate ventilation may cause an increase of temperature in the instrument, which would cause damage to the instrument. So please keep the instrument well ventilated and inspect the air outlet and the fan regularly.

#### **Use Proper Fuse.**

Please use the specified fuses.

#### **Avoid Circuit or Wire Exposure.**

Do not touch exposed junctions and components when the unit is powered on.

**Do Not Operate With Suspected Failures.**

If you suspect that any damage may occur to the instrument, have it inspected by the qualified maintenance personnel.

**Do Not Operate in Wet Conditions.**

To avoid short circuit inside the instrument or electric shock, never operate the instrument in a humid environment.

**Do Not Operate in an Explosive Atmosphere.**

To avoid personal injuries or damage to the instrument, never operate the instrument in an explosive atmosphere.

**Keep Instrument Surfaces Clean and Dry.**

To avoid dust or moisture from affecting the performance of the instrument, keep the surfaces of the instrument clean and dry.

**Prevent Electrostatic Impact.**

Operate the instrument in an electrostatic discharge protective environment to avoid damage induced by static discharges. Always ground both the internal and external conductors of cables to release static before making connections.

**Handle with Caution.**

Please handle with care during transportation to avoid damage to keys, knobs, interfaces, and other parts on the panels.

**Safety Notices and Symbols**

Safety Terms on the Product. There may be the following terms on the product:

**DANGER** It calls attention to an operation, if performed, could result in injury or hazard immediately.

**WARNING** It calls attention to an operation, if performed, maybe not result in injury or hazard immediately.

**CAUTION** It calls attention to an operation, if performed, could result in damage to the product or other devices connected to the product.

Safety Symbols on the Product. There may be the following symbols on the product:



Safety Warning   Protective   Test Ground   Shell Earth Terminal

**Measurement Category**

DPO6000/MPO6000 series digital oscilloscopes can make measurements in Measurement Category I.

**WARNING**

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

**Measurement Category Definitions**

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

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

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

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

**Ventilation Requirement**

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

**WARNING**

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

**Working Environment****Temperature**

Operating: 0°C to +50°C

Non-operating: -40°C to +70°C

**Humidity**

0°C to +30°C: ≤95% relative humidity

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

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

**WARNING**

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

**Altitude**

Operating: below 3 km

Non-operating: below 15 km

**Care and Cleaning****Care**

---

Do not store or leave the instrument where its LCD may be exposed to direct sunlight for long periods of time.

---

**CAUTION**

To avoid damage to the oscilloscope or probe, do not expose it to fog, liquids or solvent.

---

**Cleaning**

Check the oscilloscope and probe regularly according to its operating conditions. Please follow these steps to clean the external surface of the instrument:

Remove dust from outside oscilloscopes and probes with a cloth that doesn't fluff. Please be careful not to scratch the bright screen filter material.

Use a soft cloth dampened with water to clean the oscilloscope. For more thorough cleaning, use 75% isopropanol as an aqueous solvent.

---

**WARNING**

To avoid damage to the surface of the oscilloscope or probe, do not use any corrosive or chemical cleaning reagent

---

**Product End-of-Life Handling**

Recycle:

Production of the equipment requires extraction and use of natural resources. If the scrap of this product is not handled properly, then the substances the equipment contain could be harmful to the environment or human health. To avoid the release of such substances into the environment and reduce the use of natural resources, we recommend you to recycle this product appropriately to ensure that most materials are reused properly.

## DPO6000Z/MPO6000 Series Overview

DPO6000/MPO6000 Series is a Oscilloscope featuring extremely high memory depth, wide dynamic range, clear display,high waveform capture rate and as many as 16 kinds of triggering functions, 5 kinds of serial decoding functions it is a useful commissioning instrument for various fields such as communication,aerospace, defense, embedded systems, computers, research and education.

### Main features:

- ★1 GSa/s real-time sample rate of the analog channels; 64 Mpts standard and 128M maximum(6000EDU series) memory depth
- ★1 GSa/s real-time sample rate of the digital channels
- ★200 MHz, 100 MHz and 80 MHz analog channel bandwidth
- ★4 analog channels, 16(4\*4) digital channels(MPO6000 series;DPO6000 can update to use digital channels by buying LP104 digital probe)
- ★2-channel/3-channel signal source(MPO6000EDU is 3-channel ),13 kinds of waveform inside,4 sets of arbitrary waveform,200M sample rate,8Kpts waveform length
- ★60,000 wfms/s (dots display)/400,000 wfms/s (dots display quick acquisition mode) waveform capture rate
- ★Segmented acquisition function,support to capture up to 80,000 sections
- ★256 grade color display
- ★Low base noise, 500 uV/div to 10 V/div ultra-wide vertical dynamic range
- ★7.0 inch WVGA (800\*480) TFT LCD, with ultra-wide screen, vivid picture, low power consumption and long service life
- ★7 inch capacitive touch screen,support multi-touch(touch screen is optional)
- ★Adjustable waveform brightness and display brightness
- ★Up to 16 kinds of trigger functions, including 5 kinds of protocol triggers
- ★Supply 5 serial decoding options
- ★Auto measurement of 42 kinds of waveform parameters (with statistics)
- ★5 bits digital voltage meter and 6 bits hardware frequency indicator function
- ★Bode diagram function(the oscilloscopes with signal source function can use)
- ★Event search function
- ★2-window display function
- ★Built-in FFT function
- ★Multiple waveform math operation functions 【MATH】
- ★Standard interfaces: USB Device, USB Host, LAN
- ★Wifi(MPO6000EDU/DPO6000EDU can use Wifi)
- ★Optional interfaces: HDMI,UART,AUX
- ★Pass\_fail /trig\_out(EDU standard,non-EDU optional)
- ★Conform to LXI CORE 2011 DEVICE class instrument standards; enable quick, economic and efficient creation and reconfiguration of test system
- ★Supports remote command control
- ★Built-in help to facilitate information acquisition
- ★Supports multiple languages
- ★Novel and delicate industrial design and easy operation

**This list of product models**

MPO6000/DPO6000 series include the following modes. Unless otherwise specified, this manual takes MPO6204EDU as an example to illustrate the functions and operation methods of MPO6000/DPO6000 series.

<b>Modes</b>	<b>channel</b>	<b>Analog bandwidth</b>	<b>Signal generator</b>	<b>Logic analyzer</b>	<b>Wifi</b>
DPO608XB	2/4	80MHz	-	-	-
DPO610XB	2/4	100MHz	-	-	-
DPO620XB	2/4	200MHz	-	-	-
DPO608XC	2/4	80MHz	25MHz*2	-	-
DPO610XC	2/4	100MHz	25MHz*2	-	-
DPO620XC	2/4	200MHz	25MHz*2	-	-
MPO608XD	2/4	80MHz	25MHz*2	4*4 LA	-
MPO610XD	2/4	100MHz	25MHz*2	4*4 LA	-
MPO620XD	2/4	200MHz	25MHz*2	4*4 LA	-
DPO608XEDU	2/4	80MHz	-	-	Yes
DPO610XEDU	2/4	100MHz	-	-	Yes
DPO620XEDU	2/4	200MHz	-	-	Yes
MPO608XEDU	2/4	80MHz	25MHz*3	4*4 LA	Yes
MPO610XEDU	2/4	100MHz	25MHz*3	4*4 LA	Yes
MPO620XEDU	2/4	200MHz	25MHz*3	4*4 LA	Yes

## 1.Quick Start

This chapter introduces the precautions when using the oscilloscope for the first time, the front/rear panels of the oscilloscope, the user interface and the using method of the built-in help system.

### 1.1.General Inspection

#### 1.1.1.Inspect the packaging

If the packaging or foam plastic cushioning has been damaged, do not dispose it until the machine and accessories have passed both electrical and mechanical tests.

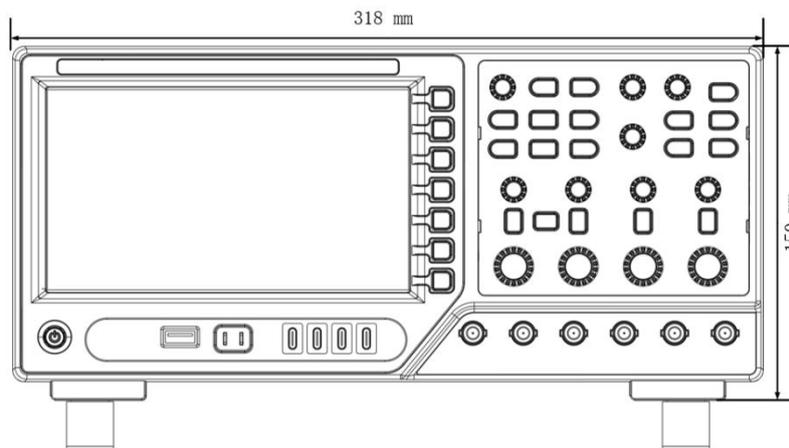
#### 1.1.2.Inspect the instrument

In case of any appearance damage, not working normally, or failure in passing the electrical and mechanical tests, contact the representative in charge of this business.

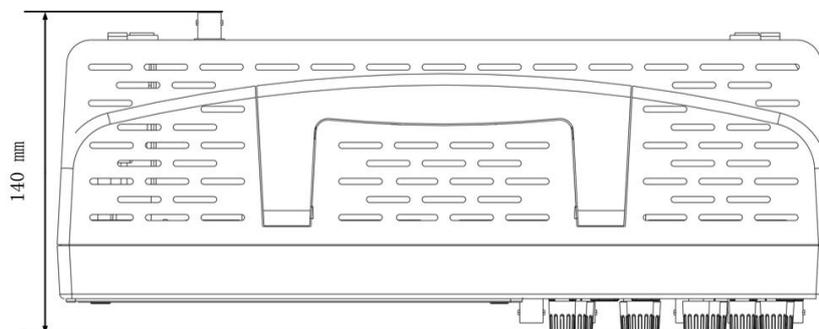
#### 1.1.3.Check the accessories

Please check the accessories according to "Appendix A:Accessories" after the manual. If the accessories are damaged or incomplete, please contact the representative in charge of this business.

### 1.2.Appearance and Dimensions



Front View

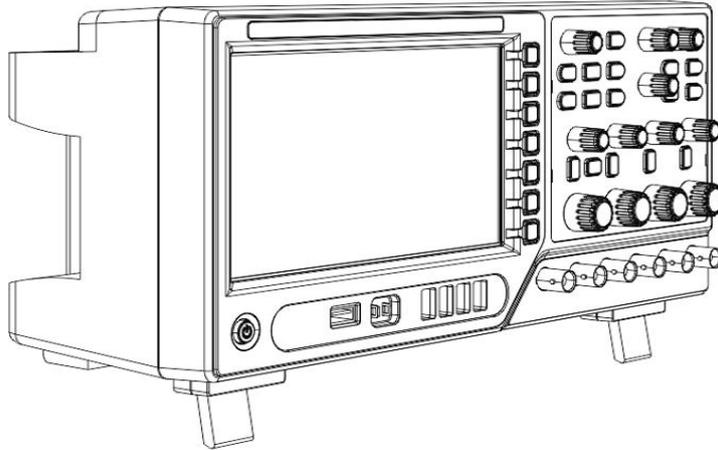


Top View

### 1.3.To Prepare the Oscilloscope for Use

#### 1.3.1.To Adjust the Supporting Legs

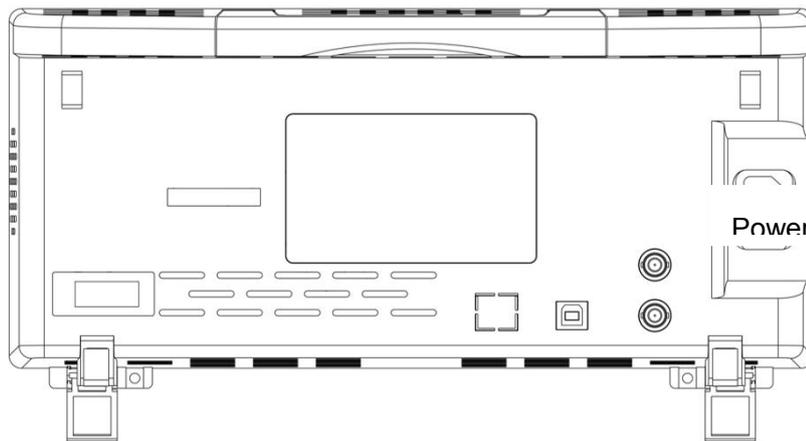
Adjust the supporting legs properly to use them as stands to tilt the oscilloscope upwards for stable placement of the oscilloscope as well as better operation and observation.



To Adjust the Supporting Legs

### 1.3.2.To Connect to Power Cord According to your Demand

The power requirements of the oscilloscope are AC 100-120 V, 50/60/400 Hz or AC 100-240 V, 50/60Hz,50W max. Please use the power cord supplied with the accessories to connect the oscilloscope to the AC power source as shown in the figure below



To Connect to Power Supply

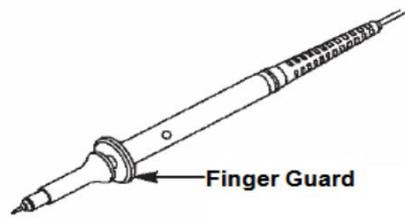
### 1.3.3.Power-on Inspection

When the oscilloscope is energized, pressing the power key  at the lower-left corner of the front panel can start the oscilloscope. During the start-up process, the oscilloscope performs a series of self-tests. In the process of self-test, the keyboard indicator light will be lit in a certain order. After the self-test, the welcome screen is displayed.

### 1.3.4.Probe Check

#### 1.3.4.1.Safety

When using the probe, keep your fingers behind the guard on the probe body to avoid electric shock. Do not touch metallic portions of the probe head while it is connected to a voltage source. Connect the probe to the oscilloscope and connect the ground terminal to ground before you start any measurements.

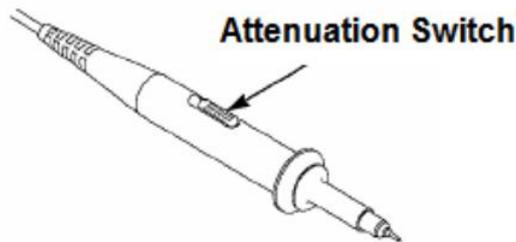


#### 1.3.4.2. Probe Attenuation Setting

Probes are of various attenuation factors which affect the vertical scale of the signal. The Probe Check function is used to verify if the Probe attenuation option matches the attenuation of the probe.

Press a vertical menu button (such as the CH1 MENU button) and select the Probe option that matches the attenuation factor of your probe.

Make sure that the Attenuation switch on the probe matches the Probe option in the oscilloscope. Switch settings are 1X and 10X. When the Attenuation switch is set to 1X, the probe limits the bandwidth of the oscilloscope to 6MHz. To use the full bandwidth of the oscilloscope, be sure to set the switch to 10X.



#### 1.3.5. To Connect the Probe

DPO6000 series oscilloscope standard equipped passive probe, MPO6000 series standard equipped passive probe and digital probe.

##### 1.3.5.1. Connect the passive probe:

1. Connect the BNC terminal of the probe to an analog channel input terminal of the oscilloscope at the front panel.
2. Connect the ground alligator clip or spring of the probe to the circuit ground terminal and connect the probe tip to the circuit to be tested.

##### 1.3.5.2. Connect the logic probe:

1. Connect one terminal of the logic probe to the digital channel input terminal at the front panel of the oscilloscope correctly (digital probe doesn't have directions).
2. Connect the signal under test to the other terminal of the logic probe. MPO 6000 series is equipped with 4 LP104 digital probes (standard).

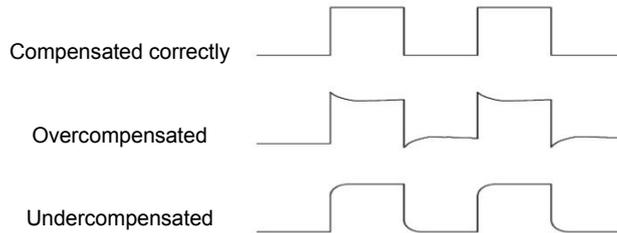
#### 1.3.6. Use of Probe Check Wizard

Every time you connect a probe to an input channel, you should use the probe check wizard to verify that this probe is operating correctly. Use the vertical menu (for example, push the CH1 MENU button) to set the Probe option attenuation factor.

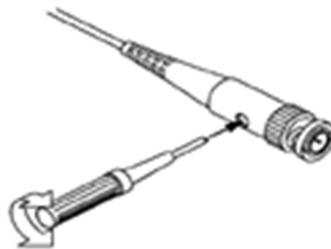
#### 1.3.7. Manual Probe Compensation

Upon the first connection of a probe and an input channel, you should manually perform this adjustment to match the probe to the input channel. Uncompensated or miscompensated probes may lead to errors or faults in measurement. To adjust the probe compensation, follow the steps below:

1. Set the Probe option attenuation in the channel menu to 10X. Set the switch on the probe to 10X and connect the probe to Channel 1 on the oscilloscope. If you use the probe hook-tip, ensure it is firmly inserted onto the probe. Attach the probe tip to the PROBE COMP ~2V@1KHz connector and the reference lead to the PROBE COMP Ground connector. Display the channel and then press the "Auto Scale" button.
2. Check the shape of the displayed waveform.

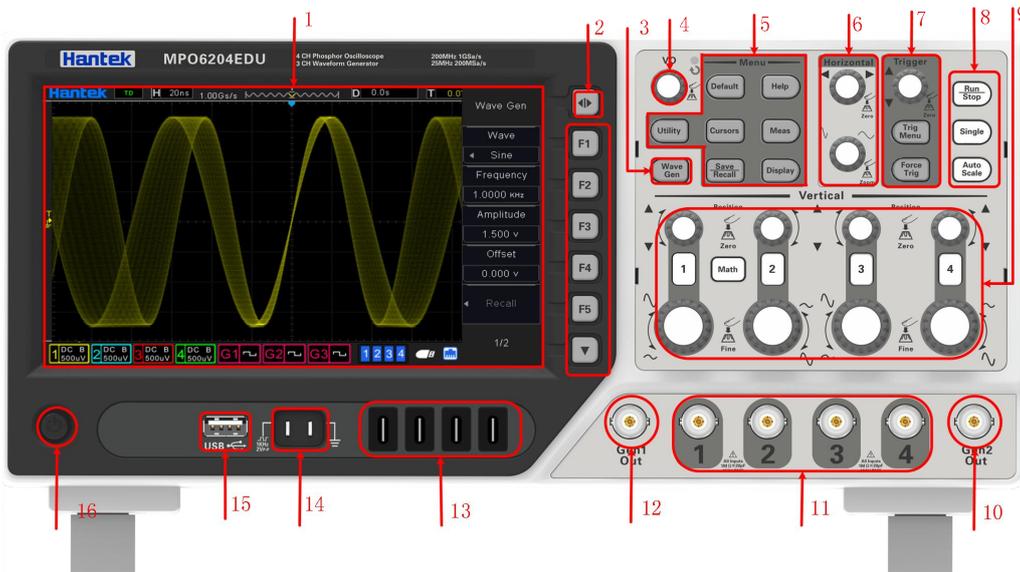


3. If necessary, use a nonmetallic screwdriver to adjust the variable capacity of your probe until the shape of the waveform turns to be the same as the above figure. Repeat this step as necessary. See the figure below for the way of adjustment.



### 1.4. Front Panel Overview

The following content briefly describes and introduces the front panel of this series of oscilloscopes, so that you can be familiar with this series of digital oscilloscopes in the shortest time.

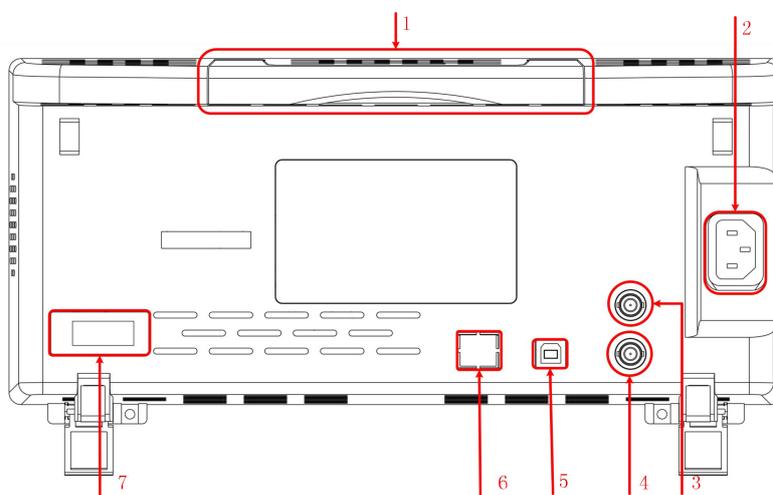


#### Front Panel Descriptions

No.	Description	No.	Description
1	LCD	9	Vertical control system

2	To turn on/off the menu	10	Signal source 1 output channel
3	Signal source function(only the oscilloscopes with signal source function)	11	CH1~CH4 signal input channel
4	Multifunction knob	12	Signal source 2 output channel
5	The functions buttons of menu	13	LA1~LA4 signal input channel (only MPO series oscilloscopes)
6	Horizontal control system	14	Probe compensation signal output terminal/ground terminal
7	Trigger control system	15	USB Host port
8	Shortcut key(Run/Stop,Single,Auto Scale)	16	Power on/off

## 1.5.Rear Panel Overview



### 1.Handle

Pull up the handle vertically for easy carry of the instrument. When you do not need the handle, press it down.

### 2.AC Power Socket

AC power input terminal. The power requirements of this oscilloscope are 100-120 V, 50/60/400 Hz or 100-240 V, 50/60 Hz, 50W max. Use the power cord provided with the accessories to connect the instrument to AC power. Then, you can press the power key at the front panel to start the instrument.

### 3.AUX Trigger Out and Pass/Fail(optional terminal)

#### Trigger Out:

The oscilloscope can output a pulse at each trigger via this interface, which is the current signal the oscilloscope capture. Connect the signal to a waveform display device and measure the frequency of the signal. The measurement result is the same with the current capture rate.

#### Pass/Fail:

The instrument can output a pulse via this connector when a failed waveform is detected during the pass/fail test. The instrument continuously outputs a low level via this connector when no failed waveform is detected.

### 4.Signal Source 3 Output(EDU series)

Signal 3 output only the build-in 3-channel signal source. When source 3 is enabled, the signal can be output through signal terminal according to the current setting.

### 5.USB Device

You can connect the oscilloscope to a PictBridge printer or PC via this interface. When a PC is connected, users can send SCPI commands using the PC software or control the oscilloscope via user-defined programming. When a printer is connected, users can print the waveform displayed on the screen using the printer.

## **6.LAN/UART**

### **LAN**

Connect the instrument to network via this interface for remote control. This oscilloscope conforms to the LXI CORE 2011 DEVICE class instrument standards and can quickly build test system.

### **UART(optional interface)**

Connect the instrument to control system via this interface for remote control.

### **7.HDMI(optional interface)**

Connect the instrument to the screen with HDMI display via this interface for greater display.

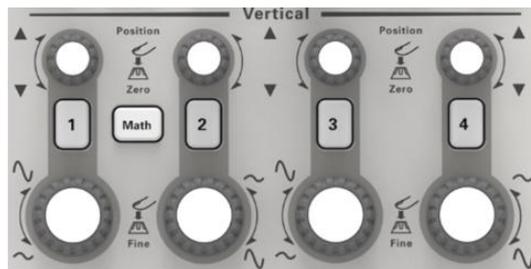
## 2.Front Panel Function Overview

### 2.1.Main keys introduction

#### 2.1.1.HORIZONTAL

	<p><b>Horizontal Offset :</b> Modify the horizontal position. The trigger point would move left or right relative to the center of the screen when you turn the knob. During the modification, waveform of all the channels would move left or right and the horizontal position message at the upper-right corner of the screen would change accordingly. Press down this knob to quickly reset the horizontal position.</p> <p><b>HORIZONTAL Time Base:</b> Modify the horizontal time base. Turn clockwise to reduce the time base and turn counterclockwise to increase the time base. During the modification, waveform of all the channels will be displayed in expanded or compressed mode and the time base message at the upper side of the screen would change accordingly. Press down this knob to quickly switch single-window or dual-window display modes.</p>
---	--

#### 2.1.2.VERTICAL



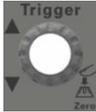
	<p>analog channel setting keys. Press any key to open the corresponding channel menu and press again to turn off the channel.</p>
	<p><b>Math:</b> Press this key to open the math operation menu, press again to close the math function</p>
	<p><b>VERTICAL Offset:</b> modify the vertical position of the current channel waveform. Turn clockwise to increase the position and turn counterclockwise to decrease. During the modification, the waveform would move up and down and the position message at the lower-left corner of the screen would change accordingly. Press down this knob to quickly reset the vertical position to zero.</p>
	<p><b>VERTICAL Voltage:</b> modify the vertical scale of the current channel. Turn clockwise to decrease the scale and turn counterclockwise to increase. During the modification, the display amplitude of the waveform would enlarge or reduce. The scale information at the lower side of the screen would change accordingly. Press down this knob to quickly switch the vertical scale adjustment mode between "Coarse" and "Fine".</p>

#### 2.1.3.Signal Source

	<p><b>Signal Source key:</b> Press this key to enter the source setting interface to set the output signal waveform and parameters</p>
---	--

**Note:** This function is only applicable to digital oscilloscopes with signal source function.

#### 2.1.4. TRIGGER

	<p><b>Trigger level:</b> modify the trigger level. Turn clockwise to increase the level and turn counterclockwise to reduce the level. During the modification, the trigger level line would move up and down and the value in the trigger level message box at the lower-left corner of the screen would change accordingly. Press down the knob to quickly reset the trigger level to the trigger data source zero point.</p>
	<p><b>Trig Menu:</b> press this key to open the trigger operation menu. This oscilloscope provides various trigger types. For more details, please refer to "<a href="#">Trigger System</a>".</p>
	<p><b>Force Trig:</b> Short press this key to generate a trigger signal forcefully. Long press this key to turn on history waveform.</p>

#### 2.1.5. AUTO

	<p><b>Auto Scale:</b> Press this key to enable the waveform auto setting function. The oscilloscope will automatically adjust the vertical scale, horizontal time base and trigger mode according to the input signal to realize optimum waveform display.</p>
---	--

#### 2.1.6. RUN/STOP

	<p><b>Run/Stop:</b> press this key to "RUN" or "STOP" waveform sampling. In the "RUN" state, the key is illuminated in green. In the "STOP" state, the key is illuminated in red.</p>
---	---

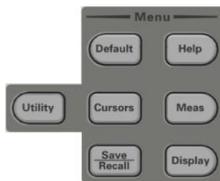
#### 2.1.7. SINGLE

	<p><b>Single:</b> Press this key to set the trigger mode to "Single". In single trigger mode, press FORCE to generate a trigger signal immediately.</p>
---	---

#### 2.1.8. Multifunctional Knob

	<p><b>V0:</b> Use this knob can do many settings, such as adjusting waveform brightness, trigger time, selecting trigger types, selecting trigger source, the waveform frequency offset and amplitude of signal source; In non-menu-operation mode, the indicator light of the knob doesn't go on, In menu operation, the indicator light of the knob goes on. Turn this knob can adjust the settings above, Turn clockwise to increase and counterclockwise to reduce, press down to select this option.</p>
---	---

#### 2.1.9. Function Menus

**Default:**

Short press this key to perform the corresponding settings. Long press to enter into Default function preset, you can select factory settings, default settings and user settings.

**Help:**

Press this key to turn on or off the online help system.

**Utility:**

press this key to enter the system function setting menu to set the system-related functions or parameters, such as the interface, sound and language. Besides, some advanced functions (such as the pass/fail test) are also supported.

**Cursor:**

press this key to enter the cursor measurement menu. The oscilloscope provides two cursor modes.

**Meas:**

press this key to open the measurement setting menu. You can set the measurement source, turn on or off the DMM, all measure, statistic function and etc. Press “all measure” to open the measurement of 42 waveform parameters. The measurement result will be displayed at the bottom of the screen.

**Save/Recall:**

press this key to enter the file save and recall interface. The types of the file that can be stored include setups, waves, parameters and CSV. Internal and external storage as well as disk management are also supported. Long press this key to save pictures to external storage device (USB disk), the picture format is .bmp.

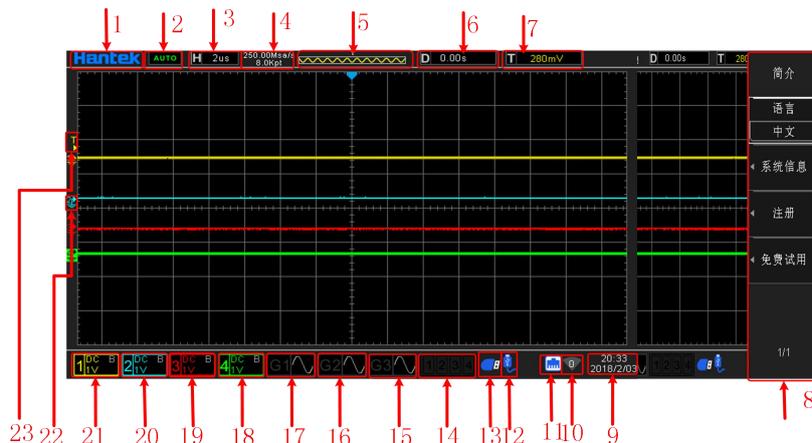
**Display:**

Short press this key to enter the display setting menu to set the display type, persistence time, wave intensity, screen grid and grid brightness.

Long press this key to clear persistence then collect or count again.

## 2.2. User Interface

This section will let you understand the front operating panel of this series of digital oscilloscopes before using them.



### 1. Hantek brand

Open in touch screen is to display QR code, scan to enter download help page quickly.

### 2. Trigger Status:

**Auto:** The oscilloscopes work in auto mode, will acquire waveform without trigger.

**READY:** All pre-trigger data has been acquired and the oscilloscope is ready to accept the trigger.

**TD:** The oscilloscope has been triggered.

**Roll:** In roll mode, the oscilloscopes will acquire and display waveform data continuously.

**Stop:** The oscilloscopes will stop acquiring waveform data.

**ARM:** Pre-trigger state.

### 3. The main time base of current window

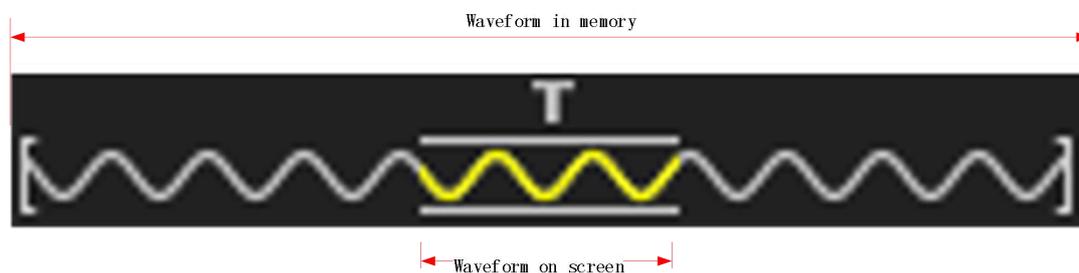
Represents the length of time represented by each grid on the horizontal axis of the screen.

### 4. Sampling rate, sampling point

Displays the current sampling rate and storage depth of the analog channel.

The sampling rate and storage depth vary with the horizontal base.

### 5. Waveform storage



### 6. Horizontal trigger time

Use the horizontal offset knob to adjust this parameter. Press the knob to quickly reset the horizontal displacement.

### 7. Trigger level

Use the trigger level knob to adjust this parameter. Press the knob to quickly reset the trigger level.

### 8. Operating menu to display the different information of each function keys

### 9. The time display of the oscilloscope

### 10. Wifi status display

### 11. Display the connection status of LAN

If the icon is lit or activated, the LAN is connected.

### 12. Display USB device status

If the icon is lit or activated, the USB guest is connected.

### 13. Display USB host status

If the icon is lit or activated, the USB host is connected.

### 14. Display LA status

If the icon is lit or activated, the LA channel is connected.

### 15. Display Gen3 status (some modes)

If the icon is lit or activated, the Gen3 channel is turned on.

### 16. Display Gen2 status (some modes)

If the icon is lit or activated, the Gen2 channel is turned on.

**17. Display Gen1 status (some modes)**

If the icon is lit or activated, the Gen1 channel is turned on.

**18. Display CH4 status**

If the icon is lit or activated, the channel 4 is turned on.

**19. Display CH3 status**

If the icon is lit or activated, the channel 3 is turned on.

**20. Display CH2 status**

If the icon is lit or activated, the channel 2 is turned on.

**21. Display CH1 status**

If the icon is lit or activated, the channel 1 is turned on.

**22. Display the zero level position of the channel**

**23. Display the trigger level position of the channel**

## 3.To Set the Sample System

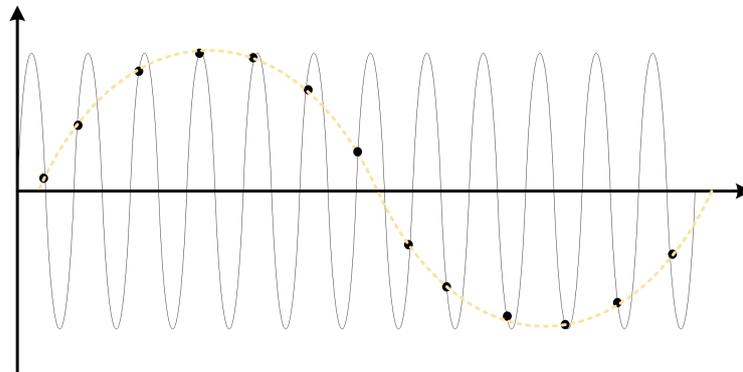
To understand the waveform acquisition system of oscilloscope, it is necessary to understand the sampling principle first and find out the relationship between bandwidth, sampling rate and storage depth.

### 3.1.Sampling principle

According to Nyquist sampling principle, in order to restore the analog signal without distortion, the sampling frequency should be more than twice of the highest frequency in the analog signal spectrum( $F_s > 2F_{max}$ )The higher the sampling rate, the closer the waveform recovered later is to the original signal, but the system requirements are higher, the conversion circuit must have a faster conversion speed.

#### 3.1.1.Waveform aliasing

If sample can't meet ( $F_s > 2F_{max}$ ), The waveform frequency is lower than the actual signal frequency when the sampled data is reconstructed.The most common aliasing is jitter on fast edges.

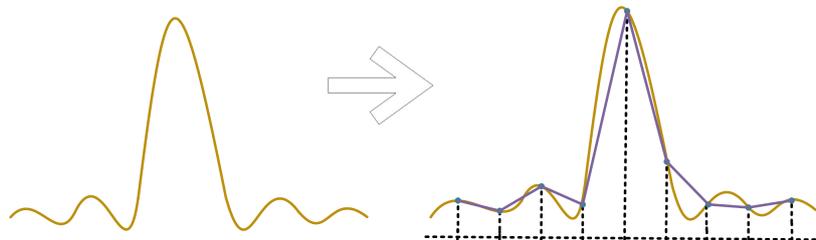


The following two measures can avoid aliasing:

- 1.Increase the sampling frequency to more than twice the maximum signal frequency;
- 2.Introducing a low pass filter or raising its parameters. The low - pass filter is usually called anti - aliasing filter, Anti-aliasing filter can limit the bandwidth of the signal and make it satisfy the condition of sampling theorem. This is possible in theory, but impossible in practice. Because the filter cannot completely filter out signals above the Nyquist frequency, so,there is always some "small" energy beyond the bandwidth required by the sampling theorem. However anti-aliasing filters can make these energies small enough to be negligible.

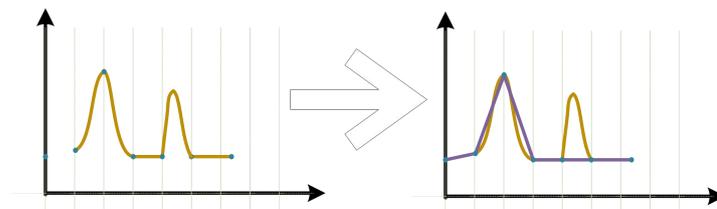
#### 3.1.2.Waveform distortion

Due to the lack of some waveform details caused by too low sampling rate, the waveform displayed by oscilloscope sampling is quite different from the actual waveform.



#### 3.1.3.Waveform loss

Because the sampling rate is too low, the waveform during the reconstruction of the sampled data does not reflect all the actual signals.



## 3.2. Sample rate and memory depth

### 3.2.1. Sample rate

The highest sample rate of DPO6000/MPO6000 is 1GSa/s. In the actual use of oscilloscopes, the sample rate is determined by the current horizontal timebase gear and memory depth. The sample rate can be changed by adjusting the horizontal timebase through the horizontal gear knob, or by changing the memory depth, and the sample rate value can be changed in real time and displayed in the status bar in the upper left corner of the screen.

### 3.2.2. Memory depth

Memory depth refers to the number of waveform points that the oscilloscope can store in a single trigger sample and it reflects the storage ability of the sample memory. DPO6000/MPO6000 series oscilloscope provides up to 24 Mpts memory depth. The relation of memory depth, sample rate and sample time of the oscilloscope fulfills the equation below:

$$\text{Sampling time} = \text{storage depth} / \text{sampling rate (Sa/s)}$$

For example, sample rate is 1GSa/s, memory depth is 32K, the actual sample time is:

$$\text{Sampling time} = \frac{32000}{1,000,000,000} \text{ s} = 32 \times 10^{-6} \text{ s} = 32 \mu\text{s}$$

## 3.3. Waveform interpolation, methods

Under real-time sampling, the oscilloscope obtains the discrete sample values of the displayed waveform. In general, the waveform shown by dots is difficult to observe. In order to increase the visibility of signal, digital oscilloscope generally adopts interpolation display mode. Interpolation is a method of "connecting all points" and using some points to calculate the whole waveform. For real-time sample using interpolation method, even if the oscilloscope only collects a few sampling points in one way, it can also use interpolation method to fill the gap between points and reconstruct the accurate waveform. Interpolation method is divided into sine interpolation, linear interpolation, step interpolation.

### 3.3.1. Linear interpolation:

Directly connect the line at the adjacent sample point. This approach is limited to the reconstruction of edge-only signals, such as square waves. Sine interpolation method uses curves to connect sampling points, which is more universal.



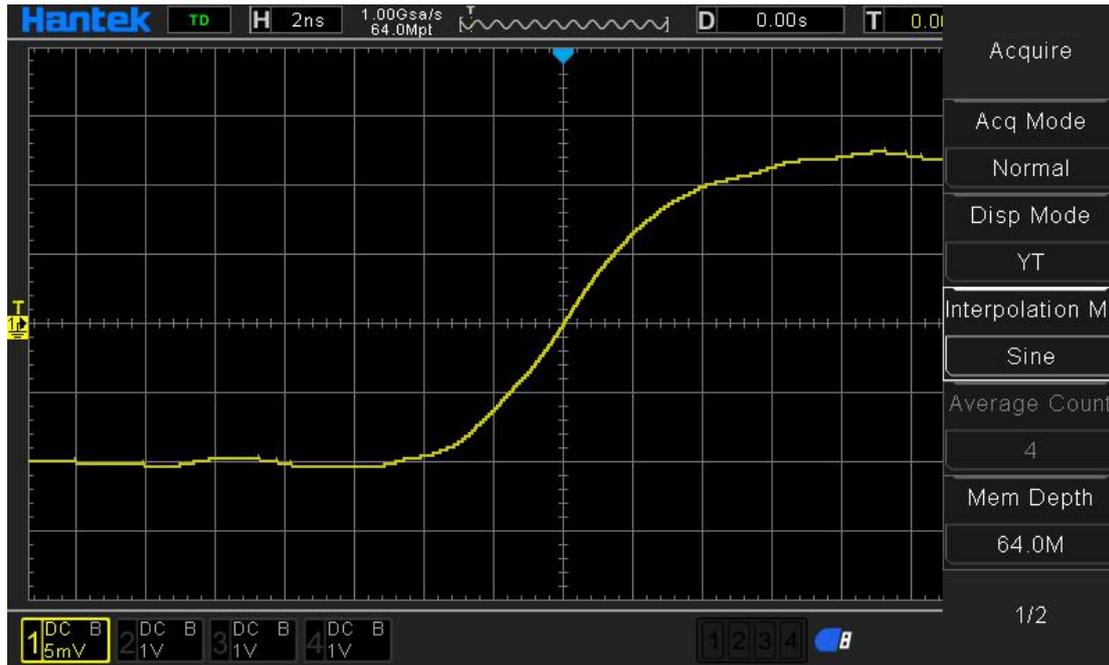
### 3.3.2.Step interpolation:

Insert the previous sample point value between adjacent samples. This interpolation algorithm is the same as the normal actual sampling of oscilloscope ADC, and adopts the sampling and holding method to interpolate data, so as to improve the signal sampling rate. For example, interpolation of digital waveform.



### 3.3.3.Sine interpolation:

By means of mathematical processing, the results are calculated in the actual sample interval. This method bends the signal waveform to produce a more realistic general shape than the pure square wave or pulse. When the sampling rate is 3 to 5 times the system bandwidth. The sine interpolation method is recommended. The following figure shows the distinct display effect after adopting the two interpolation methods.



Through the comparison of the acquisition signal waveform by above three interpolation methods, it can be clearly seen, These three interpolation methods, sine interpolation method is relatively smooth fitting signal waveform, and relatively truly reflecting the signal waveform, therefore, when we do signal interpolation, we use sine interpolation by default. Compared with sine interpolation, linear and stepped interpolation methods have the advantage of small computation, which enables quick interpolation fitting of waveform and requires less precision. When the actual sampling rate of signal exceeds the natural sampling rate of ADC during the use of oscilloscope, the interpolation mode will be started. The three interpolation modes of DPO6000 and MPO600 oscilloscopes will improve the waveform sampling rate [digital waveform only use step interpolation] and restore waveform truly and accurately. For these three interpolation modes, we will recommend the sine interpolation mode. Through the sine interpolation algorithm, the curve fitting waveform can accurately and truly restore the waveform.

### 3.4. Operational control

Two keys "Run/Stop" and "Single" on the front panel of the oscilloscope can be used to start or stop the sampling system of the oscilloscope.

When the Run/Stop key is displayed in green, it means that the oscilloscope is running, that is, the oscilloscope is collecting data. To Stop data collection, press the Run/Stop key to display red, indicating that data collection has stopped. To capture and display a Single acquisition (whether the oscilloscope is running or stopped), please press Single. When the input single signal meets the trigger condition, the oscilloscope captures, stores and displays the waveform. At this point even if there is a signal input oscilloscope will not be processed. A single reset is required for recapture.

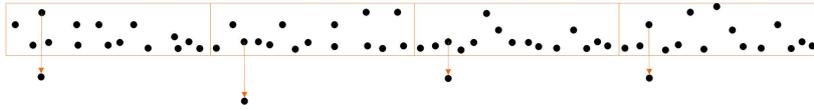
When Single is pressed, the trigger mode is temporarily set to "normal" (to prevent the oscilloscope from automatically triggering immediately). The key light is orange and the oscilloscope is waiting for the trigger condition before displaying the waveform. When the oscilloscope is triggered, it will display single acquisition and stop running, and the Run/Stop key will show red. Press Single again to capture another waveform.

### 3.5. Acquisition mode

Press Utility → Acquire → Acquire Mode on the frontal panel and use the knob to select the desired acquisition mode (the default is normal), then press down the knob to select this mode. You can also press **Acq Mode** continuously to switch the acquisition mode.

#### 3.5.1. Normal

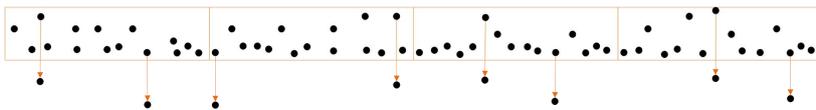
In this mode, the oscilloscope samples the signal at equal time interval to rebuild the waveform. For most of the waveform, the best display effect can be obtained using this mode.



### 3.5.2. Peak

In this mode, the oscilloscope acquires the maximum and minimum values of the signal within the sample interval to get the envelope of the signal or the narrow pulse of the signal that might be lost. In this mode, signal confusion can be prevented but the noise displayed would be larger.

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



### 3.5.3. Average

In this mode, the oscilloscope averages the waveform from multiple samples to reduce the random noise of the input signal and improve the vertical resolution. Greater number of averages can lower the noise and increase the vertical resolution; while at the same time, it will slow the response of the displayed waveform to the waveform changes.

DPO6000/MPO600 series oscilloscope adopts the method of taking the stable average of the first n collection and then taking the exponential average method. These two algorithms can always show the trend of changing waveform. The stable averaging algorithm is as follows:

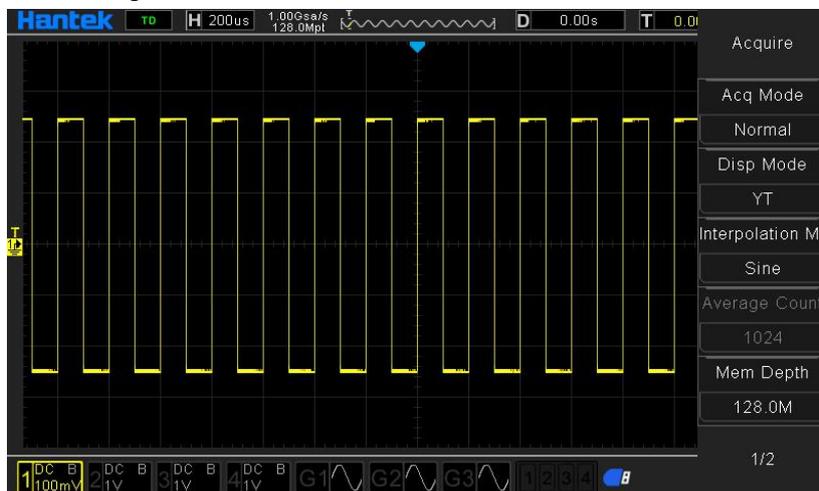
$$Ave_n = \frac{1}{n} \sum_{i=1}^n X_i = \frac{1}{n} \sum_{i=1}^{n-1} X_i + \frac{1}{n} X_n$$

$$Ave_n = \frac{n-1}{n} Ave_{n-1} + \frac{1}{n} X_n$$

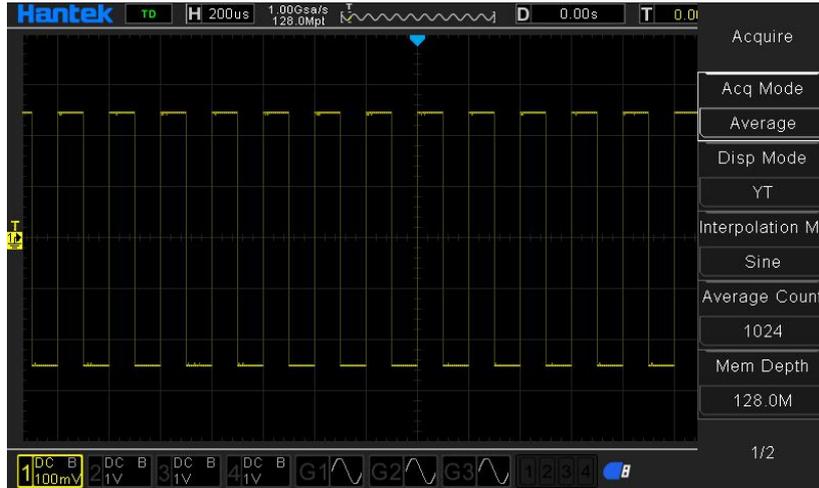
$Ave_n$  is the average sample value,  $X_i$  is the  $i$ -th sample value, and  $n$  is the current average number.

When "Average" mode is selected, press **"average time"** menu to set the desired number of averages. The number of averages can be set to 2,4,8,16,32,64,128,256,512 or 1024. The default is 2.

Examples of unaveraged waveform:



Example of waveform after 1024 average times:

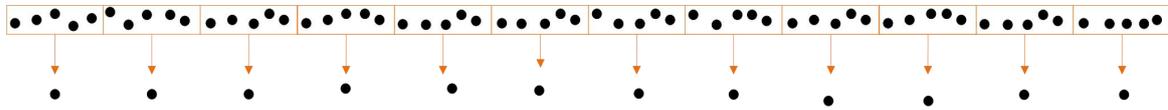


### 3.5.4.High resolution

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

In fact, the high-resolution mode is also a low-pass filtering mode, so the bandwidth of the measured signal is limited, that is, the bandwidth of the measured signal is sacrificed to improve the measurement accuracy, the number of significant digits increases as the number of adjacent points used in the calculation increases, DPO6000/MPO6000 series oscilloscope enhances to up to 12 bits.

In the high resolution mode, the oscilloscope always runs in the mode of the highest sampling rate, and the enhancement digits change with the time base of the oscilloscope.



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

### 3.6.Sample rate

The sample rate is the number of times an oscilloscope samples a signal in unit time(Sa/s), the single-channel maximum real-time sample rate of DPO6000/MPO6000 series oscilloscope is 1GSa/s. In addition, it will automatically switch according to the current acquisition mode, memory depth and time base setting, and automatically calculate the appropriate sampling rate without manual setting by users.

Function selection operation

Switch memory depth,press Utility→ acquisition→memory depth on the frontal panel.

Select the horizontal timebase knob,rotate the horizontal timebase knob on the frontal panel.

Change the acquisition mode, press Utility→ acquisition→acquisition mode on the frontal panel.

**Note:** The sample rate of the current waveform is displayed in the status bar at the upper side of the screen.

## 4.Horizontal System

### 4.1.Time Base Mode

Press **utility-acquisition-display mode menu** at the front panel,F2 display mode control menu and users can select the time base mode they need.

DPO6000/MPO6000 series oscilloscopes provide 3 kinds of time base modes:YT mode,XY mode,roll mode(The default is YT mode).

#### 4.1.1.YT mode

In this mode, the Y axis represents voltage and the X axis represents time.

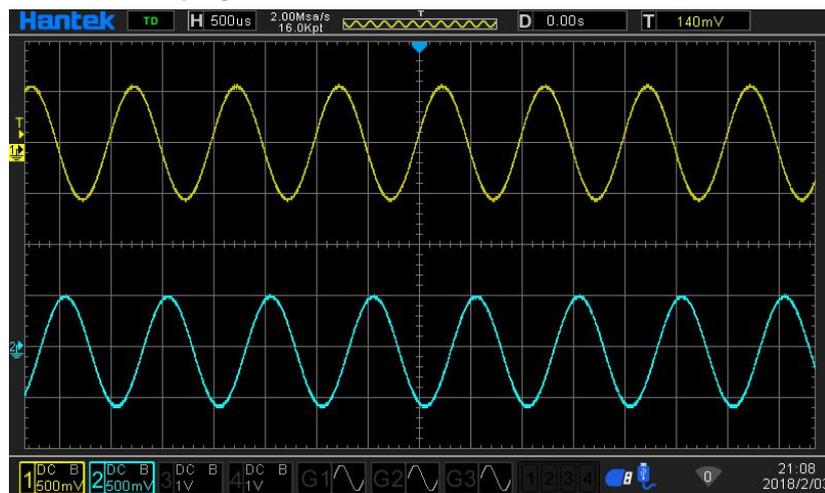
**Note:** In this mode, when the horizontal time base is greater than or equal to 100 ms, the instrument enters sweep mode.

##### **Sweep mode:**

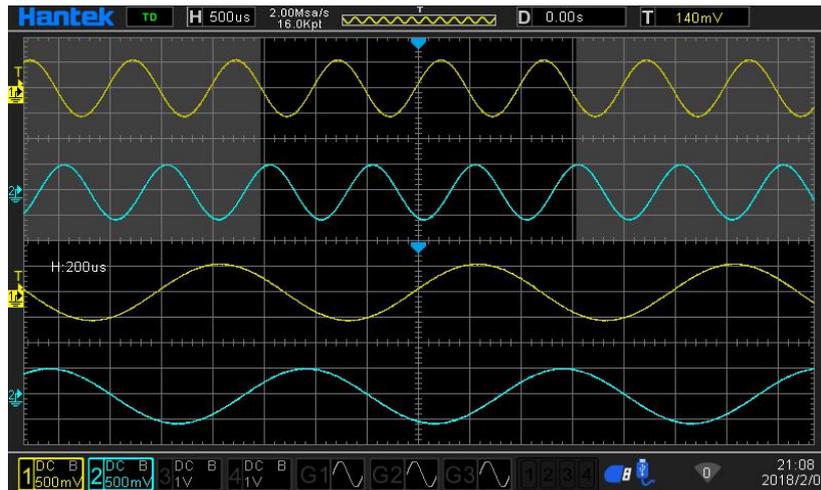
In YT mode, when the horizontal time base is set to 100 ms/div or slower, the instrument enters "sweep" mode in which the instrument first acquires the data at the left of the trigger point and then waits for a trigger event. After the trigger occurs, the instrument continues to finish the waveform at the right of the trigger point and display the current acquired waveform data.

**Note:**When slow sweep mode is used to observe low frequency signal, DC "Channel Coupling" mode is recommended.

#### 1. YT single-window display mode

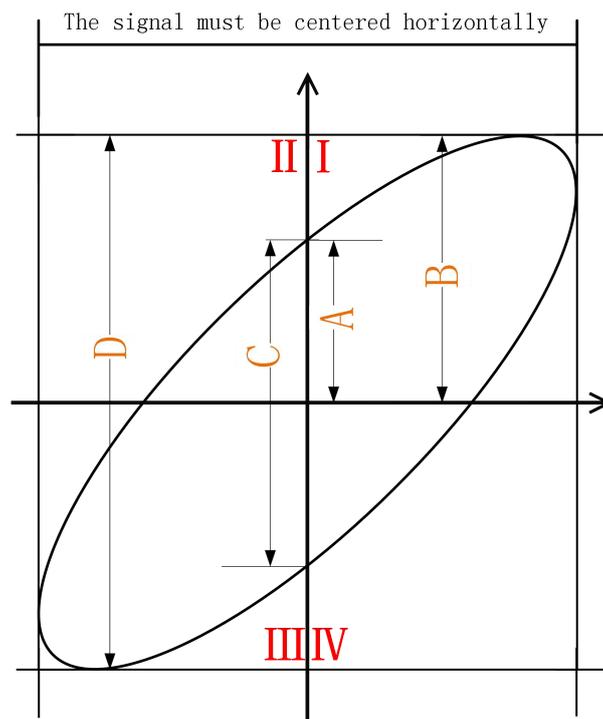


#### 2.YT dual-window display mode



#### 4.1.2.XY Mode

In this mode, the oscilloscope changes the two channels from voltage-time display mode to voltage-voltage display mode, where the X-axis and Y-axis combination of XY display mode can be CH1-CH2 or CH3-CH4. The phase deviation between two signals with the same frequency can be easily measured via Lissajous method. The figure below shows the measurement schematic diagram of the phase deviation.



Measurement Schematic Diagram of Phase Deviation

According to  $\sin\theta=A/B$  or  $C/D$  (wherein,  $\theta$  is the phase deviation angle between the two channels and the definitions of A, B, C and D are as shown in the figure above), the phase deviation angle is obtained, that is:

$$\theta=\pm \arcsin \frac{A}{B} \text{ or } \pm \arcsin \frac{C}{D}$$

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

deviation angle obtained should be within quadrant II and III, namely within  $(\pi/2 \text{ to } \pi)$  or  $(\pi \text{ to } 3\pi/2)$ . The XY function can be used to measure the phase deviation occurred when the signal under test passes through a circuit network. Connect the oscilloscope to the circuit to monitor the input and output signals of the circuit.

**Note:**

When XY mode is enabled, "**Sweep**" will be disabled automatically.

In XY mode, the oscilloscopes will select channels group automatically, CH1-CH2 or CH3-CH4.

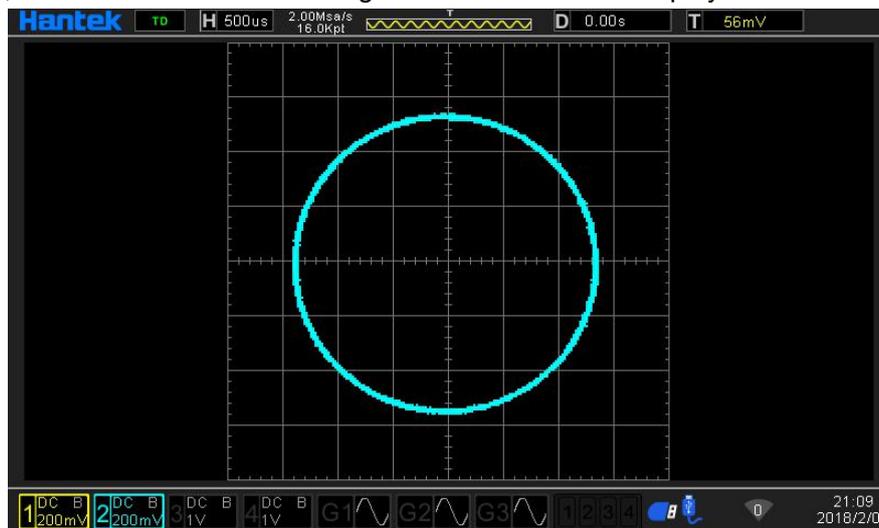
The following functions are not available in XY mode:

"Delayed Sweep", "Vectors", "Protocol Decoding", "Acquisition Mode", "Pass/Fail Test", "Digital Channel" and "To Set the Persistence Time" all don't work.

#### 4.1.2.1. XY single-window display mode

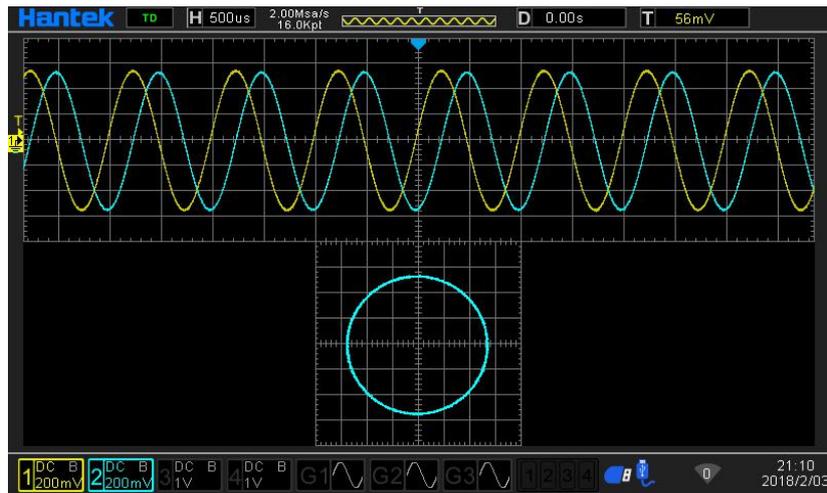
**Application example:** measure the phase deviation of the input signals of two channels.

1. Connect a sine signal to CH1 and then connect a sine signal with the same frequency and amplitude but a  $90^\circ$  phase deviation to CH2.
2. Press **Auto Scale** and then adjust the vertical positions of CH1 and CH2 to 0 V.
3. Set the time base mode to XY. Rotate **Horizontal SCALE** to adjust the sample rate properly to get better Lissajous figure for better observation and measurement.
4. Rotate **VERTICAL SCALE** of CH1 and CH2 to make the signals easy to observe. At this point, the circle as shown in the figure below should be displayed.



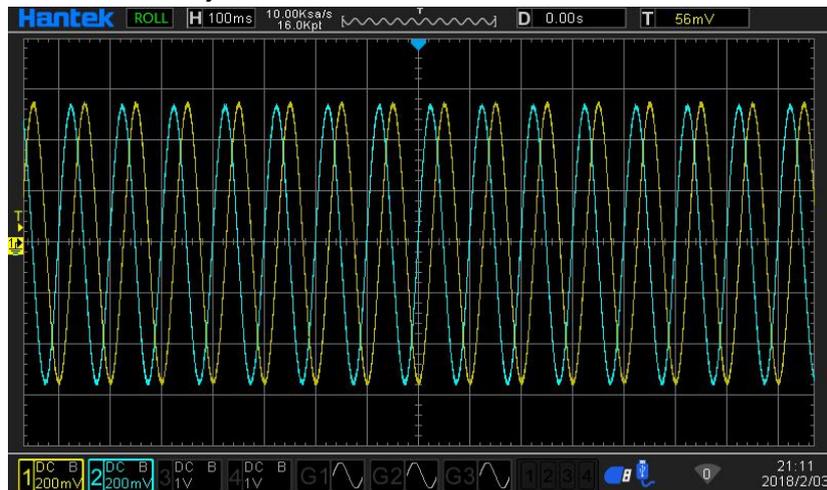
5. Observe the measurement result shown in the figure above. According to the measurement schematic diagram of the phase deviation,  $A/B (C/D) = 1$ . Thus, the phase deviation angle  $\theta = \pm \arcsin 1 = 90^\circ$ .

#### 4.1.2.2.XY dual-window display mode



#### 4.1.3.ROLL mode

In this mode, the waveform scrolls from right to left to update the display. The range of horizontal scale adjustment is from 100ms to 100s.



**Note:** When Roll mode is enabled, the waveform "horizontal position", "Dual-window mode", "Protocol Decoding", "Pass/Fail Test", "Segment acquisition", "To Set the Persistence Time", "To Trigger the Oscilloscope" all don't work.

## 5.To Set the Vertical System

### 5.1.To Enable the Analog Channel

DPO6000/MPO6000 series provide 4 analog input channels (CH1-CH4). As the setting methods of the vertical systems of the four channels are the same, this chapter takes CH1 as an example to illustrate the setting method of the vertical system. Connect a signal to the channel connector of CH1 and then press **CH1** at the front panel. The indicator light of CH1 turns on, it indicates CH1 is turned on. Press CH1 again, the indicator light of CH1 turns off, it indicates CH1 is turned off.

### 5.2.Channel Coupling

The undesired signals can be filtered out by setting the coupling mode. For example, the signal under test is a square waveform with DC offset.

When the coupling mode is "DC": the DC and AC components of the signal under test can both pass.

When the coupling mode is "AC": the DC components of the signal under test are blocked.

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

Press **CH1** → **Coupling**, select the desired coupling mode (the default is DC). The current coupling mode is displayed in the channel status label at the bottom of the screen as shown in the figure below. You can also press F1 **【V0】** continuously to switch the coupling mode.



Channel coupling mode is AC



Channel coupling mode is DC



Channel coupling mode is GND

### 5.3.Bandwidth Limit

DPO6000/MPO6000 series oscilloscopes support the bandwidth limit function which can reduce the display noise. For example, the signal under test is a pulse with high frequency oscillation. When the bandwidth limit is disabled, the high frequency components of the signal under test can pass the channel. When limiting the bandwidth to 20 MHz, the high frequency components of the signal under test that exceed 20 MHz are attenuated.

Press CH1, then press F2 continuously to switch the bandwidth limit state (the default is OFF, the character "B" will be gray). When the bandwidth limit is enabled, the character "B" will be bright in the channel status label at the bottom of the screen.

**Note:** Bandwidth limit can reduce the noise as well as attenuate or eliminate the high frequency components of the signal.

Display icon description:



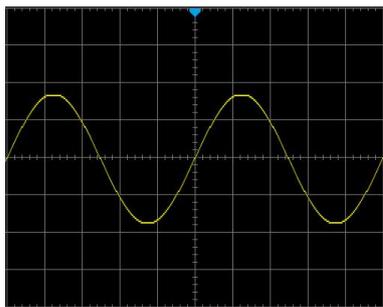
channel is full bandwidth



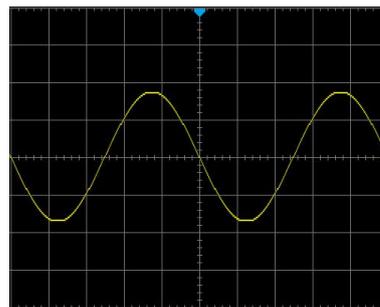
channel enables 20M bandwidth limit

### 5.4.Waveform Invert

Press **CH1** → **Invert** to turn on or off waveform invert. When waveform invert is turned off, the waveform display is normal; when waveform invert is turned on, the waveform voltage values are inverted (as shown in the figures below).



waveform invert is turned off



waveform invert is turned on

### 5.5.Probe Ratio

DPO6000/MPO6000 series allow user to set the probe attenuation ratio manually. Press **CH1** → **Probe** and use to select the desired probe ratio. The probe ratio values available are as shown in the table below.

Menu	Attenuation Ratio(display amplitude of the signal : Actual amplitude of the signal)
0.01X	0.01:1
0.02X	0.02:1
0.05X	0.05:1
0.1X	0.1:1
0.2X	0.2:1
0.5X	0.5:1
1X	1:1
2X	2:1
5X	5:1
10X	10:1
20X	20:1
50X	50:1
100X	100:1
200X	200:1
500X	500:1
1000X	1000:1
2000X	2000:1
5000X	5000:1
10000X	10000:1

### 5.6.Vertical Scale

Vertical scale refers to the voltage value per grid in the vertical direction on the screen and is usually expressed as V/div.Press **CH1** and rotate vertical **SCALE** to adjust the vertical scale. The size of the displayed waveform will be changed accordingly, clockwise to reduce the scale and counterclockwise to increase. The scale information in the channel label at the bottom of the screen will change accordingly during the adjustment. The adjustable range of the vertical scale is related to the probe ratio currently set. By default, the probe ratio is X1, and the adjustable range of the vertical scale is from 500 uV/div to 10 V/div. The vertical scale can be adjusted in "Coarse" or "Fine" mode. Press **CH1** → **scope scale** to switch the adjustment mode.

**Coarse adjustment (take counterclockwise as an example):** set the vertical scale in 1-2-5 step namely 500uV/div、1mV/div、2mV/div、5mV/div、10mV/div、20mV/div、50mV/div、

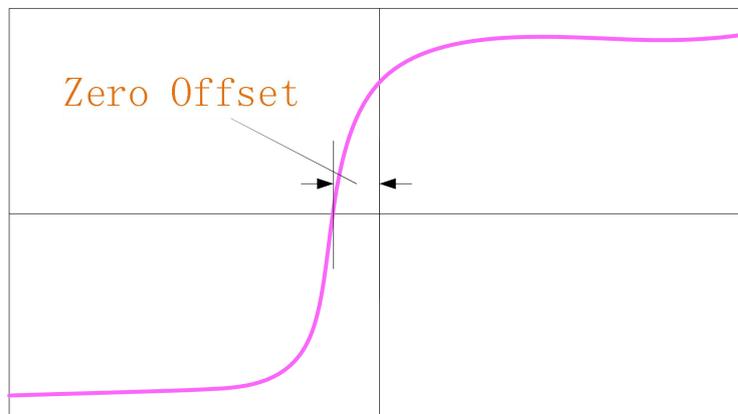
100mV/div.....10V/div.

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

**Note:** You can not only select the "Coarse" and "Fine" adjustments by scope scale menu also press **VERTICAL SCALE** to quickly switch between "Coarse" and "Fine" adjustments.

### 5.7.Delay Calibration of the Analog Channel

When using an oscilloscope for actual measurement, the transmission delay of the probe cable may bring relatively greater error (zero offset). DPO6000/MPO6000 series oscilloscopes support users to set a delay time to calibrate the zero offset of the corresponding channel. Zero offset is defined as the offset of the crossing point of the waveform and trigger level line relative to the trigger position, as shown in the figure below.



**Note:** The value of this parameter only supports analog channel settings, not supports digital channel settings.

Delay calibration, use V0 knob to set the delay calibration time you need. The range of this parameter is -100ns to 100ns. Press the knob can revert the delay time to 0.00s.

## 6.Trigger System

Trigger can be understood as an event (or action);For example, when taking a photo, the action of pressing the shutter is the trigger, which is an event that starts the camera to record the image. For oscilloscopes, when the trigger condition is established, the waveform (sampled data) will be recorded and displayed on the screen. For digital oscilloscope, it samples waveform continuously no matter whether it is stably triggered, but only stable trigger can ensure stable display. The trigger module ensures that every time base sweep or acquisition starts from the user-defined trigger condition, namely every sweep is synchronous to the acquisition and the waveform acquired overlap to display stable waveform.Trigger setting should be based on the features of the input signal, thus you need to have some knowledge of the signal under test to quickly capture the desired waveform. This oscilloscope provides abundant trigger types,which can help you to focus on the desired waveform details.

The trigger determines when the oscilloscope begins to acquire data and display waveform.Once the trigger is set up correctly, an oscilloscope can convert an unstable display or blank screen into a meaningful waveform.Here are some basic concepts of the trigger.

### 6.1.Trigger Source

Press signal source in the trigger control area of the front panel(Trig Menu) to select the desired trigger source. Analog channels CH1-CH4, digital channels D1.0-D1.3,D2.0-D2.3, D3.0-D3.3,D4.0-D4.3 can all be used as trigger source.

#### Analog channel input:

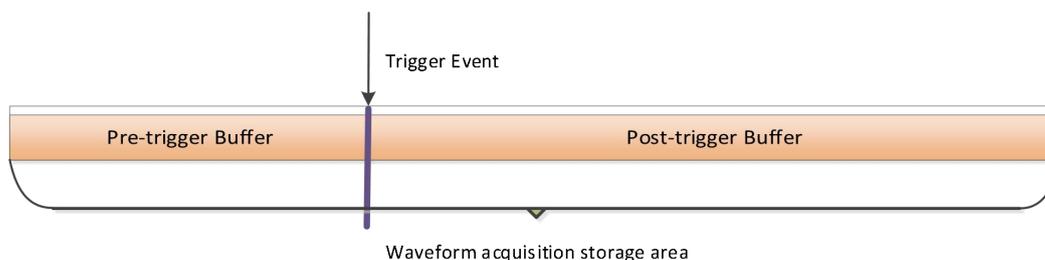
Signals input from analog channels CH1-CH4 can all be used as trigger source. If the trigger source channel selected isn't enabled,this trigger is regarded as invalid trigger.

#### Digital channel input:

Only the digital channels connected to oscilloscopes can be used as trigger source(some trigger types can't be selected,such as slope trigger).

### 6.2.oscilloscope acquisition process

The following is the schematic diagram of the acquisition memory. To easily understand the trigger event, the acquisition memory is divided into the pre-trigger buffer and post-trigger buffer.



the oscilloscope operates by first filling the pre-trigger buffer.It starts searching for a trigger after the pre-trigger buffer is filled. While searching for the trigger, the data sampled will still be transmitted to the pre-trigger buffer (the new data will continuously overwrite the previous date).After finding the trigger, the oscilloscope is collected and stored in the post-trigger storage area.

### 6.3.Trigger Mode

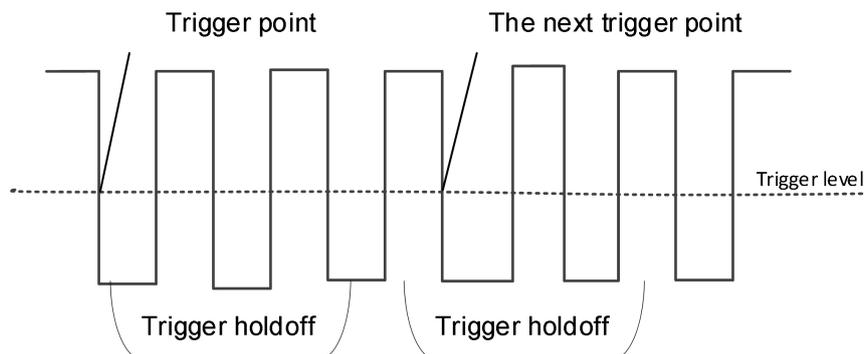
**Auto Mode** Performs the acquisition freely in absence of valid trigger. It allows the generation of untriggered waveform with the time base set to 100ms/div or slower.When an oscilloscope detects the valid trigger condition,it will complete a valid trigger acquisition. When an oscilloscope doesn't detect the valid trigger condition,it will complete a acquisition without trigger.

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

**Single trigger** The oscilloscope only trigger when the specified trigger condition is found and then stops. To perform a **single trigger**,you can press **【Single】** .

#### 6.4.Trigger Holdoff

Press [TrigMenu] to turn on trigger contents,press holdoff softkey.Press holdoff softkey. Trigger holdoff can be used to stably generate complex waveform (such as modulated waveform). Holdoff time is the amount of time that the oscilloscope waits for re-testing another trigger after testing a trigger. The oscilloscope will not trigger during the holdoff time.For a pulse train,you can adjust the holdoff time to make the oscilloscope trigger only at the first pulse in the train.



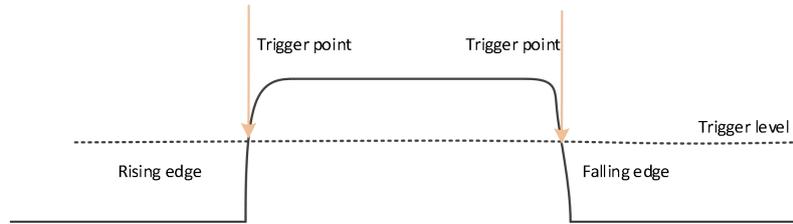
#### 6.5.Trigger Type

DPO6000/MPO6000 series oscilloscopes provide up to 16 kinds of trigger function as the followings:

- (1)Edge Trigger
- (2)Pulse Trigger
- (3)Video Trigger
- (4)Slope Trigger
- (5)Overtime Trigger
- (6)Window Trigger
- (7)Pattern Trigger
- (8)Interval Trigger
- (9)Under Amp Trigger
- (10)Delay Trigger
- (11)Setup/Hold Trigger
- (12)UART Trigger(optional)
- (13)LIN Trigger(optional)
- (14)CAN Trigger(optional)
- (15)SPI Trigger(optional)
- (16)IIC Trigger(optional)

##### 6.5.1.Edge Trigger

Edge Trigger identifies the trigger by looking for specified edges(ascending edge, descending edge,ascending or descending edge) and voltage levels on the waveform.



Press **【Trig Menu】** on the front panel to turn on the trigger function menu.

**【type】** select Edge and press V0 to confirm.

**【Data source】** select **CH1~CH4** or **LA** to be trigger source

**Note:**When use LA as trigger source,LA must be inserted.

**【Slope】** select desired trigger edges(rising edge、 falling edge、 double edge), press V0 to confirm.

**【Mode】** select acquisition mode(auto、 normal),press V0 to confirm.

**【50%】** set trigger level as the vertical central point of the peak-peak value of the trigger signal.Trigger level value is displayed on the top right corner.

**【holdoff】** set holdoff time.

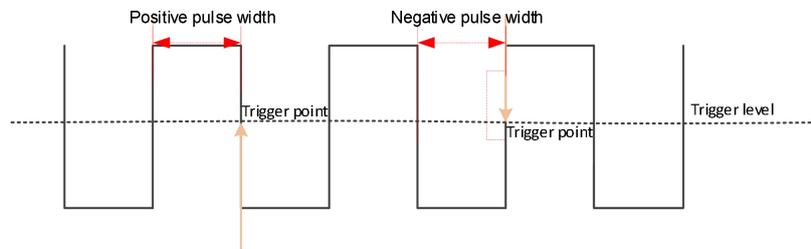
**Trigger level knob:**analog channels can modify trigger level value.Trigger symbol moves up and down according to the rotation of the knob.Rotate the trigger level knob to acquire the stable trigger.

**Digital channel:**can change trigger threshold of the digital channel by setting the threshold voltage.

**Note:**Press Auto Scale,set the trigger mode to be Edge trigger,trigger slope to be rising edge.

### 6.5.2.Pulse Trigger

Pulse Trigger sets the oscilloscope to trigger on the positive or negative pulse with a specified width.You can set trigger source,polarity(positive pulse、 negative pulse),limit conditions and pulse width in this menu.



Press **【Trig Menu】** on the front panel to enter trigger function menu.

**【Type】** select pulse and press V0 to confirm.

**【Data source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:**When use LA as trigger source,LA must be inserted.

**【Polarity】** select the positive polarity or negative polarity of the trigger.

**【When】** select the trigger conditions(>、 <、 =、 !=),and press V0 to confirm.

<(less than the time value):only trigger when the positive pulse width or negative pulse width of the input signal is less than the setting pulse width **【pulse error is 5%】** .

>(greater than the time value):only trigger when the positive pulse width or negative pulse width of the input signal is greater than the setting pulse width **【pulse error is 5%】** .

!=(unequal to the time value):only trigger when the positive pulse width or negative pulse width of the input signal is unequal to the setting pulse width **【pulse error is 5%】** .

= (equal to the time value):only trigger when the positive pulse width or negative pulse width of the input signal is equal to the setting pulse width **【pulse error is 5%】** .

**【width】** set pulse width time **【8ns~10s】** .

【50%】 set the trigger level as the vertical central point of the peak-peak value of the trigger signal. Trigger level value is displayed on the top right corner.

【Mode】 select acquisition mode(auto、 normal),press V0 to confirm.

【Holdoff】 set holdoff time.

**Trigger level knob:** analog channel can modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob. Rotate the trigger level knob to acquire the stable trigger.

**Digital channel:** can change trigger threshold of the digital channel by setting the threshold voltage.

### 6.5.3. Video Trigger

You can use video trigger to acquire most complex waveform of standard analog video signals and Hd video signals. The trigger circuit can detect the vertical and horizontal spacing of the waveform and generate a trigger based on the selected video trigger setting. This series oscilloscopes support NTSC(National Television Standards Committee) and PAL.

Press 【Trig Menu】 on the front panel to enter trigger function menu.

【Type】 select video, and press V0 to confirm.

【Data source】 select CH1~CH4 to be trigger source.

【Standard】 select the desired video standard. This series oscilloscopes support: NTSC、 PAL、 HDTV720p、 HDTV1080p and HDTV1080i.

【Synchronization】 select the video trigger mode(scan line、 video line、 odd fields、 even fields、 all fields).

Scan line: trigger on the first line found.

Video line: for NTSC and PAL/SECAM video standards, trigger on the specified line in the odd or even field.

**Note:** When this sync trigger mode is selected, you can modify the line number in **Line** menu with a step of 1. The range of the line number is from 1 to 525 (NTSC), 1 to 625 (PAL), 1 to 750(720P) or 1 to 625 (576P).

Odd fields: trigger on the rising edge of the first ramp pulse in the odd field.

Even fields: trigger on the rising edge of the first ramp pulse in the even field.

All fields: trigger on the rising edge of the first ramp pulse in the odd/even field.

【Line number】 rotate V0(PAL 1~625, NTSC 1~525)、 1 to 750(720P)、 1 to 1125 (1080P/1080i).

【polarity】 select the desired video polarity (positive polarity, negative polarity).

【Mode】 select the acquisition mode(auto、 normal), press V0 to confirm.

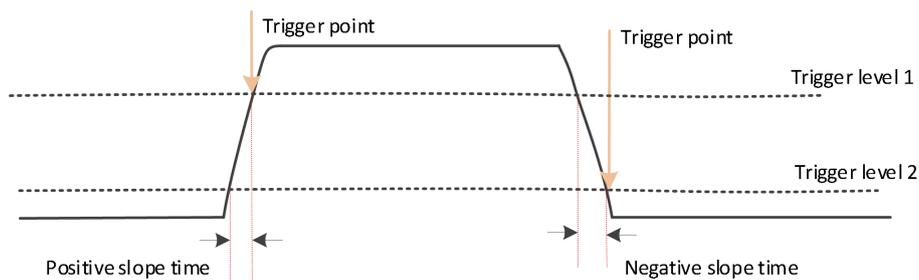
【Holdoff】 set the holdoff time.

**Trigger level knob:** modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob. Rotate the trigger level knob to acquire the stable trigger.

### 6.5.4. Slope Trigger

In slope trigger, the oscilloscope triggers on the positive or negative slope from one level to another level in the specified time.

As shown in the figure below, positive(negative) slope time is defined as the time difference between the two crossing points of trigger level line with the rising(falling) edge.



Press **【Trig Menu】** on the frontal panel to turn on trigger function menu.

**【Type】** select **slope** and press V0 to confirm.

**【Data source】** select **CH1~CH4** to be trigger source.

**【Slope】** select the desired trigger edge(rising edge or falling edge )

**【Level】** high/low level setting function,press this softkey again to select low level(V2)or high level(V1),then rotate trigger level knob to adjust the vertical position of high(low) level to obtain the desired slope time T.The corresponding displacement information changes in real time and is displayed in the status bar in the upper right corner of the screen.

**【When】** select the desired slope condition and press V0 to confirm.

The setting conditions are the same with pulse .

**【Width】** set the pulse width time **【8ns~10s】** .

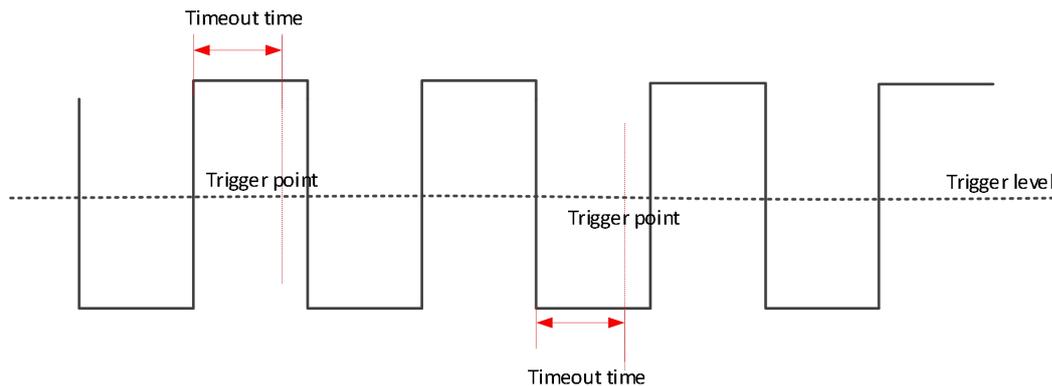
**【Mode】** select the acquisition mode(auto、 normal),press V0 to confirm.

**【Holdoff】** set the holdoff time.

**Trigger level knob:**modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob.Rotate the trigger level knob to acquire the stable trigger.

### 6.5.5.Overtime Trigger

In timeout trigger, the instrument triggers when the time interval ( $\Delta T$ ) from when the rising edge (or falling edge) of the input signal passes through the trigger level to when the neighbouring falling edge (or rising edge) passes through the trigger level is greater than the timeout time set,as shown in the figure below.



Press **【Trig Menu】** on the frontal panel to turn on trigger function menu.

**【Type】** select **Overtime**,and press V0 to confirm.

**【Data source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:**when use LA to be trigger source,LA must be inserted.

**【Polarity】** select the positive polarity or negative polarity of trigger.

**【Time】** set pulse width time **【8ns~10s】** .

**【50%】** set the trigger level as the vertical central point of the peak-peak value of the trigger signal.Trigger level value is displayed on the top right corner.

**【Mode】** select the acquisition mode(auto、 normal),press V0 to confirm.

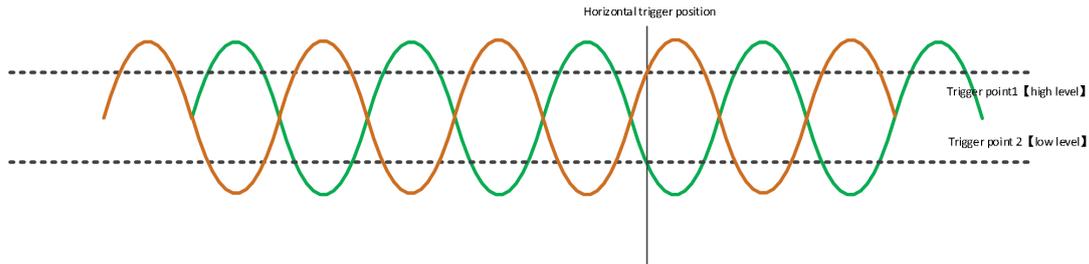
**【Holdoff】** set the holdoff time.

**Trigger level knob:**analog channel can modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob.Rotate the trigger level knob to acquire the stable trigger.

**Digital channel:**can change trigger threshold of the digital channel by setting the threshold voltage.

### 6.5.6.Window Trigger

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



If the high level and low level are both in the scope of the waveform, then the waveform triggers on the rising or falling edge at the same time.

If the high level is in the scope of the waveform, the low level isn't in the scope of the waveform, then the waveform only triggers on the rising edge.

If the high level isn't in the scope of the waveform, the low level is in the scope of the waveform, then the waveform only triggers on the falling edge.

Press **【Trig Menu】** on the frontal panel to turn on the trigger function menu.

**【Type】** select **window** and press V0 to confirm.

**【Data source】** select **CH1~CH4** to be trigger source.

**【Level】** enable high/low level setting function, press this softkey again to select low level (V2) or high level (V1), then rotate trigger level knob to adjust the vertical position of the high (low) level. The corresponding displacement information changes in real time and is displayed in the status bar in the upper right corner of the screen.

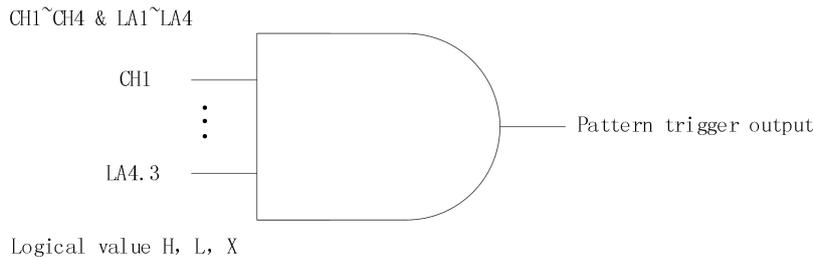
**【Mode】** select the acquisition mode (auto, normal), press V0 to confirm.

**【Holdoff】** set the holdoff time.

**Trigger level knob:** analog channel can modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob. Rotate the trigger level knob to acquire the stable trigger.

### 6.5.7. Pattern Trigger

Identify a trigger condition by looking for a specified pattern. This pattern is a logical combination of "AND" and "OR" of channels. Each channel can have a value of high (1), low (0) or invalid (X). In the pattern trigger mode, the oscilloscope will compare the actual pattern and the preset pattern of the channel, and trigger on the last channel that is the same as the preset pattern (the pattern is true). If the pattern type of each channel is fixed or all channels are set to "invalid", the oscilloscope will not trigger.



Press **【Trig Menu】** on the frontal panel to turn on trigger function menu.

**【Type】** select **Pattern** and press V0 to confirm.

**【Data source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:** when use LA to be trigger source, LA must be inserted.

**【Current channel pattern】** Set the pattern of current channel, press V0 to confirm. The preset pattern of the channel is displayed at the upper left of the screen.

1: set the pattern of the channel to be "H", namely the level is higher than the trigger level of the channel.

0: set the pattern of the channel to be "L", namely the level is lower than the trigger level of the channel.

X: set the pattern of the channel to be "X", namely this channel isn't used as a part of the pattern.

**【Mode】** select acquisition mode (auto, normal), press V0 to confirm.

【Holdoff】 set the holdoff time.

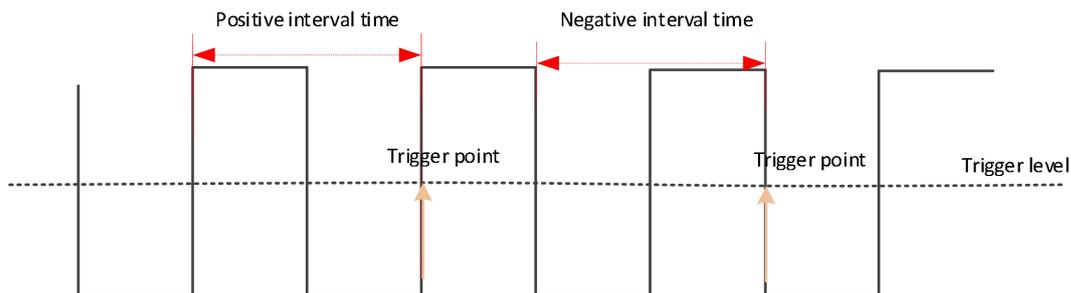
**Trigger level knob:**press channel button first then remove trigger level knob to modify the trigger level value of the corresponding channel. Trigger symbol moves up and down according to the rotation of the knob.Rotate the trigger level knob to adjust the trigger level to acquire the stable trigger.

For example,set the trigger level of CH1.Press CH1 softkey,use trigger level to modify the level.

**Digital channel:**can change trigger threshold of the digital channel by setting the threshold voltage.

### 6.5.8.Interval Trigger

The oscilloscope trigger when the interval between two continuous rising(or falling) edges satisfies the setting conditions(<, >, !=, =).



Press **【Trig Menu】** on the frontal panel to turn on trigger function menu.

【Type】 select **interval**,and press V0 to confirm.

【Data source】 select **CH1~CH4** or **LA** to be trigger source.

**Note:**when use LA to be trigger source,LA must be inserted.

【Slope】 select the positive polarity or negative polarity of the trigger.

【When】 select the trigger conditions(>、<、=、!=),and press V0 to confirm. 【the same with pulse trigger】

【Time】 set pulse width time 【8ns~10s】 .

【Mode】 select acquisition mode(auto、 normal),press V0 to confirm.

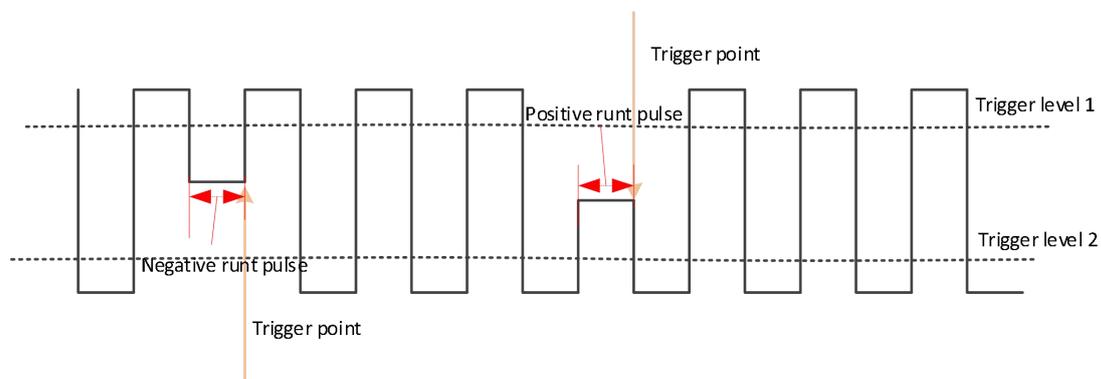
【Holdoff】 set the holdoff time.

**Trigger level knob:**analog channel can modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob.Rotate the trigger level knob to adjust the trigger level to acquire the stable trigger.

**Digital channel:**can change trigger threshold of the digital channel by setting the threshold voltage.

### 6.5.9.Under Amp Trigger

This trigger mode is used to trigger pulses that pass through one trigger level but fail to pass through the other trigger level as shown in the figure below.



The positive runt pulses pass the low level and fail to pass the high level.  
The negative runt pulses pass the high level and fail to pass the low level.

Set under AMP trigger:

Press **【Trig Menu】** on the frontal panel to turn on trigger function menu.

**【Type】** select **Under Amp** and press V0 to confirm.

**【Data source】** select **CH1~CH4** to be trigger source.

**【Polarity】** select the positive polarity or negative polarity of the trigger.

**【When】** select the desired slope conditions, and press V0 to confirm.

The setting conditions are the same with pulse.

**【Width】** set pulse width time **【8ns~10s】** .

**【Level】** enable high/low level setting function, press this softkey again to select low level (V2) or high level (V1), then rotate trigger level knob to adjust the vertical position of the high (low) level. The corresponding displacement information changes in real time and is displayed in the status bar in the upper right corner of the screen.

**【Mode】** select acquisition mode (auto、normal), press V0 to confirm.

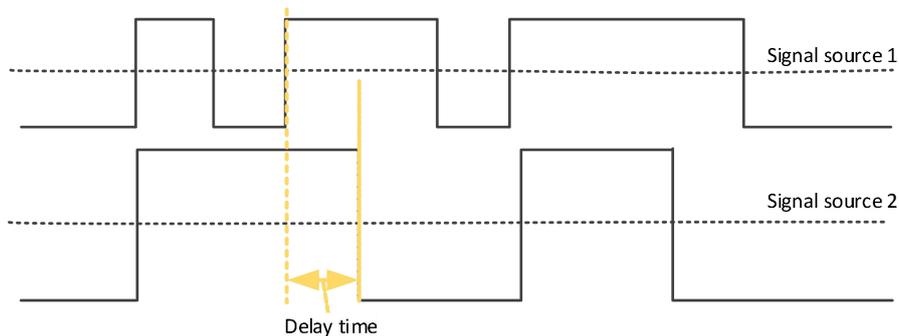
**【Holdoff】** set holdoff time.

**Trigger level knob:** can modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob. Rotate the trigger level knob to adjust the trigger level to acquire the stable trigger.

### 6.5.10. Delay Trigger

In delay trigger, you need to set signal source 1 and signal source 2. The oscilloscope triggers when the time difference between the specified edges of source 1 (Edge 1) and source 2 (Edge 2) meets the preset time limit, as shown in the figure below.

**Note:** Edge 1 and Edge 2 must be neighbouring edges.



Press **【Trig Menu】** on the frontal panel to turn on trigger function menu.

**【Type】** select **delay** and press V0 to confirm.

**【Data source 1】** select **CH1~CH4** or **LA** to be trigger source.

**Note:** when use LA to be trigger source, LA must be inserted.

**【Slope 1】** select the desired trigger edge (rising edge or falling edge)

**【Data source 2】** select **CH1~CH4** or **LA** to be trigger source.

**Note:** when use LA to be trigger source, LA must be inserted.

**【Slope 2】** select the desired trigger edge (rising edge or falling edge)

**【When】** select the desired slope conditions, and press V0 to confirm.

The setting conditions are the same with pulse.

**【Width】** set pulse width time **【8ns~10s】** .

**【Mode】** select acquisition mode (auto、normal), press V0 to confirm.

**【Holdoff】** set the holdoff time.

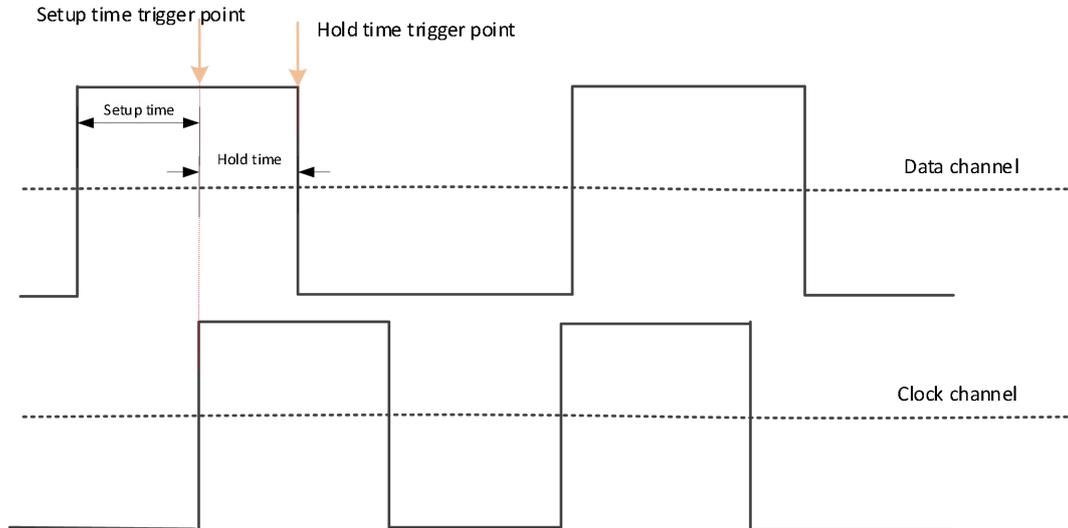
**Trigger level knob:** press channel button first then remove trigger level knob to modify the trigger level value of the corresponding channel. Trigger symbol moves up and down according to the rotation of the knob. Rotate the trigger level knob to adjust the trigger level to acquire the stable trigger.

For example, set the trigger level of CH1. Press CH1 softkey, use trigger level to modify the level.

**Digital channel:** can change trigger threshold of the digital channel by setting the threshold voltage.

### 6.5.11. Setup/Hold Trigger

In setup/hold trigger, you need to set the data signal line and clock signal line. The setup time starts when the data signal passes the trigger level and ends at the coming of the specified clock edge; the hold time starts at the coming of the specified clock edge and ends when the data signal passes the trigger level again (as shown in the figure below). The oscilloscope triggers when the setup time or hold time is lower than the preset time.



Press **【Trig Menu】** on the frontal panel to turn on the trigger function menu.

**【Type】** select **setup/hold** and press V0 to confirm.

**【DAT】** select **CH1~CH4** or **LA** to be trigger source.

**Note:** when use LA to be trigger source, LA must be inserted.

**【Data type】** select the desired data type (high level or low level).

**【CLK】** select **CH1~CH4** or **LA** to be CLK trigger source.

**Note:** when use LA to be trigger source, LA must be inserted.

**【Slope】** select the desired trigger edge (rising edge or falling edge)

**【Polarity】** select hold mode (setup or hold)

Setup: the time in which the data is stable before the clock edge comes.

Hold: the time in which the data is stable after the clock edge comes.

**【When】** select the desired trigger condition, and press V0 to confirm.

The setting conditions are the same with pulse.

**【Width】** set pulse width time **【8ns~10s】**.

**【Mode】** select the acquisition mode (auto, normal), press V0 to confirm.

**【Holdoff】** set the holdoff time.

**Trigger level knob:** press channel button first then remove trigger level knob to modify the trigger level value of the corresponding channel. Trigger symbol moves up and down according to the rotation of the knob. Rotate the trigger level knob to adjust the trigger level to acquire the stable trigger.

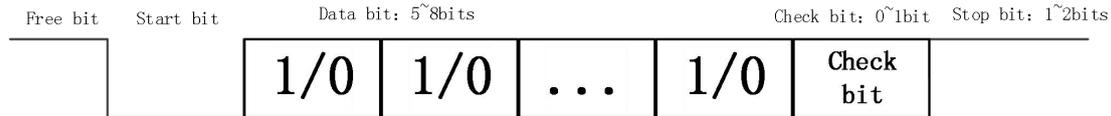
For example, set the trigger level of CH1. Press CH1 softkey, use trigger level to modify the level.

**Digital channel:** can change trigger threshold of the digital channel by setting the threshold voltage.

### 6.5.12. UART Trigger (optional)

UART bus is a serial communication mode for data transmission between computers or between computers and terminals. UART serial protocol transmits a character as a frame of data, and its frame structure consists of 1 bit starting bit, 5~8 bits data bit, 1 bit check bit

and 1~2 bits stop bit.



The format is shown in the figure above. DPO6000/MPO6000 series oscilloscope can be triggered when detecting the frame start and stop bit of UART signal, error frame, check error or specified data.

Press **【Trig Menu】** on the frontal panel to enter trigger function menu.

**【Type】** select **UART** and press **V0** to confirm.

**【Data source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:** when use LA to be trigger source, LA must be inserted.

**【Baud rate】** select baud rate.

DPO6000/MPO6000 series oscilloscopes provide the following baud rate for customers

110	300	600	1200	2400
4800	9600	14400	19200	38400
57600	115200	230400	380400	460400
user-defined				

If customers can't find the desired baud rate in the table above, can select user-defined, then set your own baud rate.

**【User-defined】** the user can set his own baud rate {only when selecting user-defined, this menu can be used }

**【When】** set UART trigger conditions:

**Start bit:** when UART start bit appears, the oscilloscope will trigger at the center of the bit.

**Stop bit:** when UART stop bit appears, the oscilloscope will trigger at the center of the bit. No matter the stop bit of the equipment under test is 1, 1.5, 2, the machine disposes as 1 bit.

**Data:** the oscilloscope triggers at the stop bit when data is accepted normally and the accepted UART data is equal to the data set by users.

**Odd-even check error:** the oscilloscope triggers at the stop bit when data is accepted normally and the data odd-even check goes wrong.

**Data bit error:** the oscilloscope will trigger when the data goes wrong during the check of start bit and stop bit.

**【Free level】** the level value when UART is free (level/high level), default setting is high level.

**【Odd-even check】** set the UART data under test to be checked or not (odd check, even check or none).

**【Data bits】** set the length of the UART data under test (can select 5, 6, 7, 8 bits).

**【Data】** the user-defined trigger data (only useful when trigger type is data).

**【Mode】** select acquisition mode (auto, normal), press **V0** to confirm.

**【Holdoff】** set the holdoff time.

**【Decode】** switch decoding function.

**【Label location】** the position of decoding data drawn on the screen (only useful when decoding function is turned on)

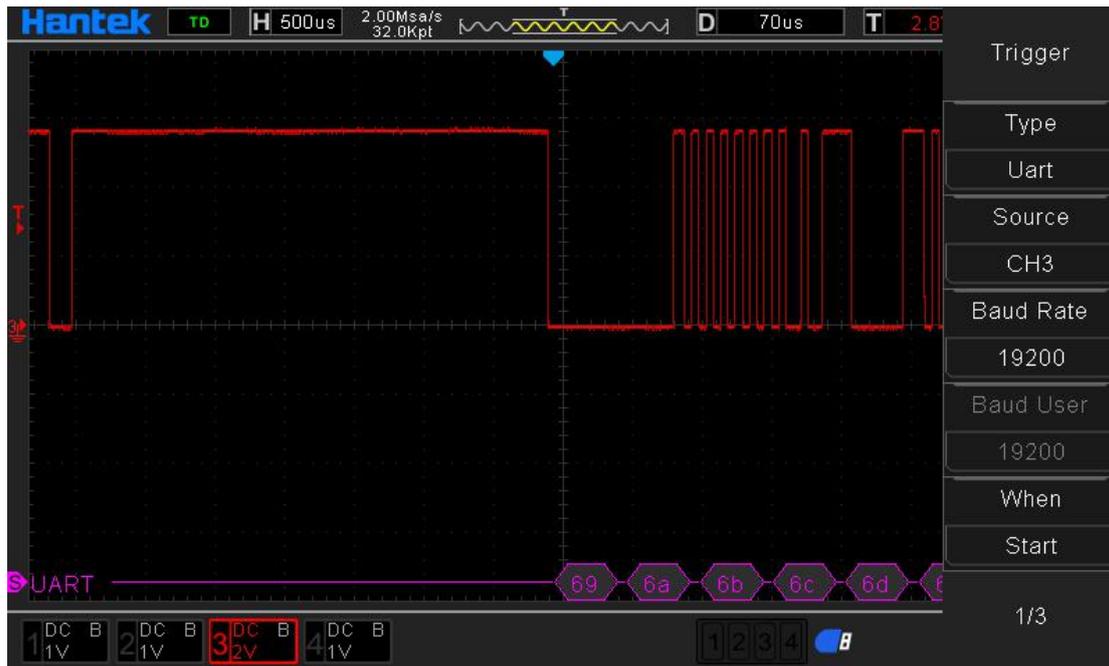
**【Table】** data recording mode.

**Trigger level knob:** the analog channel can modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob.

**Digital channel:** can change trigger threshold of the digital channel by setting the threshold voltage.

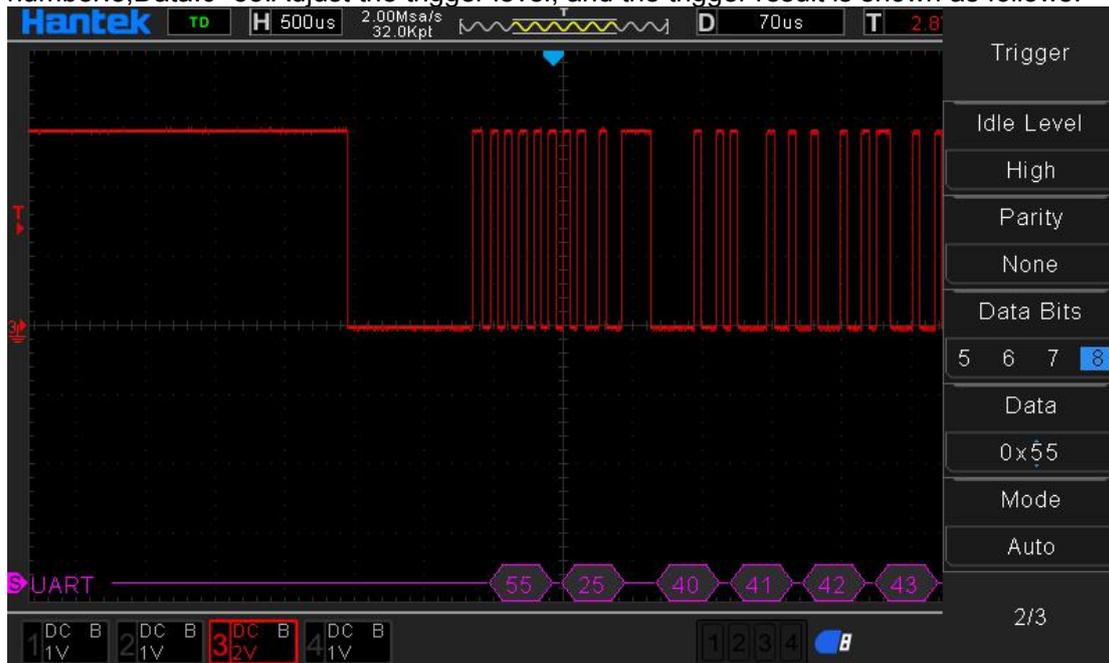
Trigger setting 1:

Data source: CH3; Baud rate: 19200; When: start; Free level: high; Parity check: none; Data number: 8. Adjust the trigger level, and the trigger result is shown as follows:



Trigger setting 2:

Data source:CH3;Baud rate:19200;When:Data;Free level:high;Parity check:none;Data number:8;Data:0x55.Adjust the trigger level, and the trigger result is shown as follows:



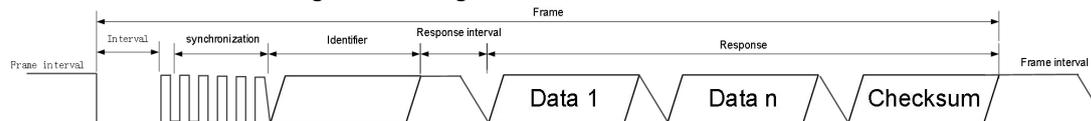
UART decoding interpretation:

1. Decoded data is displayed in hexadecimal;
2. Display color of decoded data is purple;
3. When "?" or "please adjust the time base" appears, the time base needs to be adjusted to get the decoded result.

### 6.5.13.LIN Trigger(optional)

LIN bus is a low-cost serial communication network defined for automotive distributed electronic system, and it is a supplement to controller area network (CAN) and other automotive multiplexing networks, Suitable for applications that do not require too much on bandwidth, performance or fault tolerance of the the network. LIN bus is based on SCI(UART) data format and adopts the mode of single master controller/multi-slave device, which is a special case in UART.

LIN trigger can be triggered at the rising edge of the synchronization interrupt exit of the LIN single-line bus signal (which marks the beginning of the message frame), frame ID, or frame ID and data. LIN signal message frame is shown as below:



Press **【Trig Menu】** on the frontal panel to turn on the trigger function menu.

**【Type】** select **LIN** and press **V0** to confirm.

**【Data source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:** when use **LA** to be trigger source, **LA** must be inserted.

**【Baud rate】** select baud rate.

DPO6000/MPO6000 series oscilloscopes provide the following baud rate for customers.

110	300	600	1200	2400
4800	9600	14400	19200	38400
57600	115200	230400	380400	460400
user-defined				

If customers can't find the desired baud rate in the table above, can select user-defined, then set your own baud rate.

**【User-defined】** the user can set his own baud rate {only when selecting user-defined, this menu can be used }

**【When】** set LIN trigger conditions:

**interval field:** edge trigger when LIN interval ends

**synchronization field:** trigger when the data of LIN synchronization field is accepted.

**ID field:** trigger when the data of LINID field is accepted.

**Synchronous code error:** trigger when the data of LIN synchronous field is accepted and isn't equal to 0x55.

**Frame ID:** trigger when LINID field data is accepted and the ID data is equal to the user-defined ID.

**Frame ID and data:** trigger when LIN data is accepted normally and the ID and the data are both equal to the user-defined data.

**【Identifier】** the identifier data when LIN triggers.

**【Data1】** the data when LIN triggers.

**【Data2】** the data when LIN triggers.

**【Data3】** the data when LIN triggers.

**【Data4】** the data when LIN triggers.

**【Free level】** the level value of LIN frame gap(level/high level), default setting is high level.

**【Mode】** select acquisition mode(auto、 normal), press **V0** to confirm.

**【Holdoff】** set the holdoff time.

**【Decoding】** switch decoding function.

**【Label location】** the position of the decoding data drawn on the screen **【only useful when decoding function is turned on】**

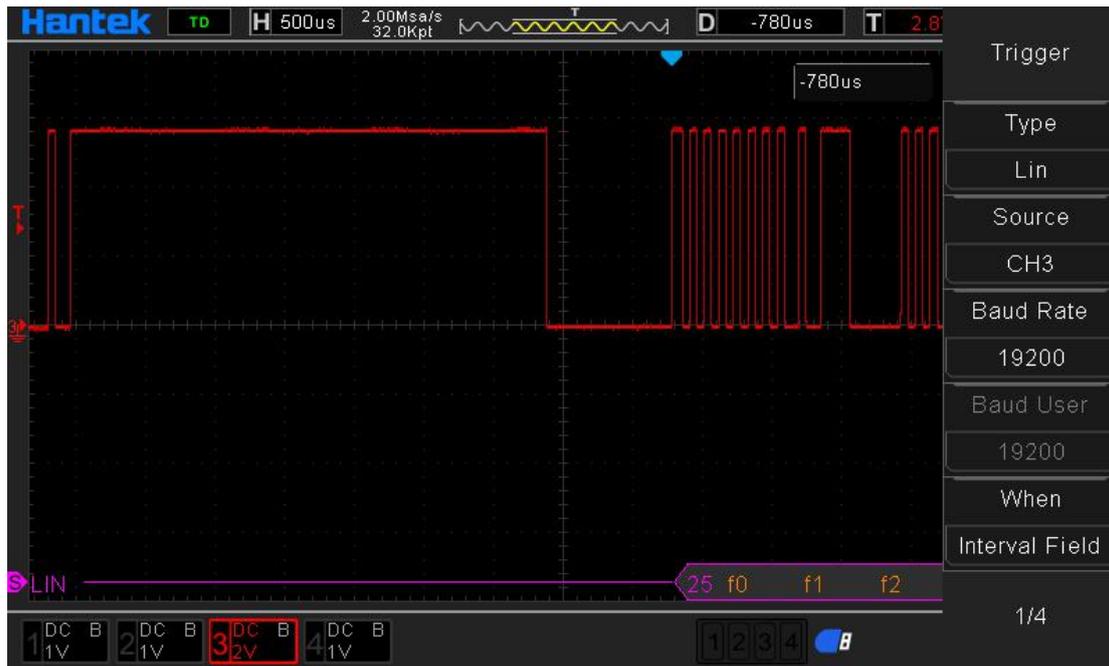
**【Table】** data recording mode

**Trigger level knob:** the analog channel can modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob.

**Digital channel:** can change trigger threshold of the digital channel by setting the threshold voltage.

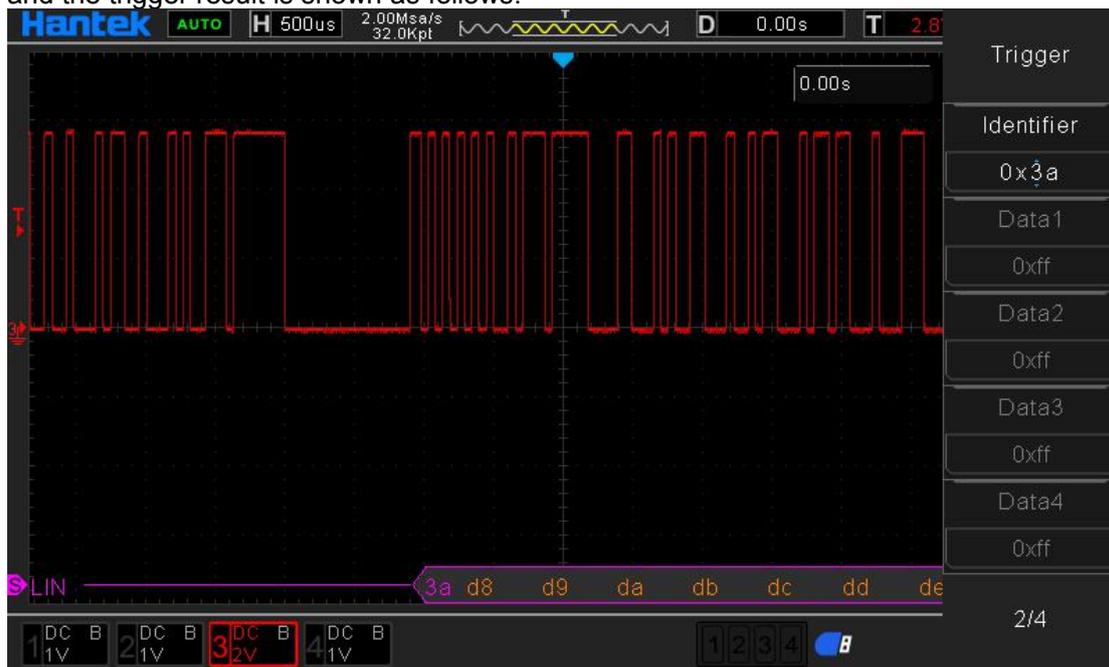
Trigger setting 1:

Data source: CH3; Baud rate: 19200; When: interval field; Free level: high; Adjust the trigger level, and the trigger result is shown as follows:



Trigger setting 2:

Data source:CH3;Baud rate:19200;When:identifier;Free level:high;Adjust the trigger level, and the trigger result is shown as follows:



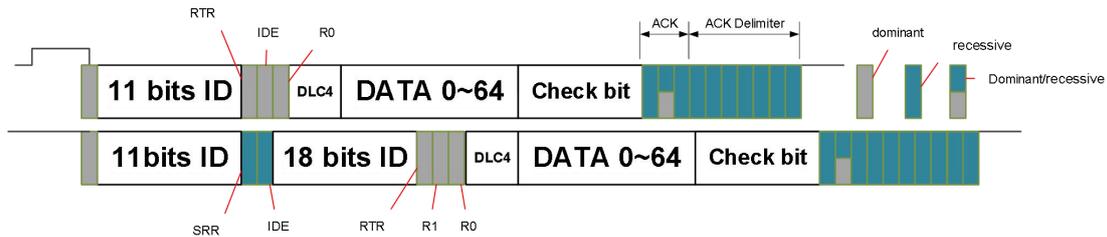
LIN decoding interpretation:

1. Decoded data is displayed in hexadecimal;
2. The display color of "identifier" and "checksum" is purple, and the display color of "data" is orange;
3. When "?" or "please adjust the time base" appears, the time base needs to be adjusted to get the decoded result;
4. In LIN decoding results, the synchronization field "55" is not decoded and displayed.

#### 6.5.14.CAN Trigger(optional)

CAN is short for Controller Area Network. It was developed by BOSCH, a German company famous for researching and producing automotive electronic products, and eventually becomes an international standard(ISO 11898), it is one of the most widely used field buses in the world. In North America and Western Europe, CAN bus protocol

has become the standard bus of automotive computer control system and embedded industrial control LAN, and has J1939 protocol designed for large trucks and heavy machinery vehicles with CAN as the underlying protocol.



CAN trigger setting:

Press **【Trig Menu】** on the frontal panel to turn on the trigger function menu.

**【Type】** select CAN and press V0 to confirm.

**【Signal source】** select CH1~CH4 or LA to be trigger source.

**Note:** when use LA to be trigger source, LA must be inserted.

**【Baud rate】** select baud rate.

DPO6000/MPO6000 series oscilloscopes provide the following baud rate for customers

10000	20000	33300	50000	62500
83300	100000	125000	250000	500000
800000	1000000	User-defined		

If customers can't find the desired baud rate in the table above, can select user-defined, then set your own baud rate.

**【User-defined】** the user can set his own baud rate {only when selecting user-defined, this menu can be used }

**【When】** set CAN trigger conditions:

**The start of the frame:** trigger from the central of the start bit of CAN.

**Remote frame ID:** trigger when the frame type is CAN frame, frame ID is accepted and ID is equal to the user-defined value.

**Data frame ID:** trigger when CAN type is data frame, frame ID is accepted and ID is equal to the user-defined value.

**Frame ID:** trigger when CAN finishes accepting frame ID and ID is equal to the user-defined value.

**Frame ID and data:** Trigger when CAN data is accepted and ID and data is equal to the user-defined value.

**Wrong frame:** trigger when the wrong frame of CAN is detected.

**All errors:** trigger when the oscilloscope detects the wrong frame, bit error, confirmation error of CAN.

**Confirmation error:** trigger when the confirmation error of CAN is detected.

**Overload frame:** trigger when the overload frame of CAN is detected.

**【Identifier】** the identifier data when CAN trigger

**【Data1】** the data at the time of CAN trigger.

**【Data2】** the data at the time of CAN trigger.

**【Data3】** the data at the time of CAN trigger.

**【Data4】** the data at the time of CAN trigger.

**【Free level】** the level value in the interval of CAN frame (level/high level), default setting is high level.

**【Mode】** select acquisition mode (auto, normal), press V0 to confirm.

**【Holdoff】** set the holdoff time.

**【Decoding】** switch decoding function.

**【Label location】** the position of the decoding data drawn on the screen (only useful when decoding function is turned on)

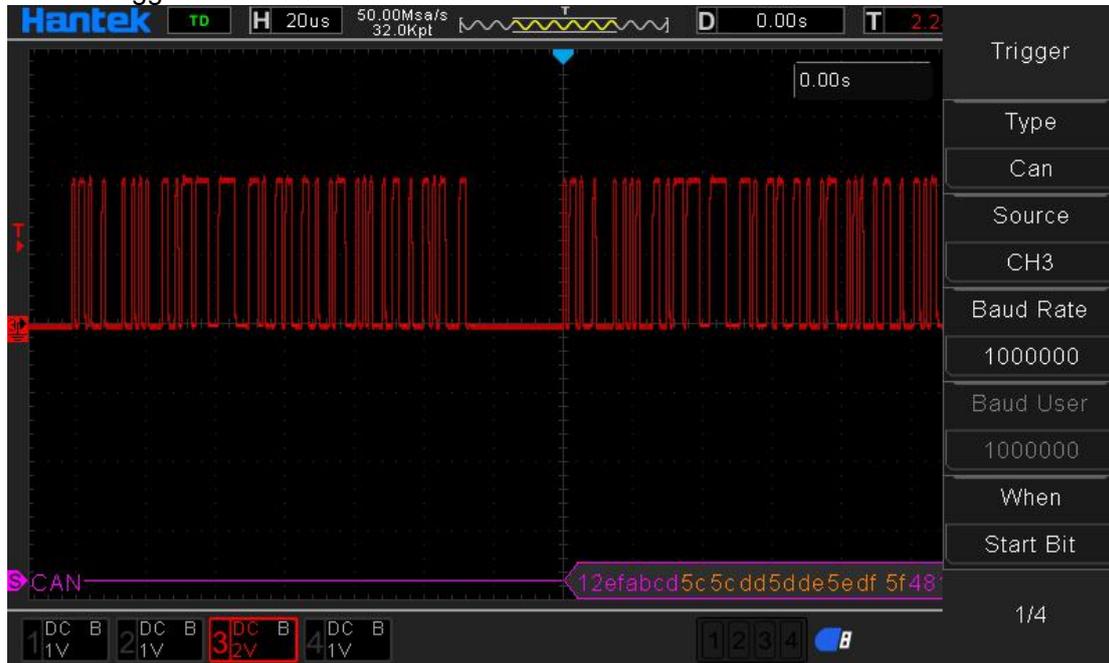
**【Table】** data recording mode.

**Trigger level knob:** the analog channel can modify the trigger level value. Trigger symbol moves up and down according to the rotation of the knob.

**Digital channel:** can change trigger threshold of the digital channel by setting the threshold voltage.

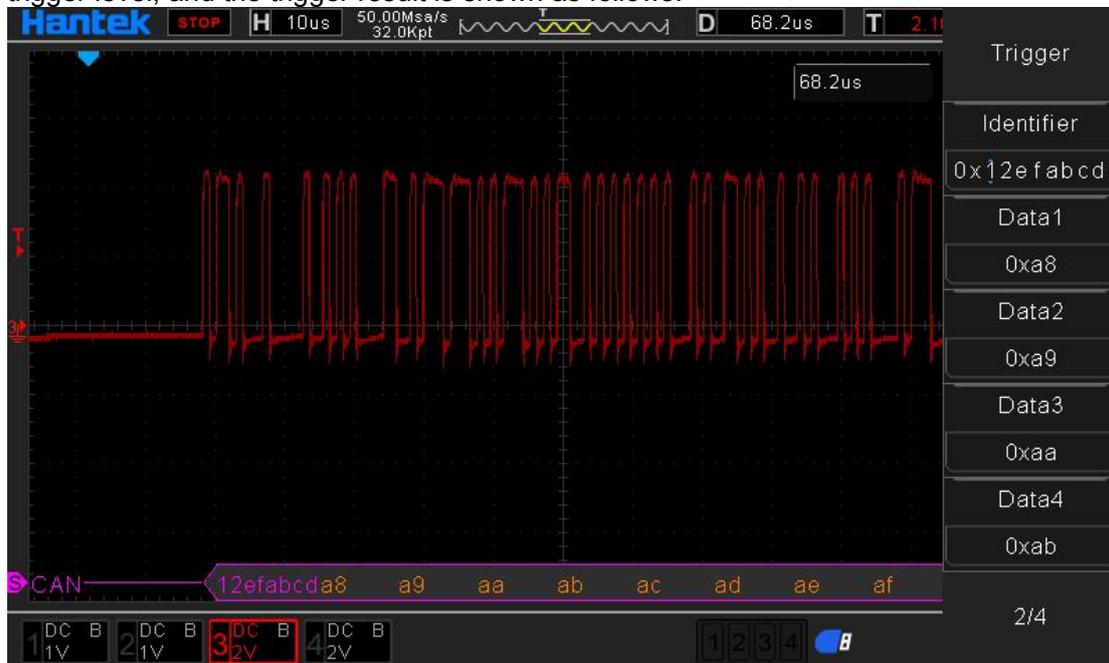
Trigger setting 1:

Data source: CH3; Baud rate: 1000000; When: start bit; Free level: low. Adjust the trigger level, and the trigger result is shown as follows:



Trigger setting 2:

Data source: CH3; Baud rate: 1000000; When: data frames and data; Identifier: 12efabcd; Data1: 0xa8; Data2: 0xa9; Data3: 0xaa; Data4: 0xab; Free level: low; Mode: normal. Adjust the trigger level, and the trigger result is shown as follows:



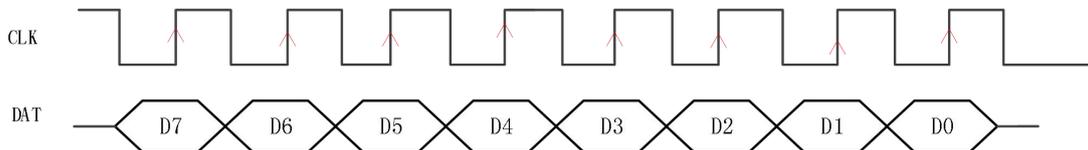
CAN decoding interpretation:

1. Decoded data is displayed in hexadecimal;
2. The display color of "frame ID" and "check bit" is purple, and the display color of "data" is orange;
3. When "?" or "please adjust the time base" appears, the time base needs to be adjusted to get the decoded result.

### 6.5.15.SPI Trigger(optional)

SPI is the short of Serial Peripheral Interface.It is a synchronous serial interface technology launched by Motorola corporation, is a high-speed, full duplex, synchronous communication bus.

In SPI trigger, when a timeout condition is met, the oscilloscope will trigger when the specified data is found. When using SPI trigger, SCL clock sources and SDA data sources need to be specified.SPI bus timing diagram is as follows:



Press **【Trig Menu】** on the frontal panel to turn on trigger function menu.

**【Type】** select SPI and press V0 to confirm.

**【Clock source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:**when use LA to be trigger source, LA must be inserted.

**【Slope】** select the desired trigger edge(rising edge、falling edge、double edge),press V0 to confirm.

**【Data source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:**when use LA to be trigger source, LA must be inserted.

**【Overtime】** The timeout must be greater than the period of the CLK clock source.

**【Data bits width】** set up the data bits width.

**【Data bit】** set up the data bit.

**【Mode】** select acquisition mode(auto、normal),press V0 to confirm.

**【Holdoff】** set the holdoff time.

**【Decoding】** switch decoding function.

**【Label location】** the position of the decoding data drawn on the screen(only useful when decoding function is turned on)

**【Table】** data recording mode.

**Trigger level knob:**he analog channel can modify the trigger level value.Trigger symbol moves up and down according to the rotation of the knob.

**Digital channel:**can change trigger threshold of the digital channel by setting the threshold voltage.

Trigger setting:

Clock source:CH3;Slope:rising;Data source:CH4;Timeout:366ns;Data bit width:8;Data bits:7bits.Adjust the trigger level, and the trigger result is shown as follows:



SPI decoding interpretation:

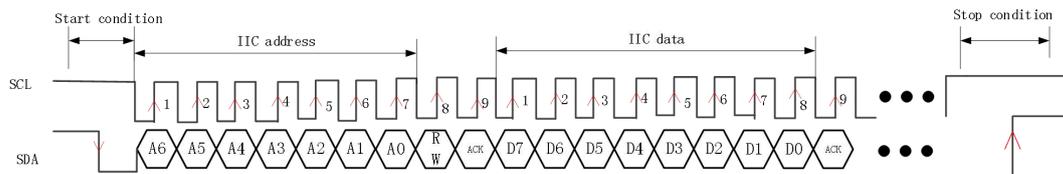
1. Decoded data is displayed in hexadecimal;
2. The display color of "data" is purple;
3. When "?" or "please adjust the time base" appears, the time base needs to be adjusted to get the decoded result.

#### 6.5.16. IIC Trigger (optional)

IIC is the short of Inter-Integrated Circuit BUS. This bus is designed by NXP (original PHILIPS) company, which is mainly used for master-slave communication between master controller and slave device. It is used in occasions of small data quantity, with short transmission distance and only one host at any time and other characteristics. IIC address addressing mode is divided into 7-bit addressing mode and 10-bit addressing mode.

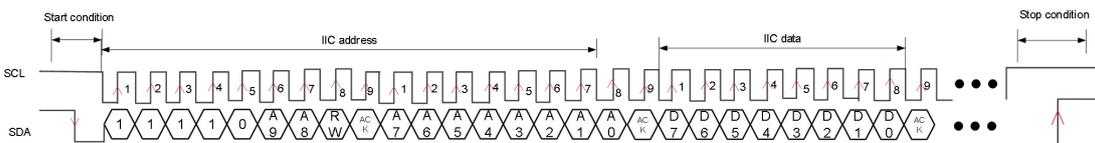
##### 7-bit addressing

In the 7-bit addressing process, the slave address is transmitted at the first byte after the signal is started. The first seven bits of this byte are the slave address, and the eighth bit is the read-write bit, where 0 represents write and 1 represents read.



##### 10-bit addressing

10-bit addressing and 7-bit addressing of I2C bus are compatible, so that devices with both 7-bit address and 10-bit address modes can be used on the same bus. In 10-bit address transmission, the first byte is a special reserved address to indicate that the current transmission is a 10-bit address.



IIC (internal integrated circuit bus) signal setting includes the serial data (SDA) line and the serial clock (SCL) line connected to the oscilloscope, and then specifies the threshold voltage level of the input signal.

Press **【Trig Menu】** on the frontal panel to turn on the trigger function menu.

**【Type】** select IIC and press V0 to confirm.

**【Address width】** select **7-bit or 10-bit**.

**【Clock source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:**when use LA to be trigger source,LA must be inserted.

**【Data source】** select **CH1~CH4** or **LA** to be trigger source.

**Note:**when use LA to be trigger source,LA must be inserted.

**【When】** set IIC trigger conditions:

**Starting bit:**trigger when IIC starting conditions are detected.

**Stopping bit:**trigger when IIC stopping conditions are detected.

**No response:**trigger when IIC bus response level is wrong.

**Address:**trigger when IIC address data that is accepted is equal to the user-defined value.

**Restart:**trigger when the new starting conditions appear before stopping conditions appear.

**Address and data:**trigger when IIC address is accepted and the data **【4-byte data】** is equal to the user-defined value.

**【Address】** the identifier data at the time of IIC trigger.

**【Data1】** the data at the time of IIC trigger.

**【Data2】** the data at the time of IIC trigger.

**【Data3】** the data at the time of IIC trigger.

**【Data4】** the data at the time of IIC trigger.

**【Mode】** select acquisition mode(auto、 normal),press V0 to confirm.

**【Holdoff】** set the holdoff time.

**【Decoding】** switch decoding function.

**【Label location】** the position of the decoding data drawn on the screen(only useful when decoding function is turned on)

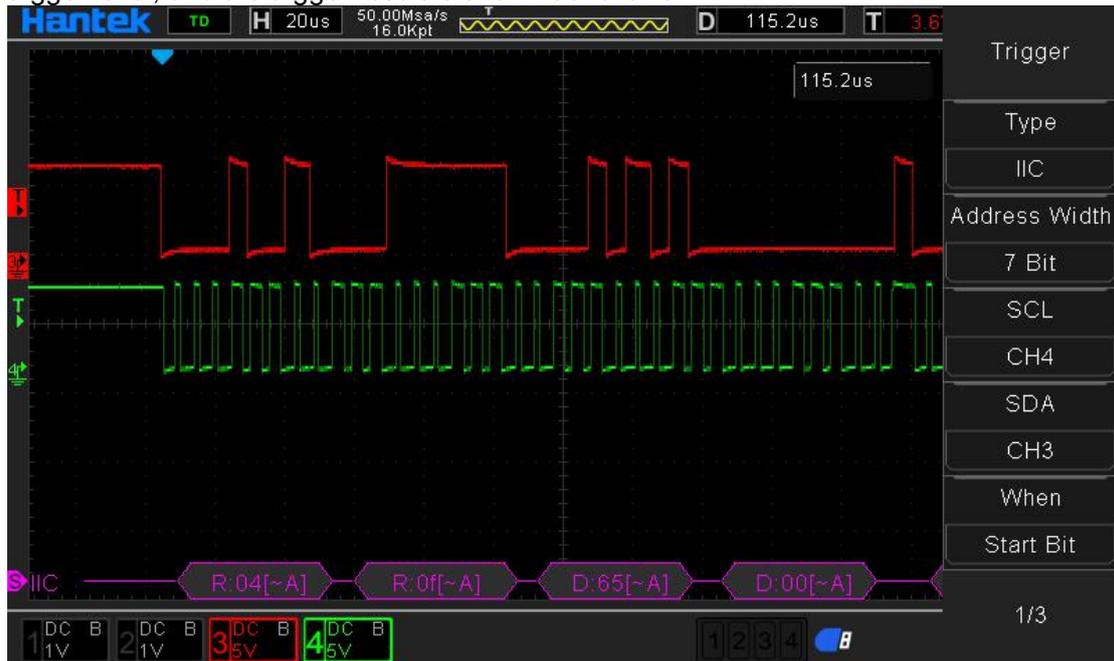
**【Table】** data recording mode.

**Trigger level knob:**he analog channel can modify the trigger level value.Trigger symbol moves up and down according to the rotation of the knob.

**Digital channel:**can change trigger threshold of the digital channel by setting the threshold voltage.

Trigger setting:

Address width:7 digits;Clock source:CH4;Data source:CH3;When:start bit.Adjust the trigger level, and the trigger result is shown as follows:



IIC decoding interpretation:

1. Decoded data is displayed in hexadecimal;
2. The display colors of "address" and "data" are purple;"W" means write operation, "R" means read operation, "D" means decode data, "~A" means no confirmation bit;
3. When "?" or "please adjust the time base" appears, the time base needs to be adjusted to get the decoded result.

## 6.6.Protocol Decoding

Protocol analysis can be used by users to discover errors, debug hardware and accelerate development easily, DPO6000/MPO6000 series oscilloscopes provide 5 kinds of common protocol decoding, including UART, LIN, CAN, I2C and SPI.

**Note:** Only when the user correctly sets protocol trigger setting parameters can get the correct decoding result.

### 6.6.1.Decoding mode

The decoding function is only available at the time of protocol trigger. Turn on the decoder switch, and the magenta parallel decoder cursor appears on the left side of the oscilloscope [default is at the bottom of the screen]. By selecting the label location menu, the user can change the label position through V0 to make the decoded position waveform has a better display position, so it's convenient for the user to analyze the decoded data.

### 6.6.2.Table mode

The table function is only available at the time of protocol trigger. When the form mode is turned on, the oscilloscope will not display the acquisition data of the waveform, only display the correct protocol decoded data. The decoded data will be transmitted to the screen in real time, and the form mode can decode data continuously. When an oscilloscope runs in table mode, only the F1, F2, F3, RUN/Stop softkey is available. Other keys cannot be used. If the users need to set other parameters of the serial port, they need to exit the table mode. After users make a pause, the decoded data can be exported for analysis.

## 7.Math Operation

### 7.1.Addition

Add the waveform voltage values of signal source 1 and 2 point by point and display the results.

Operating steps

- 1.press **【Math】** on the frontal panel to enter math function menu.
- 2.Press the soft keys of source 1 and source 2, respectively, and rotate the multifunctional knob V0 to select the source of Math operation.All four analog channels can be used as source 1 and source 2.
- 3.Press the operation soft key, then rotate the multifunctional knob V0 to select + for addition operation.The resulting mathematical waveform is displayed on the screen and marked "M".

### 7.2.Subtraction

Subtract the waveform voltage values of signal source 2 from that of source 1 point by point and display the results.

- 1.press **【Math】** on the frontal panel to enter math function menu.
- 2.Press the soft keys of source 1 and source 2, respectively, and rotate the multifunctional knob V0 to select the source of Math operation.All four analog channels can be used as source 1 and source 2.
- 3.Press the operation soft key, then rotate the multifunctional knob V0 to select - for subtraction operation.The resulting mathematical waveform is displayed on the screen and marked "M".

### 7.3.Multiplication

Multiply the waveform voltage values of signal source 1 and 2 point by point and display the results.

- 1.press **【Math】** on the frontal panel to enter math function menu.
- 2.Press the soft keys of source 1 and source 2, respectively, and rotate the multifunctional knob V0 to select the source of Math operation.All four analog channels can be used as source 1 and source 2.
- 3.Press the operation soft key, then rotate the multifunctional knob V0 to select \* for multiplication operation.The resulting mathematical waveform is displayed on the screen and marked "M".

Scale: press the scale soft key, then rotate the multifunctional knob to select the vertical scale.

### 7.4.Division

Divide the waveform voltage values of signal source 2 by that of source 1 point by point and display the results. It can be used to analyze the multiple relationships of waveform of two channels.

- 1.press **【Math】** on the frontal panel to enter math function menu.
- 2.Press the soft keys of source 1 and source 2, respectively, and rotate the multifunctional knob V0 to select the source of Math operation.All four analog channels can be used as source 1 and source 2.
- 3.Press the operation soft key, then rotate the multifunctional knob V0 to select / for division operation.The resulting mathematical waveform is displayed on the screen and marked "M".

Scale: press the scale soft key, then rotate the multifunctional knob to select the vertical scale.

**Note:** When the voltage of signal source 2 is 0V, the division result is invalid and "NAN" is displayed at the bottom of the screen.

### 7.5.FFT

FFT uses analog input channels or reference waveform to compute the fast Fourier

transform. FFT is used to transform time domain signals to frequency components (frequency spectrum). Select FFT function and plotted FFT spectrum shows the relationship between dB V and frequency. The horizontal reading changes from time to frequency (Hz) and the vertical reading from V to dB. The FFT operations can be used for the following purposes:

Measure harmonic components and distortion in the system.

Display the characteristics of the noise in DC power

Analyze vibration

Display FFT spectrum:

1. Press **【Math】** on the frontal panel to turn on math function menu.
2. Press the soft key of **【operator】**, and rotate the multifunctional knob V0 to select FFT. The resulting mathematical waveform is displayed on the screen and marked "M".
3. Press the source soft key and rotate the multifunctional knob V0 to select the FFT operating source. All four analog channels can be used as signal sources.
4. Press the center soft key and rotate the multifunctional knob V0 to adjust the frequency of the frequency domain waveform corresponding to the horizontal center of the screen.
5. Press the span soft key and rotate the multifunctional knob V0 to adjust the horizontal scale of the frequency domain waveform.
6. Press the vertical unit soft key to select the vertical axis unit. Units of the vertical axis can use either a logarithmic scale to show the dB of the vertical amplitude or a linear scale to show the Vrms of the vertical amplitude. dB Vrms is recommended if you need to display FFT spectrum over a relatively large dynamic range.
7. Press the scale soft key to select the vertical scale.
8. Press the window soft key to select the appropriate window.

Spectral leakage can be greatly reduced by using window functions. The range provides six different characteristic FFT window functions for measuring different waveform. You need to select window functions according to the different waveform and their characteristics. Please read the table below carefully and make appropriate choices based on the input signals.

Window	Measurement	Characteristics
Rectangular	Transient or short pulse waveform	Dedicated window for discrete window similar to the situation when no window is multiplied
Hanning	Period waveform	Better frequency resolution Poorer amplitude resolution
Hamming	Transient or short pulse waveform	A little bit better frequency resolution than Hanning
Blackman	Single frequency signal, search for higher order harmonics	The best amplitude resolution; the poorest frequency resolution
Bartlett (Triangle)	Stronger narrow band signal	Better frequency resolution
Flattop	Period waveform	Better amplitude resolution Poorer frequency resolution

9. Press the alone soft key to select only display FFT result and not display source channel.

**Note:**

1. A signal with a dc component or bias will result in an error or bias in the FFT waveform component. To reduce the dc component, the "channel coupling" of the source can be set to "ac" mode.

2. In order to reduce the random noise and aliasing frequency components of repeated or single pulse events, the "acquisition mode" of oscilloscope can be set to "average" mode.

**Use the cursor to measure the FFT waveform.**

To make cursor measurements, First press [Cursors] in the front panel of the oscilloscope to turn on the cursor measurement.

Press the Mode soft mode key to select manual or trace. Use AX and BX cursors to measure the frequency and difference between two frequencies (BX-AX). Measure amplitude and amplitude difference using AY and BY cursor (BY-AY).

## 7.6. Integral

Calculate the integral of the selected source. For example, you can use integral to measure the pulse energy or the area under a waveform.

1. Press **【Math】** on the frontal panel to turn on math function menu.
2. Press the source 1 and source 2 soft keys, respectively, and rotate the multifunctional knob V0 to select the source for the Math operation. All four analog channels can be used as source 1 and source 2.
3. Press the soft key of **【operator】**, and then rotate the multifunctional knob V0 to select integral operation. The resulting mathematical waveform is displayed on the screen, marked "M".

## 7.7. Differential

Calculate the discrete time derivative of the selected source. For example, you can use differentiate to calculate the instantaneous slope of a waveform.

1. Press **【Math】** on the frontal panel to turn on math function menu.
2. Press the source 1 and source 2 soft keys, respectively, and rotate the multifunctional knob V0 to select the source for the Math operation. All four analog channels can be used as source 1 and source 2.
3. Press the soft key of **【operator】**, and then rotate the multifunctional knob V0 to select differential operation. The resulting mathematical waveform is displayed on the screen, marked "M".

## 7.8. Square root

Calculate the square root of the selected source point by point and display the results. When the operation is invalid, "NAN" is displayed at the bottom of the screen.

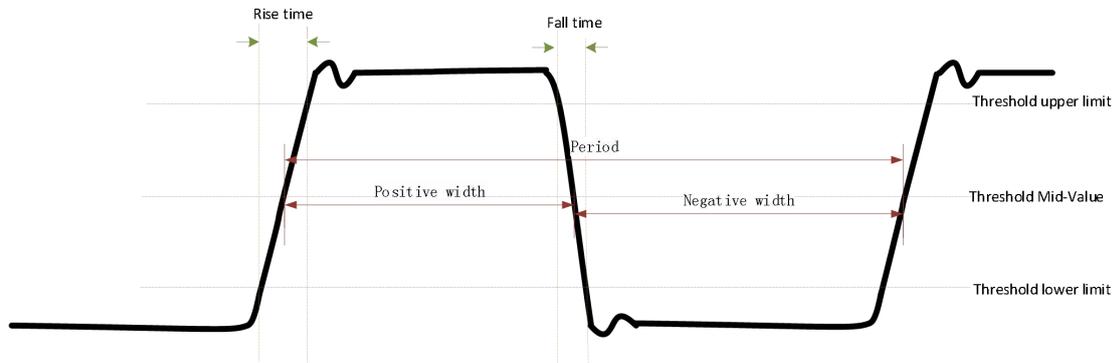
1. Press **【Math】** on the frontal panel to turn on math function menu.
2. Press the source 1 and source 2 soft keys, respectively, and rotate the multifunctional knob V0 to select the source for the Math operation. All four analog channels can be used as source 1 and source 2.
3. Press the soft key of **【operator】**, and then rotate the multifunctional knob V0 to select square root operation. The resulting mathematical waveform is displayed on the screen, marked "M".

## 8.Measurement

### 8.1.Measurement parameter specification

DPO6000/MPO6000 series oscilloscopes provide auto measurements of 42 waveform parameters and the statistics and analysis of the measurement results. What's more, you can also use the digital multimeter(DVM) to realize more precise frequency measurement.

#### 8.1.1.Time parameter



1. **Period[Pk-Pk]**:defined as the time between the middle threshold points of two consecutive, like-polarity edges.
2. **[Frequency]**:defined as the reciprocal of period.
3. **[Rise Time]**:the time for the signal amplitude to rise from the threshold lower limit to the threshold upper limit.
4. **[Fall Time]**: the time for the signal amplitude to fall from the threshold upper limit to the threshold lower limit.
5. **[+Width]**: the time difference between the threshold middle value of a rising edge and the threshold middle value of the next falling edge of the pulse.
6. **[-Width]**:the time difference between the threshold middle value of a falling edge and the threshold middle value of the next rising edge of the pulse.
7. **[+Duty]**:the ratio of the positive pulse width to the period.

$$+ \text{Duty} = \frac{\text{positive pulse}}{\text{period}} \times 100\%$$

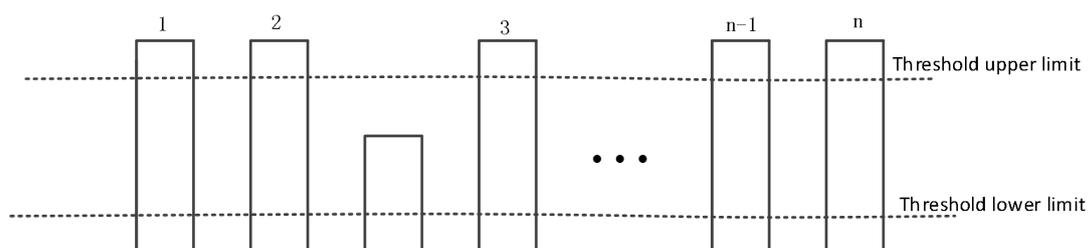
8. **[-Duty]**:the ratio of the negative pulse width to the period.

$$- \text{Duty} = \frac{\text{negative pulse}}{\text{period}} \times 100\%$$

9. **[BWidth]**:The time from the first edge of the data source to the last edge of the data source.
  10. **[Vmax Time]**:the time corresponding to the waveform maximum value (Vmax).
  11. **[Vmin Time]**:the time corresponding to the waveform minimum value (Vmin).
- Note:** The default values of the threshold upper limit, threshold middle value and threshold lower limit are 90%, 50% and 10% respectively.

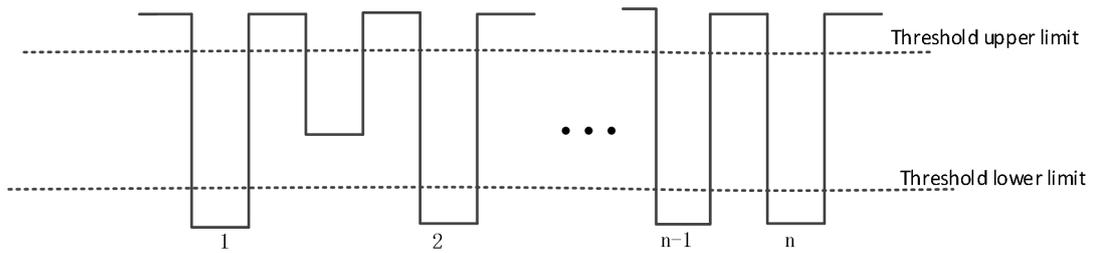
#### 8.1.2..Count Values

1. **[+Pulse Count]**:the number of positive pulses that rise from below the threshold lower limit to above the threshold upper limit.

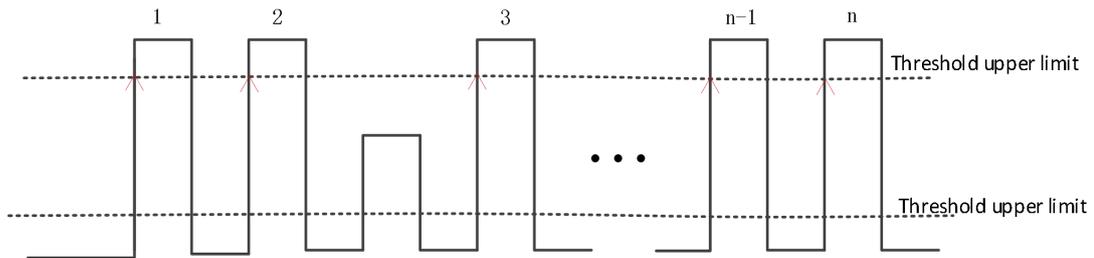


24. **[-Pulse Count]**:he number of negative pulses that fall from above the threshold upper

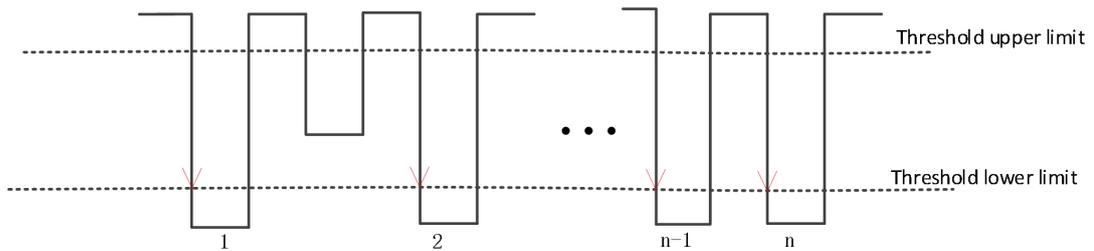
limit to below the threshold lower limit.



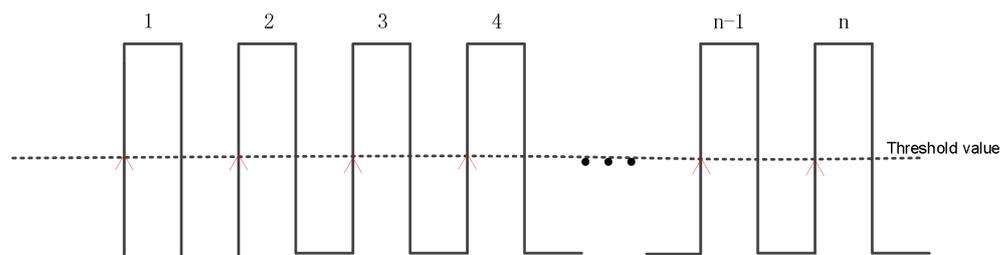
25. **[Rise Pulse Count]**:the number of rising edges that rise from below the threshold lower limit to above the threshold upper limit.



26. **[Fall Pulse Count]**:the number of falling edges that fall from above the threshold upper limit to below the threshold lower limit.

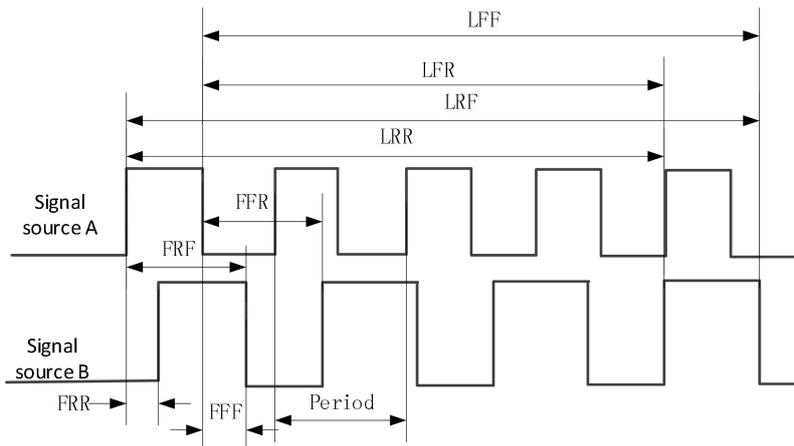


27. **[Trigger Count]**:the number of rising edges above threshold.



**Note:**The above measurement items only apply to analog channels.The default values of the threshold upper limit and threshold lower limit are 90% and 10% respectively.

### 8.1.3.Delay and Phase



Source A and source B, namely source A and source B in the measurement setting menu can be any channel of CH1-CH4 or D0-D15. For the setting method, please refer to the introduction in "**Measurement Setting**".

1. **FRR[First Delay Rise Rise]**:  $\uparrow\mathbf{A} \rightarrow \uparrow\mathbf{B}$ : the time difference between the first rising edges of source A and source B.
2. **FFF[First Delay Fall Fall]**:  $\downarrow\mathbf{A} \rightarrow \downarrow\mathbf{B}$ : the time difference between the first falling edges of source A and source B.
3. **FRF[First Delay Rise Fall]**:  $\uparrow\mathbf{A} \rightarrow \downarrow\mathbf{B}$ : the time difference between the first rising edge of source A and the first falling edge of source B.
4. **FFR[First Delay Fall Rise]**:  $\downarrow\mathbf{A} \rightarrow \uparrow\mathbf{B}$ : the time difference between the first falling edge of source A and the first rising edge of source B.
5. **LRR: [Last Delay Rise Rise]**  $\uparrow\mathbf{A} \rightarrow \uparrow\mathbf{B}$ : the time between the first rising edge of source A and the last rising edge of source B.
6. **LRF[Last Delay Rise Fall]**:  $\uparrow\mathbf{A} \rightarrow \downarrow\mathbf{B}$ : the time between the first rising edge of source A and the last falling edge of source B.
7. **LFR[Last Delay Fall Rise]**:  $\downarrow\mathbf{A} \rightarrow \uparrow\mathbf{B}$ : the time between the first falling edge of source A and the last rising edge of source B.
8. **LFF[Last Delay Fall Fall]**:  $\downarrow\mathbf{A} \rightarrow \downarrow\mathbf{B}$ : the time between the first falling edge of source A and the last falling edge of source B.
9. **[Phase Rise Time]**:  $\uparrow\mathbf{A} \rightarrow \mathbf{B}$ : phase difference calculated according to "**Delay 1 $\rightarrow$ 2**" and the period of source 1, expressed in degree.
10. **[Phase Fall Time]** :  $\downarrow\mathbf{A} \rightarrow \mathbf{B}$ : phase difference calculated according to "**Delay 1 $\rightarrow$ 2**" and the period of source 1, expressed in degree.

The phase calculation formula:

$$\text{Phase} = \frac{\text{Delay}}{\text{period1}} \times 360^\circ$$

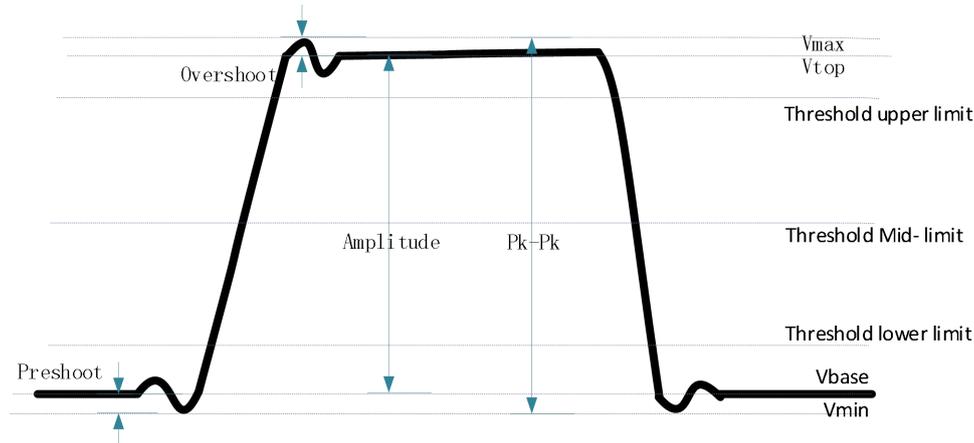
Wherein,

*Phase* represents "Phase $\uparrow\mathbf{A} \rightarrow \mathbf{B}$ " or "Phase $\downarrow\mathbf{A} \rightarrow \mathbf{B}$ ".

*Delay* represents "Delay **FRR**" or "Delay **FFF**".

*Period1* represents the period of data source 1.

#### 8.1.4. Voltage Parameters



1. **Vmax**: the voltage value from the highest point of the waveform to the GND.
2. **Vmin**: the voltage value from the lowest point of the waveform to the GND.
3. **Pk-Pk**: the voltage value from the highest point to the lowest point of the waveform.  
Peak-peak value = maximum value – minimum value
4. **Vtop**: the voltage value from the flat top of the waveform to the GND.
5. **Vbase**: the voltage value from the flat base of the waveform to the GND.
6. **Vampl**: the voltage value from the top of the waveform to the base of the waveform.  
Amplitude = top value – base value
7. **Vmid**: the actual voltage value corresponding to the threshold middle value.
8. **Average**: the arithmetic average value on the whole waveform or on the gating area.
9. **PerAverage**: the arithmetic average value of the first period of the waveform.

$$\text{Average} = \frac{\sum x_i}{n}$$

Wherein,  $x_i$  is the measurement result of the  $i$ th point,  $n$  is the number of points being measured.

10. **Vrms**: the root mean square value on the whole waveform or the gating area.
11. **PerVrms**: the root mean square value within a period.

$$\text{RMS} = \sqrt{\frac{\sum_{i=1}^n x_i^2}{n}}$$

Wherein,  $x_i$  is the measurement result of the  $i$ -th point,  $n$  is the number of points being measured.

12. **Overshoot**: the ratio of the difference of the maximum value and top value of the waveform to the amplitude value.
13. **Preshoot**: the ratio of the difference of the minimum value and base value of the waveform to the amplitude value.
14. **Fall Overshoot**: the ratio of the difference of the maximum value and top value of the waveform to the amplitude value.
15. **Rise Preshoot**: the ratio of the difference of the minimum value and base value of the waveform to the amplitude value.
16. **Variance**: the average of the sum of the squares for the difference between the amplitude value of each waveform point and the waveform average value on the whole waveform or on the gating area.

The variance reflects the fluctuation degree of the waveform. The calculation formula is as follows:

$$\text{Variance} = \frac{\sum_{i=1}^n (Vamp(i) - \text{Average})^2}{n}$$

Wherein,  $Vamp(i)$  is the amplitude of the  $i$ -th point,  $\text{Average}$  is the waveform average value,  $n$  is the number of points being measured.

## 8.2. Grid scale measurement

Grid scale: This method can be used to estimate the frequency and voltage amplitude of

waveform rapidly and intuitively, Simple measurement can be made by the division and scale coefficient of the grid.

For example, a simple measurement can be made by calculating the relevant primary and secondary scales and multiplying by the proportionality coefficient.

If calculated between the maximum and minimum of the waveform there are 6 main vertical scale divisions, and given that the scaling coefficient is 50mV/div, the peak-to-peak voltage can be calculated as follows:

6 divisions×50mV/div=300mV

### 8.3.Auto Measurement

#### Quick Measurement after AUTO

When the oscilloscope is correctly connected and has detected valid input signal, press **AUTO** to set the waveform automatically and open the following function menu:

**Single-period**: set the oscilloscope to display single-period signal automatically. Measure the "Period" and "Frequency" of the current signal within a single period and display the measurement results at the bottom of the screen.

**Multi-period**: set the oscilloscope to display multi-period signal automatically. Measure the "Period" and "Frequency" of the current signal within multiple periods and display the measurement results at the bottom of the screen.

**Auto measuring range**: set the "horizontal vertical scale, horizontal scale and vertical scale"

**Data source**: set the auto channel to "only display", "all".

### 8.4.One-key Measurement of 42 Parameters

Press the corresponding softkey under **MENU** at the left of the screen, to quickly measure 42 waveform parameters and realize "One-key" measurement. The measurement results can be displayed at the bottom of the screen in various font sizes.

The time and voltage parameters icons in the measurement items and the measurement results on the screen are always marked in the same color with the channel (**Measure** → **Source**) currently measured.

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

The oscilloscope automatically executes all computing modes. This measurement uses waveform recording points, which are more accurate than grid and cursor measurements. The displayed readings of auto measurement are periodically updated with new data obtained by the oscilloscope.

Press **Meas** to perform auto measurement. Automatic measurement of 42 parameters is provided, and up to 4 parameters can be displayed at a time.

Perform the following steps, select voltage or time parameters for automatic measurement:

1. Press **Meas** on the frontal panel to enter measurement function menu.
2. Press **signal source** softkey, and rotate V0 to select measuring channel.
3. Press **type** softkey, and rotate V0 to select the desired measuring parameters.
4. Press V0 to add measurement parameters, measurement parameters and measurement values will be displayed in the directory, and the statistical status will be updated in real time.
5. Turn off the statistics function, press statistics softkey and select turn off.

The measuring display area can display up to 4 measuring parameters and carry out measurement according to the selection sequence. If the fifth measurement parameter is added, the first measurement parameter will be deleted.

**Note**: if the parameters do not match the measurement conditions, "\*\*\*\*\*" will be displayed.

#### To Clear the Measurement parameters

Press clear all softkey to clear all the measurement parameters displayed on the screen.

#### Statistics function

Count and display the current value, average value, minimum value, maximum value and

root mean square error of up to 4 open measurement items.

1. Press **Meas** on the frontal panel to enter measurement function menu.
2. press **statistic** softkey and select open.

### All Measurement

All measurement could measure all the parameters of the current signal source and display the results on the screen.

Follow these steps to measure all parameters:

1. Press **Meas** on the frontal panel to enter measurement function menu.
2. press **all measurement** softkey and select open.
3. press **signal source** softkey to select the measurement signal source(CH1~CH4).

## 8.5.Cursor Measurement

**Cursors:**This method allows the user to make measurements by moving the cursor. Cursors always appear in pairs, and the readings displayed are the measured values. There are two types of cursors: amplitude cursor and time cursor.

Amplitude cursor: the amplitude cursor is displayed as a horizontal dotted line to measure the parameters in the vertical direction.

Time cursor: the time cursor is displayed as a vertical dotted line to measure the parameters in the horizontal direction.

**The cursor measurement includes two modes: Manual mode and Tracking mode.**

### 1.Manual mode:

Horizontal cursors or vertical cursors appear in pair to measure time or voltage, and the position of the cursors can be manually regulated. The signal source should be set as a waveform to be measured before the cursors are used.

### 2.Tracking mode:

A horizontal cursor is intersected with a vertical cursor to form a cross cursor. The cross cursor is automatically located on the waveform, and the horizontal position of the cross cursor on the waveform is regulated by selecting "Source A" or "Source B" and rotating the V0 knob. The coordinates of the cursor point will be displayed on the screen of the oscilloscope.

Press **【CURSOR】** to enter the cursor menu

Options	Settings	Annotation
Mode	manual tracking	Select a measuring cursor and display.
Data Source	CH1~CH4 MATH	Select a signal source and measure by cursor.
Select cursor	AX(BX) AXBX AY(BY) AYBY	The selected highlight cursor can move freely.You can select 2 cursors and move them at the same time.The frame behind the cursor display the position of the cursor.

**Move the cursors:**Press the key near the select cursor and rotate V0 to move the cursor.The cursor can only be moved when the cursor menu is open.

## 8.6.Digital Voltage Meter(DVM)

DVM is mainly used for voltage and frequency measurement,and monitoring the signal when the oscilloscope is running or stopping.

DVM has 5 digits AC RMS、DC、AC+DC RMS voltage measurement and 6 digits frequency measurement.

### 8.6.1.Enable the digital voltage meter

Press Meas→ DVM→enable **【on,off】** on the frontal panel.

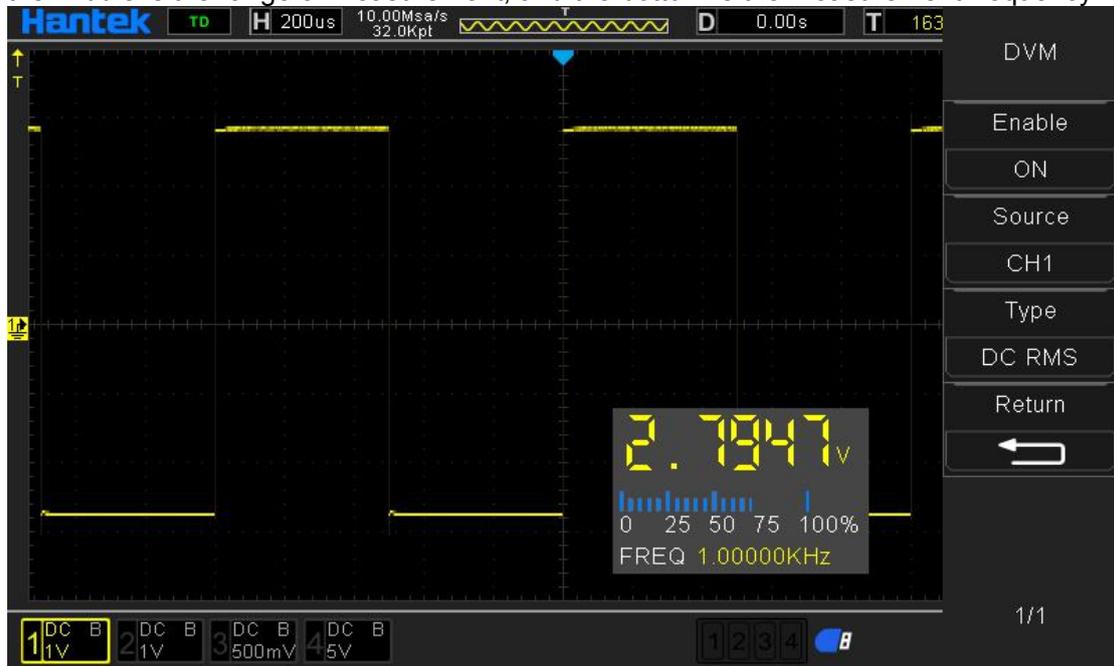
### 8.6.2.Set the data source

Press Meas→ DVM→data source **【CH1,CH2,CH3,CH4】** on the frontal panel.

### 8.6.3.Set types

Press Meas → DVM → types 【AC RMS,DC RMS,DC】 ON the frontal panel.

After DVM measurement is turned on, the following figure is displayed. DVM display results include the range, which is determined by the vertical tap position and vertical offset of the channel. The range is the screen range. The above is the measurement result, the middle is the range of measurement, and the bottom is the measurement frequency.

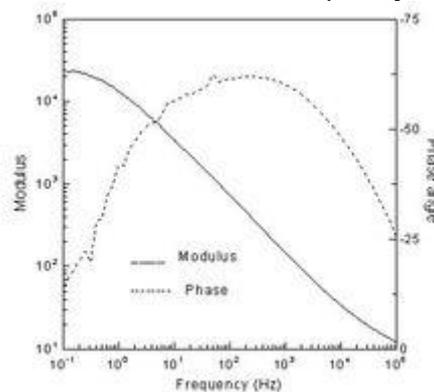


## 9. Baud diagram (the modes with signal source)

The bode diagram is a semi-log coordinate graph of the transfer function of a linear time-invariant system with respect to frequency, its horizontal axis is frequency, The vertical axis is represented on a log scale. The frequency response of the system can be observed by using the bode diagram. Also known as amplitude frequency response and phase frequency response curve graph. A bode diagram is usually a combination of two diagrams, An amplitude-frequency diagram shows the change in the decibel value of the frequency response gain with respect to the frequency, and a phase-frequency diagram shows the change in the phase of the frequency response with respect to the frequency. The figure of the bode diagram is related to the gain of the system, the number and position of poles and zeros, As long as you know the relevant information, with a simple calculation can draw an approximate bode diagram, this is the advantage of using the bode diagram.

### 9.1. The sketch of the baud diagram

Baud diagram is also known as amplitude frequency response and phase frequency response curve graph, Generally, it is the rectangular coordinate of amplitude and phase relative to rotor speed on the fundamental frequency of rotating machinery.



Baud diagram

The logarithmic frequency characteristic is drawn by the method of polyline approximation. The general drawing method of the bode diagram:

When drawing the baud diagram, it is divided into three frequency bands. The first is the amplitude-frequency characteristic, and the order is the middle frequency band, the low frequency band and the high frequency band. Combining the frequency characteristics (or frequency response) of the three frequency bands to form the amplitude-frequency characteristics of the full frequency band, and then the corresponding phase-frequency characteristics are drawn according to the amplitude-frequency characteristics.

### 9.2. The application of baud diagram

When studying the frequency response of amplification circuit, due to the wide frequency range of signal (from a few Hz to above a few hundred MHz), the amplification factor of amplification circuit is also very large (up to one million times). To compress the coordinate and expand the field of vision, when drawing the frequency characteristic curve, the frequency coordinate adopts exponential scale, while the amplitude (in dB) or phase angle adopts logarithmic scale. The amplitude-frequency characteristics and phase-frequency curves drawn in such semi-logarithmic coordinates are called logarithmic frequency characteristics or baud diagram.

The data in the baud diagram contains new (blue) data and old (green) data. The "sampling noise" in the traditional system can be seen from the diagram.

When analyzing the stability of the negative feedback amplifier circuit in the course of analog electronic technology, colleges and universities usually adopt the baud diagram analysis method.

### 9.3.Enable baud diagram

Press Utility→ DVM→enable 【on,off】 on the frontal panel.

### 9.4.Source of the baud diagram setting

Press Utility→ baud diagram→setting on the frontal panel

The following settings can be completed by source of the baud diagram setting.

Input source can set CH1-CH4.

Output source can set CH1-CH4.

**Note:**

The input and output sources are mutually exclusive.

The minimum value of minimum frequency is 100 MHZ.

The maximum value of maximum frequency is 20 MHZ.

Amplitude 10mV~7V.

### 9.5.The diagram setting of the baud diagram

Press Utility→ baud diagram→diagram on the frontal panel.

Gain scale—>Set the grid step value of the gain.

Gain offset—>Set the grid central value of the gain.

Phase scale—>Set the grid step value of the phase.

Phase offset—>Set the grid central value of the gain.

### 9.6.Operation of the baud diagram

Press Utility→ baud diagram→operation on the frontal panel.

## 10. Digital Channel

MPO 6000 series oscilloscopes provide 4 analog channels as well as 16 digital channels. DPO6000 series oscilloscopes can be upgraded to use 16 digital channels by buying LP104 digital probe. For the digital channels, the oscilloscope compares the voltages acquired in each sample with the preset logic threshold. If the voltage of the sample point is greater than the logic threshold, it will be stored as logic 1; otherwise, it will be stored as logic 0. The oscilloscope displays the logic 1 and logic 0 in figure form for users to easily detect and analyze the errors in circuit design (hardware design and software design). This chapter introduces how to use the digital channels of the mixed signal digital oscilloscopes. Before using the digital channels, connect the oscilloscope and the device under test using LP104 logic probe provided in the accessories. The digital channel input terminal support hot plug 【plug and play】 and the insertion interface has no direction, which is easy for users to insert digital probes.

### 10.1. To Select the Digital Channel

Press Utility → logic analyzer → data source 【D1, D2, D3, D4】 on the frontal panel.

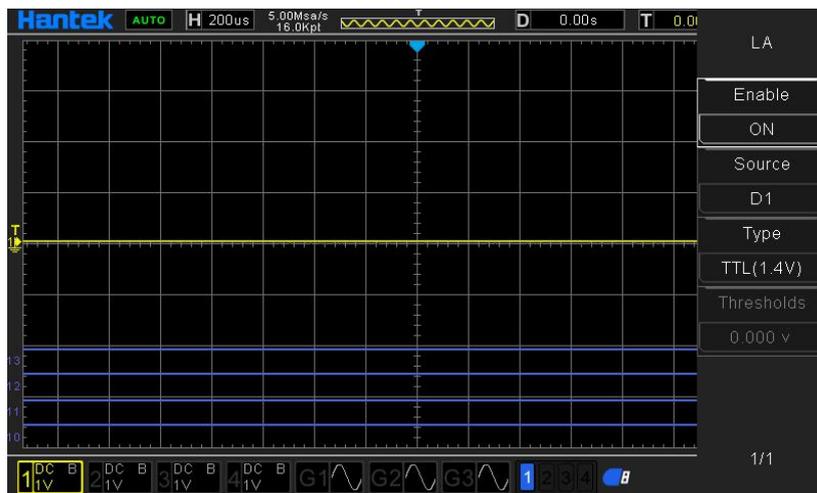
### 10.2. To Turn on the Digital Channel

Connect the digital probe, the logic analyzer enables the switch to open automatically, the corresponding digital channel at the bottom of the screen shows blue, indicating that it has been connected. Meanwhile, the channel indicator light on the logic probe is on.

**Note:** the digital channel cannot be suspended, only after access to the logic analyzer probe to control its enable.

For digital channel input, refer to the description of channel input in the technical specifications section of this article.

Note: the number of channels simultaneously opened by digital channel and analog channel shall not exceed 4, otherwise "The Channel Can Not Be Open Now" is displayed.



### 10.3. To set the gate line type of the digital channel

Press Utility → logic analyzer → types 【TTL(1.4 V)】

- 5.0 V CMOS(+2.5 V)
- 3.3 V CMOS(+1.65 V)
- 2.5 V CMOS(+1.25 V)
- 1.8 V CMOS(+0.9 V)
- ECL(-1.3 V)

PECL(+3.7 V)

LVDS(+1.2 V)

0V

User defined **【-7V~7V】** on the frontal panel.

## 11. Waveform generator

DPO6000C/ MPO6000D built-in 2 channels(MPO6000EDU built-in 3 channels ),25MHZ signal source combines the signal source and oscilloscope,which greatly facilitates the engineers who need to use both signal source and oscilloscope.The built-in signal source can output 13 kinds of basic waveform, including sine, square, pulse, triangle,noise,DC, sinc,exponential, half-distortion, Lorentz, dual tone and multi-frequency, Gauss, Ecg, meanwhile provides 4 arbitrary waveform.Users can edit,load and output the arbitrary waveform through WaveEditor software.

This chapter will introduce how to use the oscilloscope built-in signal source.Since the functions and setting methods of two or three channels signal source are the same, this chapter takes source 1 as an example to illustrate.

Press/click on WaveGen area on the screen to enter the control menu of the signal source.

### 11.1. Output basic waveform

#### 11.1.1. To Output Sine

##### 1. Select waveform

Press WaveGen → waveform → sine on the frontal panel.

##### 2. Select the data source

Press WaveGen → data source on the frontal panel.

##### 3. Turn on/off the channel of signal source

Press WaveGen → enable on the frontal panel.

##### 4. Set the frequency

Press WaveGen → frequency on the frontal panel.

**Note:**For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

##### 5. Set the amplitude

Press WaveGen → amplitude on the frontal panel Press amplitude softkey to set the amplitude of the current signal.When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

##### 6. Set the offset voltage

Press WaveGen → offset on the frontal panel.

**Note:**When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to (3.5V - the current amplitude/2); when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

##### 7. Set the phase

Press WaveGen → phase on the frontal panel.

The range is from 0° to 360°.

##### 8. Set the load impedance

Press WaveGen → Load impedance on the frontal panel to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

##### 9. Align phase

Press WaveGen → Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

## 10.Modulation

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface. For detailed introduction of the modulation function, please refer to the section "**Modulation**".

## 11.Burst

Press WaveGen → Burst on the frontal panel. For detailed introduction of the burst function, please refer to the section "**Burst**".

### 11.1.2.To Output Square

#### 1.Select waveform

Press WaveGen→ waveform→square on the frontal panel.

#### 2.Select the data source

Press WaveGen→data source on the frontal panel.

#### 3.Turn on/off the channel of signal source

Press WaveGen-→enable on the frontal panel.

#### 4.Set the frequency

Press WaveGen →frequency on the frontal panel.

**Note:** For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

#### 5.Set the amplitude

Press WaveGen →amplitude on the frontal panel.

Press amplitude softkey to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**".When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 6.Set the offset voltage

Press WaveGen →offset on the frontal panel.

**Note:**When the impedance is High Z, the range is from  $(-3.5\text{ V} + \text{the current amplitude}/2)$  to  $(3.5\text{ V} - \text{the current amplitude}/2)$ ;when the impedance is set to 50 Ω, the range is from  $(-1.75\text{ V} + \text{the current amplitude}/2)$  to  $(1.75\text{ V} - \text{the current amplitude}/2)$ .

#### 7.Set the phase

Press WaveGen →phase on the frontal panel.

The range is from 0° to 360°

#### 8.set the duty

The duty cycle of the square wave represents the amount of time in each cycle when the waveform is at a high level.At low frequencies the range is 0.001% to 99.996%, and at higher frequencies the range is narrowed.

#### 9.Set the load impedance

Press WaveGen → impedance on the frontal panel to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

## 10.Align phase

Press WaveGen →Align phase on the frontal panel.

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed.

At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

## 11.Modulation

Press WaveGen → Modulation on the frontal panel for detailed introduction of the modulation function, please refer to the section "**Modulation**".

## 12.Burst

Press WaveGen → Burst on the frontal panel. For detailed introduction of the burst function, please refer to the section "**Burst**".

### 11.1.3.To Output Ramp

#### 1.Select waveform

Press WaveGen→ waveform→ramp on the frontal panel.

#### 2.Select the data source

Press WaveGen→ data source.

#### 3.Turn on/off the channel of signal source

Press WaveGen-→enable on the frontal panel.

#### 4.Set the frequency

Press WaveGen →frequency on the frontal panel.

#### Note:

For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

#### 5.Set the amplitude

Press amplitude softkey to to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**".When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 6.Set the offset voltage

Press WaveGen →offset on the frontal panel.

**Note:**When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to(3.5V - the current amplitude/2);when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

#### 7.Set the phase

Press WaveGen →phase on the frontal panel.

The range is from 0°to 360°

## 8.Symmetry

Symmetry only applies to triangular waves.Symmetry represents the amount of time that the triangular wave is rising in each period.

## 9.Set the load impedance

Press WaveGen → impedance on the frontal panel

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

## 10.Align phase

Press WaveGen →Align phase on the frontal panel.

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase.For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed.At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

## 11.Modulation

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface.For detailed introduction of the modulation function, please refer to the section "**Modulation**".

**Note:**when select the waveform to be "pulse" "DC" or "noise",there is no such setting.

## 12.Burst

Press WaveGen → Burst on the frontal panel. For detailed introduction of the burst function, please refer to the section "**Burst**".

### 11.1.4.To Output noise

#### 1.Select waveform

Press WaveGen→ waveform→noise on the frontal panel.

#### 2.Select the data source

Press WaveGen→ data source.

#### 3.Turn on/off the channel of signal source

Press WaveGen-→enable on the frontal panel.

#### 4.Set the amplitude

Press WaveGen →amplitude on the frontal panel.

Press amplitude softkey to to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**".When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 5.Set the offset voltage

Press WaveGen →offset on the frontal panel

**Note:**When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to(3.5V - the current amplitude/2);when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

#### 6.Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

## 7.Align phase

Press WaveGen →Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase.For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed.At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

## 8.Burst

Press WaveGen →Burst on the frontal panel. For detailed introduction of burst function, please refer to the section "**Burst**".

### 11.1.5.To Output Pulse

#### 1.Select waveform

Press WaveGen→ waveform→ pulse on the frontal panel.

#### 2.Select the data source

Press WaveGen→ data source.

#### 3.Turn on/off the channel of signal source

Press WaveGen-→enable on the frontal panel.

#### 4.Set the frequency

Press WaveGen →frequency on the frontal panel.

**Note:**For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

#### 5.Set the amplitude

Press WaveGen →amplitude on the frontal panel.

Press amplitude softkey to to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**".When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 6.Set the offset voltage

Press WaveGen →offset on the frontal panel.

**Note:**When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to(3.5V - the current amplitude/2);when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

#### 7.Set the phase

Press WaveGen →phase on the frontal panel .

The range is from 0°to 360°

#### 8.set the width

Set the pulse width.

#### 9.set the leading

Use the V0 knob to select the leading or trailing edge.

## 10. Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

## 11. Align phase

Press WaveGen → Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase.

For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed.

At this point, press AlignPhase softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

### 11.1.6. To output DC

#### 1. Select waveform

Press WaveGen → waveform → DC on the frontal panel.

#### 2. Select the data source

Press WaveGen → data source.

#### 3. Turn on/off the channel of signal source

Press WaveGen → enable on the frontal panel.

#### 4. Set the offset voltage

Press WaveGen → offset on the frontal panel

**Note:** When the impedance is High Z, the range is from  $(-3.5 \text{ V} + \text{the current amplitude}/2)$  to  $(3.5 \text{ V} - \text{the current amplitude}/2)$ ; when the impedance is set to 50 Ω, the range is from  $(-1.75 \text{ V} + \text{the current amplitude}/2)$  to  $(1.75 \text{ V} - \text{the current amplitude}/2)$ .

#### 5. Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

#### 6. Align phase

Press WaveGen → Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

### 11.1.7. To Output Sinc

#### 1. Select waveform

Press WaveGen → waveform → Sinc on the frontal panel.

## 2. Select the data source

Press WaveGen → data source.

## 3. Turn on/off the channel of signal source

Press WaveGen → enable on the frontal panel.

## 4. Set the frequency

Press WaveGen → frequency on the frontal panel.

**Note:** For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

## 5. Set the amplitude

Press WaveGen → amplitude on the frontal panel.

Press amplitude softkey to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**". When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

## 6. Set the offset voltage

Press WaveGen → offset on the frontal panel.

**Note:** When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to (3.5V - the current amplitude/2); when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

## 7. Set the phase

Press WaveGen → phase on the frontal panel .

The range is from 0° to 360°.

## 8. Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

## 9. Align phase

Press WaveGen → Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

## 10. Modulation

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface. For detailed introduction of the modulation function, please refer to the section "**Modulation**".

**Note:** when select the waveform to be "pulse" "DC" or "noise", there is no such setting.

## 11. Burst

Press WaveGen → Burst on the frontal panel.

For detailed introduction of burst function, please refer to the section "**Burst**".

### 11.1.8. To Output Index

### 1.Select waveform

Press WaveGen→ waveform→ index on the frontal panel.

### 2.Select the data source

Press WaveGen→ data source.

### 3.Turn on/off the channel of signal source

Press WaveGen-→enable on the frontal panel.

### 4.Set the frequency

Press WaveGen →frequency on the frontal panel.

**Note:** For different waveform,the frequency ranges are different. See oscilloscope technical specifications for details.

### 5.Set the amplitude

Press WaveGen →amplitude on the frontal panel.

Press amplitude softkey to to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**".When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

### 6.Set the offset voltage

Press WaveGen →offset on the frontal panel.

**Note:**When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to (3.5V - the current amplitude/2);when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

### 7.Set the phase

Press WaveGen →phase on the frontal panel .

The range is from 0°to 360°

### 8.Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

### 9.Align phase

Press WaveGen →Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

### 10.Modulation

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface.For detailed introduction of the modulation function, please refer to the section "**Modulation**".

**Note:**when select the waveform to be "DC" , "noise" or "Sinc",there is no such setting.

### 11.Burst

Press WaveGen →Burst on the frontal panel.

For detailed introduction of burst function, please refer to the section "**Burst**".

### 11.1.9.To Output Semi-distortion

#### 1.Select waveform

Press WaveGen → waveform → semi-distortion on the frontal panel.

#### 2.Select the data source

Press WaveGen → data source.

#### 3.Turn on/off the channel of signal source

Press WaveGen → enable on the frontal panel.

#### 4.Set the frequency

Press WaveGen → frequency on the frontal panel.

**Note:** For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

#### 5.Set the amplitude

Press WaveGen → amplitude on the frontal panel.

Press amplitude softkey to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**". When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 6.Set the offset voltage

Press WaveGen → offset on the frontal panel.

**Note:** When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to (3.5V - the current amplitude/2); when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

#### 7.Set the phase

Press WaveGen → phase on the frontal panel .

The range is from 0° to 360°

#### 8.Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

#### 9.Align phase

Press WaveGen → Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

#### 10.Modulation

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface. For detailed introduction of the modulation function, please refer to the section "**Modulation**".

**Note:** when select the waveform to be "pulse" "DC" or "noise", there is no such setting.

## 11. Burst

Press WaveGen → Burst on the frontal panel.

For detailed introduction of burst function, please refer to the section "**Burst**".

### 11.1.10. To Output Lorentz

#### 1. Select waveform

Press WaveGen → waveform → lorentz on the frontal panel.

#### 2. Select the data source

Press WaveGen → data source.

#### 3. Turn on/off the channel of signal source

Press WaveGen → enable on the frontal panel.

#### 4. Set the frequency

Press WaveGen → frequency on the frontal panel.

**Note:** For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

#### 5. Set the amplitude

Press WaveGen → amplitude on the frontal panel.

Press amplitude softkey to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**". When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 6. Set the offset voltage

Press WaveGen → offset on the frontal panel.

**Note:** When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to (3.5 V - the current amplitude/2); when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

#### 7. Set the phase

Press WaveGen → phase on the frontal panel .

The range is from 0° to 360°

#### 8. Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

#### 9. Align phase

Press WaveGen → Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

#### 10. Modulation

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface. For detailed introduction of the modulation function, please refer to the section

**"Modulation".**

**Note:**when select the waveform to be "pulse" "DC" or "noise",there is no such setting.

**11.Burst**

Press WaveGen →Burst on the frontal panel.

For detailed introduction of burst function, please refer to the section "**Burst**".

**11.1.11.To Output Dual tone multiple frequency****1.Select dual tone multiple frequency**

Press WaveGen→ waveform→ dual tone multiple frequency on the frontal panel.

**2.Select the channel of the signal source**

Press WaveGen→ data source.

**3.Turn on/off the channel of signal source**

Press WaveGen-→enable on the frontal panel.

**4.Set the frequency**

Press WaveGen →frequency on the frontal panel.

**Note:** For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

**5.Set the amplitude**

Press WaveGen →amplitude on the frontal panel.

Press amplitude softkey to to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**".When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

**6.Set the offset voltage**

Press WaveGen →offset on the frontal panel.

**Note:**When the impedance is High Z, the range is from  $(-3.5\text{ V} + \text{the current amplitude}/2)$  to  $(3.5\text{ V} - \text{the current amplitude}/2)$ ;when the impedance is set to 50 Ω, the range is from  $(-1.75\text{ V} + \text{the current amplitude}/2)$  to  $(1.75\text{ V} - \text{the current amplitude}/2)$ .

**7.Set the phase**

Press WaveGen →phase on the frontal panel .

The range is from 0°to 360°

**8.Set the load impedance**

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

**9.Align phase**

Press WaveGen →Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

**10.Modulation**

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface. For detailed introduction of the modulation function, please refer to the section "**Modulation**".

**Note:** when select the waveform to be "pulse" "DC" or "noise", there is no such setting.

## 11. Burst

Press WaveGen → Burst on the frontal panel.

For detailed introduction of burst function, please refer to the section "**Burst**".

### 11.1.12. To Output Gauss

#### 1. Select waveform

Press WaveGen → waveform → gauss on the frontal panel.

#### 2. Select the data source

Press WaveGen → data source.

#### 3. Turn on/off the channel of signal source

Press WaveGen → enable on the frontal panel.

#### 4. Set the frequency

Press WaveGen → frequency on the frontal panel.

**Note:** For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

#### 5. Set the amplitude

Press WaveGen → amplitude on the frontal panel.

Press amplitude softkey to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**". When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 6. Set the offset voltage

Press WaveGen → offset on the frontal panel.

**Note:** When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to (3.5 V - the current amplitude/2); when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

#### 7. Set the phase

Press WaveGen → phase on the frontal panel .

The range is from 0° to 360°

#### 8. Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

#### 9. Align phase

Press WaveGen → Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

## 10.Modulation

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface. For detailed introduction of the modulation function, please refer to the section "**Modulation**".

**Note:** when select the waveform to be "pulse" "DC" or "noise", there is no such setting.

## 11.Burst

Press WaveGen → Burst on the frontal panel.

For detailed introduction of burst function, please refer to the section "**Burst**".

### 11.1.13.To Output ECG signal

#### 1.Select waveform

Press WaveGen → waveform → Ecg on the frontal panel.

#### 2.Select the data source

Press WaveGen → data source.

#### 3.Turn on/off the channel of signal source

Press WaveGen → enable on the frontal panel.

#### 4.Set the frequency

Press WaveGen → frequency on the frontal panel.

**Note:** For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

#### 5.Set the amplitude

Press WaveGen → amplitude on the frontal panel.

Press amplitude softkey to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**". When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 6.Set the offset voltage

Press WaveGen → offset on the frontal panel.

**Note:** When the impedance is High Z, the range is from  $(-3.5 \text{ V} + \text{the current amplitude}/2)$  to  $(3.5 \text{ V} - \text{the current amplitude}/2)$ ; when the impedance is set to 50 Ω, the range is from  $(-1.75 \text{ V} + \text{the current amplitude}/2)$  to  $(1.75 \text{ V} - \text{the current amplitude}/2)$ .

#### 7.Set the phase

Press WaveGen → phase on the frontal panel .

The range is from 0° to 360°

#### 8.Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

#### 9.Align phase

Press WaveGen → Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase

between the two waveform automatically.

## 10.Modulation

Press WaveGen → Modulation on the frontal panel to enter the modulation setting interface.For detailed introduction of the modulation function, please refer to the section "**Modulation**".

**Note:**when select the waveform to be "pulse" "DC" or "noise",there is no such setting.

## 11.Burst

Press WaveGen →Burst on the frontal panel.

For detailed introduction of burst function, please refer to the section "**Burst**".

### 11.1.14.To Output Arbitrary Waveform

#### 1.Select waveform

Press WaveGen→ waveform→ Arbitrary on the frontal panel.

#### 2.Select the data source

Press WaveGen→ data source.

#### 3.Turn on/off the channel of signal source

Press WaveGen-→enable on the frontal panel.

#### 4.Set the frequency

Press WaveGen →frequency on the frontal panel.

**Note:** For different waveform, the frequency ranges are different. See oscilloscope technical specifications for details.

#### 5.Set the amplitude

Press WaveGen →amplitude on the frontal panel.

Press amplitude softkey to to set the amplitude of the current signal. For the setting method, please refer to the section "**Parameter Setting Method**".When the impedance is set to High Z, the range is from 20 mVpp to 7 Vpp; when the impedance is set to 50 Ω, the range is from 10 mVpp to 3.5 Vpp.

#### 6.Set the offset voltage

Press WaveGen →offset on the frontal panel.

**Note:**When the impedance is High Z, the range is from (-3.5 V + the current amplitude/2) to (3.5V - the current amplitude/2);when the impedance is set to 50 Ω, the range is from (-1.75 V + the current amplitude/2) to (1.75 V - the current amplitude/2).

#### 7.Set the phase

Press WaveGen →phase on the frontal panel .

The range is from 0°to 360°

#### 8.Set the load impedance

Press WaveGen → impedance on the frontal panel.

Press impedance softkey to set the load input impedance of the signal generator. It can be set to "High Z" or "50Ω".

#### 9.Align phase

Press WaveGen →Align phase on the frontal panel .

The channels of the signal source will be re-configured after pressing Align phase to output according to the preset frequency and phase. For two signals of which the frequencies are the same or are multiples, this operation can align their phases.

The oscilloscope acquires the waveform of the two channels and displays the waveform stably. After switching the status of the channel switch, the phase deviation between the

two waveform is changed. At this point, press **AlignPhase** softkey and the waveform shown on the oscilloscope will restore to the phase deviation of the current phase between the two waveform automatically.

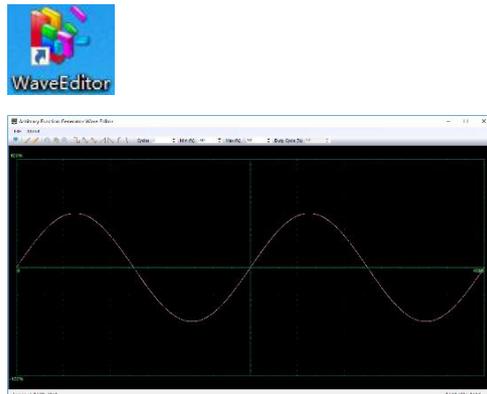
## 9.Recall

Press WaveGen → recall on the frontal panel.

Enter the call out menu, you can call out waveform Settings.

## 11.2.Edit Arbitrary Waveform

User can't edit arbitrary waveform directly under **WaveGen** interface for the device. Firstly, please double click "WaveEditor Setup.exe" in WaveEditor folder in CD disk and install Arbitrary Waveform Editor software according to installation wizard. After the software installed successfully, you can see **WaveEditor** icon on desktop. Double-click the icon to enter arbitrary waveform generator window.



### Menu:

**Import from CSV:** Import the CSV format file to the arbitrary waveform generator window.

**Export as CSV:** Save as CSV format file.

**Import from ARB:** Import the ARB format file to the arbitrary waveform generator window.

**Export as ARB:** Save as ARB format file.

**Note:** The device can recall ARB format file in USB disk, but CSV format file can't be recalled by the device.

### Toolbar buttons

 : Download waveform data to the device.

 : **Smooth Drawing Mode.** You can draw any waveform shape using the left-mouse.

 : **Line Drawing Mode.** You can click on the waveform to draw a straight line from the previous point.

 : **Zoom tools.** To zoom the time axis in or out, click the + or - zoom button and then click on the waveform area. Click the **100%** button to restore the time axis to its original scale.

 : **Standard waveform shapes.** Draw a standard waveform with the settings specified in the numerical controls below the toolbar.

 : **Cycles.** The number of cycles to draw. This control is used in conjunction

with the **Standard waveform shapes buttons**. Select one of the standard waveform shapes and then set the number of cycles, and it will draw the requested number of cycles of the waveform.

**Min (%) -50** : **Minimum**. When one of the **Standard waveform shapes buttons** is pressed, this control sets the minimum signal level.

**Max (%) 50** : **Maximum**. When one of the **Standard waveform shapes buttons** is pressed, this control sets the maximum signal level.

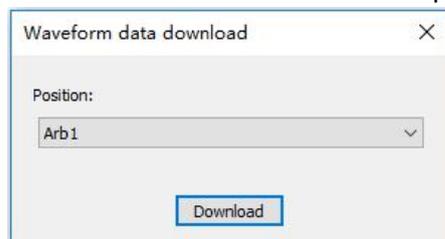
**Duty Cycle (%) 50** : **Duty cycle**. When a square, triangular or ramp waveform is selected using one of the **Standard waveform shapes buttons**, this control sets the duty cycle of the signal. Duty cycle is defined as the time that the signal spends above zero volts divided by the total cycle time. Thus, a symmetrical square or triangular wave has a duty cycle of 50%. Reducing the duty cycle shortens the positive part of the cycle and lengthens the negative part.

**Note:** The **Frequency, Amplitude, Offset** parameter of the ARB waveform can not be regulated in this WaveEditor software, but it can be done by tuning on the device directly after the waveform data is downloaded to device .

Do not use the WaveEditor and the DSO software at the same time, it will cause errors.

### Output Arbitrary Waveform

1. Press the **【 Wave Gen 】** button on the front panel to turn on arbitrary waveform generator function and enter waveform generator function menu.
2. Connect the device with PC which has installed the WaveEditor software using a USB cable.
3. Double click the WaveEditor icon to open the software.
4. Select a waveform file or draw an arbitrary waveform; then click the  in toolbar and select a waveform data download position to download waveform data to the device.



5. The waveform will output from GEN OUT BNC port.

**Also, you can recall ARB format file in USB disk to output a waveform.**

1. Press the **【 Wave Gen 】** button on the front panel to enter waveform generator function menu.
2. Press **waveform** softkey and rotate V0 to select Arb1~Arb4, then press V0 to confirm.
3. Press **recall** softkey and select desired recalled ARB format file in USB disk.

4. The waveform will output from GEN OUT BNC port.

## 12.Modulation

The built-in signal generator of DPO6000/MPO6000 series oscilloscope supports AM,FM,PM.PWM modulation function. The modulated waveform consists of the carrier waveform and modulating waveform. The carrier waveform is the waveform output from the signal generator and the modulating signal can be the built-in sine, square, sawtooth waveform.

### 12.1.AM

Amplitude Modulation( AM)namely the amplitude of the carrier waveform changes with the amplitude of the modulating waveform.

#### 1.Enable the modulation fuction

Press WaveGen→ modulation→modulation on the frontal panel

#### 2.Set AM modulation

Press WaveGen→ modulation→types→ A M

#### 3.Set the Modulating Waveform

Press WaveGen→ modulation→waveform sin,square,triangular.

#### 4.Set the Modulation Frequency

Press WaveGen→ modulation→modulation frequency

#### 5.Set the Modulation Depth

Press WaveGen→ modulation→modulation depth

### 12.2.FM

FM (Frequency Modulation), namely the frequency of the carrier waveform changes with that of the modulating waveform.

#### 1.Enable the modulation fuction

Press WaveGen→ modulation→modulation on the frontal panel

#### 2.Set FM modulation

Press WaveGen→ modulation→types→FM

#### 3.Set the Modulating Waveform

Press WaveGen→ modulation→waveform sin,square,triangular.

#### 4.Set the Modulation Frequency

Press WaveGen→ modulation→modulation frequency

#### 5.Set the Modulation Offset

Press WaveGen→ modulation→modulation offset

### 12.3.1Phase

Modulated waveform generally consists of carrier waveform and modulation waveform. For PM (Phase Modulation),the phase of carrier waveform changes with the transient voltage of modulation waveform.

#### 1.Enable the modulation fuction

Press WaveGen→ modulation→modulation on the frontal panel

#### 2.Set PM modulation

Press WaveGen→ modulation→types→PM

**3.Set the Modulating Waveform**

Press WaveGen→ modulation→waveform sin,square,triangular.

**4.Set the Modulation Frequency**

Press WaveGen→ modulation→modulation frequency

**5.Set the Modulation Offset**

Press WaveGen→ modulation→modulation offset

**12.4.Duty Modulation**

Modulated waveform of PWM(Pulse Width Modulation)consists of carrier waveform and modulation waveform. the pulse width of carrier waveform changes with the transient voltage of modulation waveform.

**1.Enable the modulation fuction**

Press WaveGen→ modulation→modulation on the frontal panel

**2.Set PWM modulation**

Press WaveGen→ modulation→types→PM

**3.Set the Modulating Waveform**

Press WaveGen→ modulation→waveform sin,square,triangular.

**4.Set the Modulation Frequency**

Press WaveGen→ modulation→modulation frequency

**5.Set the Modulation Depth**

Press WaveGen→ modulation→modulation depth

## 13.Burst

DPO6000/MPO6000 can output waveform with a specified number of cycles from a single channel or two channels at the same time. When DPO6000/MPO6000 trigger, manual trigger source control pulse string output.

### 1.Enable Burst Function

Press WaveGen→ burst on the frontal panel

### 2.Set the Count Value

Press WaveGen→ burst→count on the frontal panel

### 3.Set data Source

Press WaveGen→ burst→data source on the frontal panel

#### **Manual:**

At this time, "trigger" button is optional. Press "trigger" once and the signal source outputs a waveform with a set number of cycles.

#### **Pass/Fail:**

Run the [pass/fail] test, open "output" and select "output to DDS". After the waveform is output according to the set output mode, specify the waveform with the set number of cycles output by the signal source.

#### **DSO Trigger:**

Each time the waveform in an oscilloscope is triggered, the current signal source outputs a waveform with a set number of cycles.

## 14.Pass/Fail Test

The template test function is mainly used for the evaluation of signal quality, by setting a template in the oscilloscope to test the waveform in real time, count the number of test times, the number of failed data frames, the failure rate, the test time and so on in the specified test conditions, and display test results and output test data. Testing can save time and cost in the design and manufacturing process, and ensure that customers can receive higher quality products faster.

### 14.1.To Enable Pass/Fail Test

Press Utility →Pass/Fail →On to turn on test, The shadow area denotes the failure area, and if the waveform measured at certain time point during the measurement process passes through the failure area, it will be treated as failed frame.

**Note:** When the time base is set to X-Y, ROLL or the horizontal time base is set to 200 ms/div or slower under YT mode and the instrument enter the "slow sweep" mode, the pass/fail test function is not available.

When the test function is enabled, you can set the output form of the test results. For the details, please refer to the introduction in "**Test and Output**".You can also select the signal source, set the test rule range as well as save and recall the test rule. For the details, refer to the introductions below.

#### To Select Source

Before selecting the source, you need to connect the signal under test to the analog channel input terminal of the oscilloscope. Press **Source** softkey to select the channel (CH1-CH4) to be tested. **Note:** Only channels enabled can be selected.

#### Rule Range

Users can define their desired test rule range.Press **horizontal** and **vertical** softkey respectively,rotate V0 to adjust the pass/fail rule range. Press **Create** softkey to apply the rule currently created.

#### Test and Output

You can set the output form of the test results by the following method.

Press **Message** softkey to select "ON" or "OFF". When "ON" is selected, the test results will be displayed at the lower left corner of the screen.

Press **Stop** softkey to select "ON" or "OFF".

**ON:** When the waveform that conforms to the set "mode" is detected, if **Aux "output"** is selected as "on", the back panel [**Trigger Out**] will output a pulse. If the "STOP" is opened, the oscilloscope will stop testing and enter the "STOP" state, and the pulse will disappear.At this point, if the **Message** is selected "open", the screen will display the results of the last statistics.

**OFF:** When the waveform that conforms to the set "mode" is detected, if **Aux "output"** chooses "on", then the back panel [**Trigger Out**] will output a pulse. If the "STOP" is closed at this time, the oscilloscope will still continue to test.At this time, if the **Message** is selected "open", the statistics results on the screen will be updated in real time.

#### Press Mode softkey:

Pass:when a passed waveform is detected, the instrument displays the statistic results and outputs a pulse.And the beeper doesn't sound.

Fail:when a passed waveform is detected, the instrument displays the statistic results and outputs a pulse.And the beeper doesn't sound.

Pass ring:when a passed waveform is detected, the instrument displays the statistic

results and outputs a pulse. And the beeper sounds (not related to the on/off state of the sound).

Fail ring: when a failed waveform is detected, the instrument displays the statistic results and outputs a pulse and the beeper sounds (not related to the on/off state of the sound).

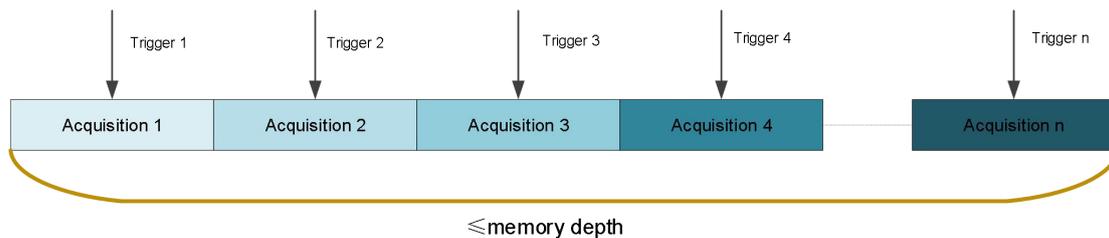
Press **Output** softkey to show the output of test results from the **Aux** at the rear panel. When the oscilloscope detects a waveform that does not conform to the set "mode", the rear panel Aux will not output the waveform. If no waveform that does not conform to the set "mode" is detected, the output pulse will continue.

## 15.Segmentation Acquisition

When triggering several times in the acquisition process of the segmented storage, the data obtained from the sampling of each trigger is stored in the storage space of each segment.

### 15.1.The principle of segmentation storage.

As shown in the figure below,the data acquired when the first trigger occurs is stored in the first segment storage space, and the data acquired when the second trigger occurs is stored in the second segment storage space. By the same analogy, the oscilloscope enters the pause state to finish the acquisition process when the number of segments set by the user is acquired. Read the first segment of the acquisition for display.



Take MPO6000EDU as an example,total storage depth is 128Mpts, The calculation formula of segment number distribution is as follows:

$$N=128*1024*1024/ \left[ \text{The current storage capacity integer to the Nth power of 2} \right] -1$$

The maximum storage of MPO6000EDU is 128Mpts, Under the condition of maintaining 1GSa/s sampling rate, the segmented storage range is supported: 1~80000 segments.

### 15.2.Enable Segmentation Acquisition Function

Utility → acquisition → segmentation acquisition → segmentation acquisition to enter segmentation acquisition and turn on or off segmentation acquisition function.

### 15.3.Set the Number of Segments of Segmentation Acquisition

Utility→acquisition→segmentation acquisition→segmentation setting to set the number of segments of segmentation acquisition.

### 15.4.Set to see the segment value after the segmentation acquisition

Utility → historical waveform → frame number to set to display the waveform of corresponding frame number on the screen after the segmentation acquisition.

### 15.5.Turn on/off list

Utility→historical waveform→list to turn on/off the segmentation acquisition list function.

## 16. Historical Waveform

When you start to use the oscilloscope, the oscilloscope will automatically acquire and store data, can acquire up to 8000 frames. Open the historical waveform can observe the waveform stored before, and measure and analyze them. It is also possible to acquire individual details or time-varying waveform at different speeds by setting time intervals and waveform navigation. Each record is indexed by a timestamp, historical waveform function can record the input waveform over a period of time before STOP.

Waveform playback function can play the current recorded waveform. During the playback, the information shown in the figure below will be displayed in the upper right corner of the screen. The data on the left side of the figure is the specific frame displayed on the current screen, which changes constantly during playback. The data on the right represents the maximum number of frames recorded.

### 16.1. Enable Segmentation Acquisition Function

1. Press  to play the next frame manually.
2. Press  to play the segmentation acquisition waveform in automatic sequence, the playback interval can be set by the interval, press  to pause autoplay function.
3. Press  to play the last frame manually.
4. The minimum interval for autoplay is 10ms.

## 17.Display Control

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

### 17.1.To Select the Display Type

Press Display → display type to set the waveform display mode to "Vectors" or "Dots".

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

Dots: display the sample points directly. You can directly view each sample point and use the cursor to measure the X and Y values of the sample point.

### 17.2.To Set the Persistence Time

Press Display → Persistence Time to set the persistence time of the oscilloscope to Min, specific values (Min, 1 s, 5 s, 10 s and 30 s) or Infinite. In the following part, the frequency sweep signal of a sine waveform is used to demonstrate the waveform effects in different persistence times.

#### 1. Min value

Enable to view waveform changing in high refresh rate.

#### 2.Specific Values

Enable to observe glitch that changes relatively slowly or glitch with low occurrence probability. The persistence time can be set to 1 s, 5 s, 10s and 30 s.

#### 3.Infinite

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

### 17.3.To Set the Screen Grid

Press Display → grid to set the screen grid type.

Only show vertical and horizontal center grids.

Only show vertical and horizontal center grids, and use the dot display to draw other horizontal and vertical grids.

Only show vertical and horizontal center grids, and use the line display to draw other horizontal and vertical grids.

### 17.4.To Set the Waveform Brightness

Press Display → Waveform brightness, or in the case of non-menu operations, rotate V0 to adjust the brightness of the waveform of the channel. The default is 50%, the adjustable range is 0-100%.

### 17.5.To Set the Grid Brightness

Press Display → Grid brightness to set the brightness of the screen grid, rotate V0 also can adjust the grid brightness. The default is 50%, the adjustable range is 0-100%.

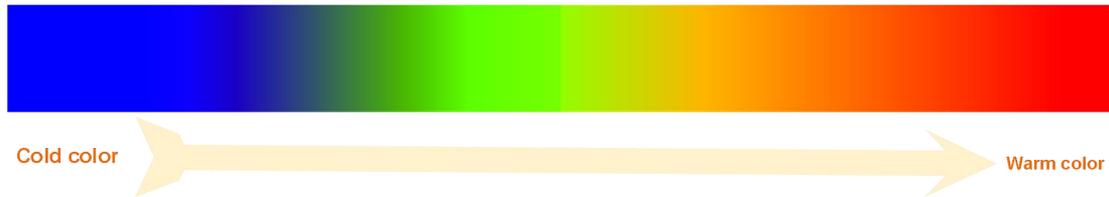
### 17.6.To Set the Screen Brightness

Press Display → Screen brightness to set the brightness of the screen, rotate V0 also can adjust the screen brightness. The default is 80%, the adjustable range is 0-100%.

### 17.7.To Set the Color Mode

#### 1.Color temperature mode

Color temperature function uses color changes to reflect the size of frequency of the waveform. The higher the frequency of the waveform, the warmer the color. The lower the frequency, the colder the color. Below is a gradient of cool and warm colors.



Press Display on the frontal panel → color mode → color temperature, select “enable” to enable color temperature function. You can judge the probability of the waveform appearing by comparing the color of the current waveform with that shown in the figure below.

## 2. Grayscale

Grayscale function uses the bright and dark colors to reflect the size of frequency of the waveform. The higher the frequency of the waveform, the brighter the color. The lower the frequency, the darker the color.



## 18.Store and Recall

Users can save the current settings, reference waveform, CSV file, and screen image of the oscilloscope in internal memory or external USB storage device in various formats. The saved settings, waveform, reference waveform and so on can be recalled when the USB storage device is connected to USB HOST port on the frontal panel. The series oscilloscope supports the storage of settings, waveform, reference waveform, CSV file, the default is setting.

The user can save the waveform of analog channel to the reference waveform position in the oscilloscope and then compare it with other waveform so as to judge the reason of the fault.

### 18.1.Save

#### 18.1.1.Storage Type

##### 1.Setting

Save the oscilloscope settings to internal or external storage, the format is .set.

##### 2.Waveform(binary system)

Save the oscilloscope waveform to internal or external storage, the format is .lwf.

##### 3.Reference Waveform

Save the reference waveform of the oscilloscope to internal or external storage, the format is .ref. When recalling, the reference waveform will display on the screen directly.

##### 4.CSV

Save the waveform data to external storage, you can specify the file name and the path to save, the format is .csv. The storage file contains the waveform data of the displayed logical channel and the main setting information of the oscilloscope. CSV files don't support to be recalled.

#### 18.1.2.Save Contents

Save the files of the oscilloscope to internal or external storage, Internal flash can store up to 9 Settings files (No.1~No.9). The saved setting files can be recalled.

## 18.2.Recall

#### 18.2.1.Recall Type

##### 1.Setting

Save the oscilloscope settings to internal or external storage, the format is .set.

##### 2.Waveform(binary system)

Save the oscilloscope waveform to internal or external storage, the format is .lwf.

##### 3.Reference Waveform

Save the reference waveform of the oscilloscope to internal or external storage, the format is .ref. When recalling, the reference waveform will display on the screen directly.

#### 18.2.2.Recall Contents

Save the files of the oscilloscope to internal or external storage, Internal flash can store up to 9 Settings files (No.1~No.9). The saved setting files can be recalled.

## 19.Auxiliary Function Setting

### 19.1.Language

This oscilloscope supports menus in multiple languages as well as provides Chinese and English help information, prompt messages and interface display. Press **Utility** → **Language**, rotate V0 to select the desired language, press V0 to select the language.

### 19.2.The switch of touch screen(not available for the machine without touch screen function)

This series oscilloscope supports to open and close the touch screen manually, press **Utility** → **touch screen switch** to open and close the touch screen.

### 19.3.System Information

press **Utility** → **system information** to view the system information of your oscilloscope. The system information includes the manufacturer, model, serial number as well as software version and hardware version.

### 19.4.Pass/Fail

After selecting this type, in the pass/fail test, when the oscilloscope detects a waveform that satisfied to the pass/fail rule and output mode, it will output a pulse through the connector; When no waveform confirming to the above conditions is detected, the connector no output.

### 19.5.Update the Firmware

DPO6000/MPO6000 can update the firmware by USB storage device, the whole process is about 5 minutes.

Please update the oscilloscope by the following methods:

1. Plug the USB flash disc in which a firmware program is saved in USB Host interface on the front panel of the oscilloscope.
2. Utility → Update → Update firmware → Manually select the firmware file → start the upgrade process.
3. Use V0 to select the .up file to update, and press V0 to select the updating file.
4. Press F3 to start the updating process.
5. The prompt message that says you have updated successfully will pop after updating.
6. The oscilloscope will restart automatically.

### 19.6.Self-calibration

The self-calibration program can quickly make the oscilloscope reach the best working state to get the most precise measurement values. You can perform self-calibration at any time especially when the change of the environment temperature is up to or more than 5°C. Make sure that the oscilloscope has been warmed up or operated for more than 30 minutes before the self-calibration. Disconnect all the input channel connections, and then press **Utility** → **Self-Cal**.

### 19.7.Function complex

Users can set the type of the signal output from the [Aux/Trigger Out] connector at the rear panel. The function of trigger out and the result of pass/fail can also be realized through "[pass/fail] → output to DDS" and "[signal generator] → burst → data source selection through/fail or DSO Trigger". For specific operation steps, please refer to the introduction of "through failure" and "burst release" chapters.

Press **Utility** → **Function complex** to select the desired output type.

#### Trig Output

After this type is selected, the oscilloscope outputs a signal that can reflect the

current capture rate of the oscilloscope at each trigger, Connect this signal to a waveform display device, measure the signal frequency and the measurement result equals the current acquisition rate.

### **Pass/Fail**

After this type is selected, the oscilloscope will output a pulse signal when the waveform that satisfied with the pass/fail rules and output mode is detected, transferring this signal to other control systems for convenient viewing of test results.

## **19.8.System Settings**

### **1.Set the (key)sound**

When the sound is enabled, you can hear the sound of the beeper when you press a menu softkey or when the prompt message pops up. Press **Utility** → **System set** → **Sound** to select off or on. The default is on.

### **2.Time Setting**

## **19.9.Self-test**

Self-test includes screen test, keyboard test and lighting test, which is mainly used to check whether the oscilloscope has the display color deviation, the response sensitivity of keys and knobs, and whether some key lights can be lit, etc mechanical problems of oscilloscopes.

## 20.Remote control

Remote control of the oscilloscope mainly has the following three ways:

1. USB control
2. Control by LAN
3. Via serial port control

### 20.1.USB control

Connect the USB Device interface of the back panel of the oscilloscope with the USB Host interface of the computer with USB data cable. The USB connected icon is displayed in the lower right corner of the oscilloscope screen.

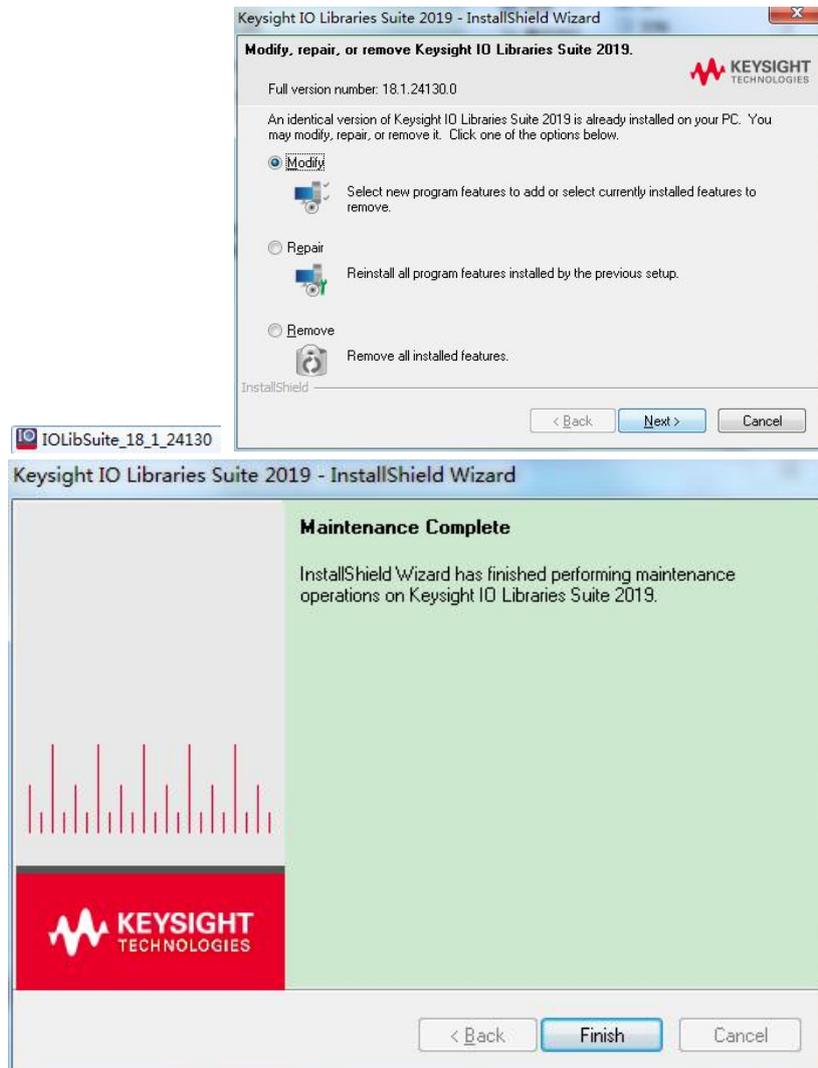
Install IO software:

#### Keysight IO libraries suite installed

Click the following url to download the latest Keysight IO Libraries Suite:

<http://www.keysight.com/main/software.jspx?ckey=2175637&lc=chi&cc=CN&nid=-11143.0.00&id=2175637>

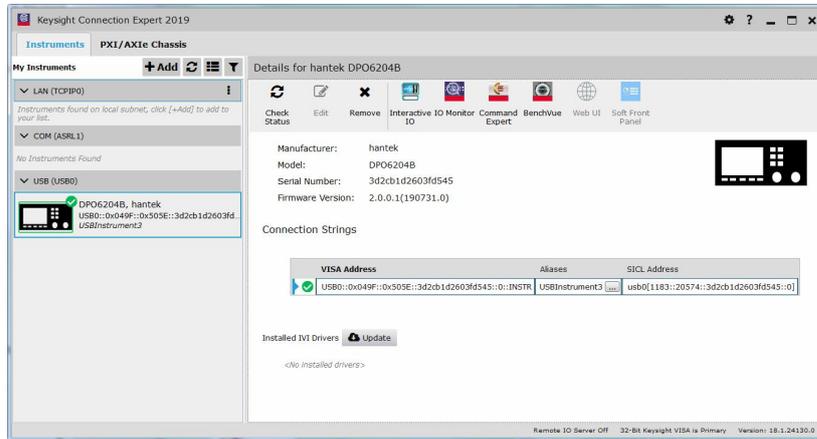
Double-click the application to start the installation. The installation process may take a few minutes, depending on the installation prompt and step by step.



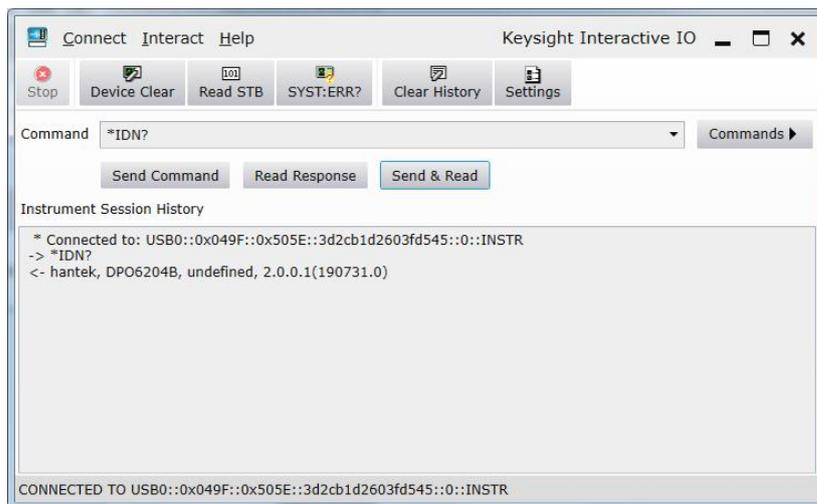
After the installation, you can see the IO program running in the lower right corner of the screen.



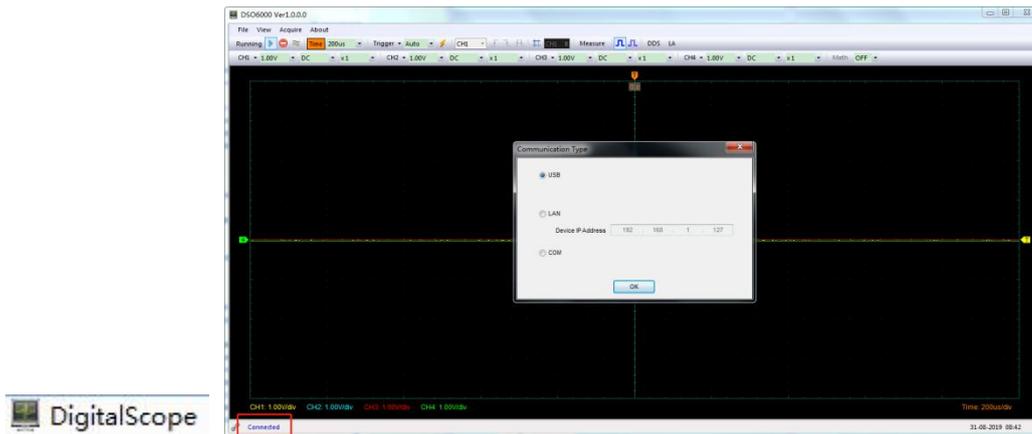
Double-click to open the IO program, you can see the connected oscilloscope name displayed below My instrument-usb.



The figure shows the connected state of USB device. Click on the "interactive IO", a command is sent arbitrarily, and the computer communicates with the device.



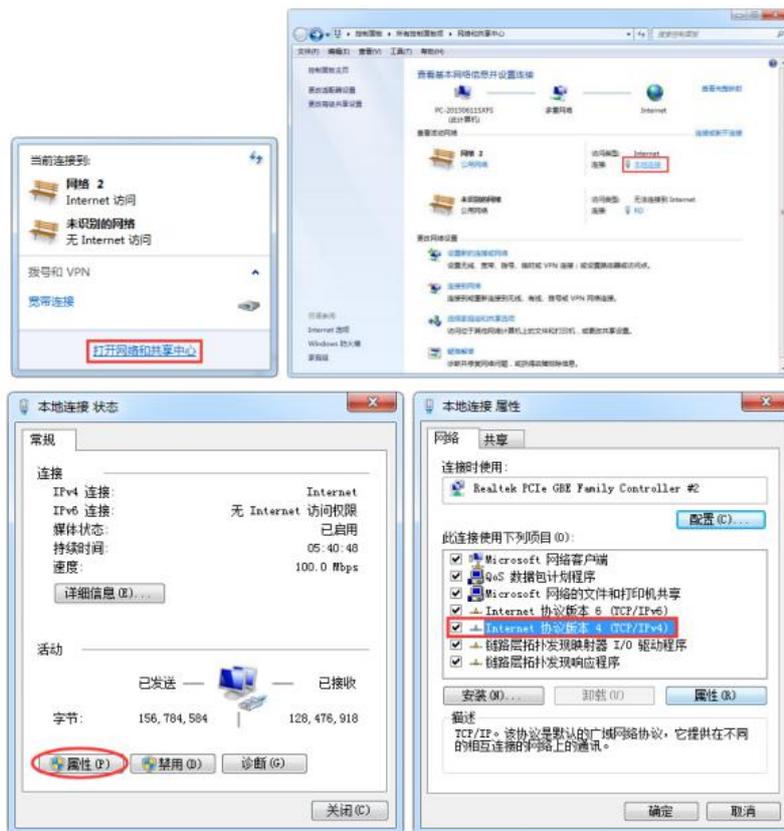
Double-click to open the host computer software, click file - connection - connection mode, and select the current connection mode. At this point, the lower left corner of the upper computer software shows the connected state.

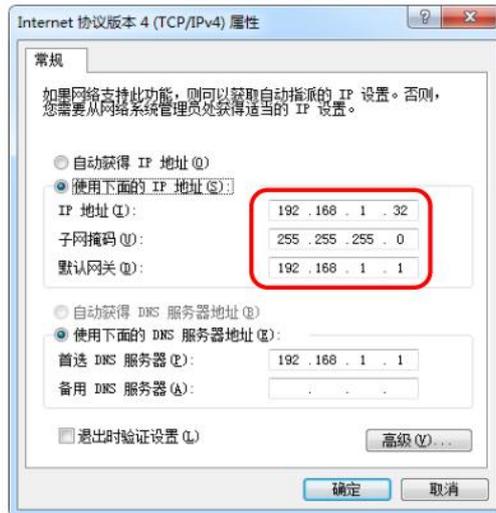


## 20.2.Control by LAN

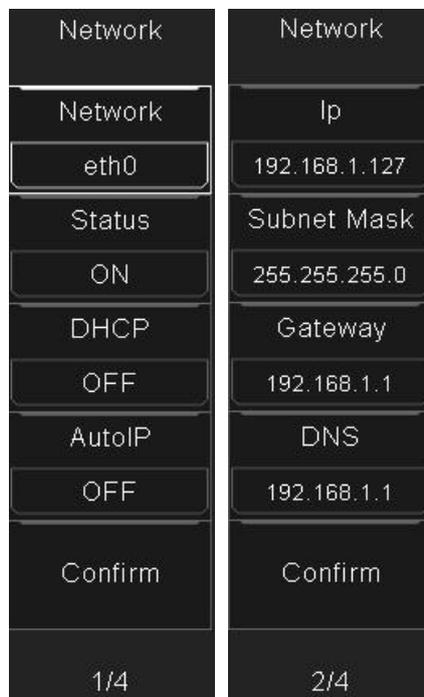
Connect the network port of the back panel of the oscilloscope to the USB Host interface of the computer with the network cable. The lower right corner of the oscilloscope screen shows the connected network icon.

Open network and sharing center → local connection → property → Internet protocol version 4 (TCP/IPv4), set the computer's IP address to 192.168.1.2 or any other IP (192.168.1.2~192.168.1.255) except oscilloscope IP (the default IP is 192.168.1.127), set the subnet mask as the default value. The lower right corner of the oscilloscope screen shows the connected network icon.

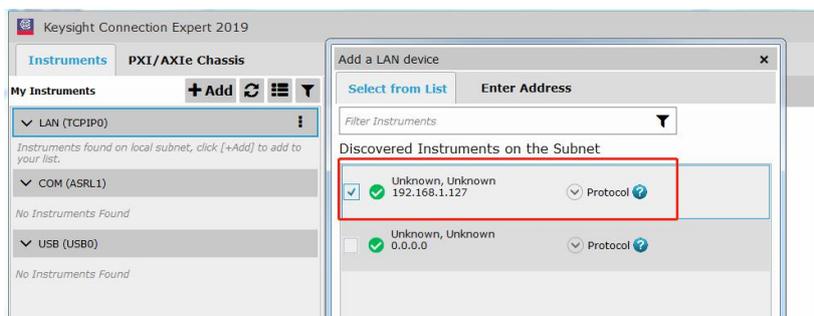




Set oscilloscope network parameters: [Utility] -- system Settings -- network, and confirm after setting corresponding IP address, subnet mask, gateway and DNS.



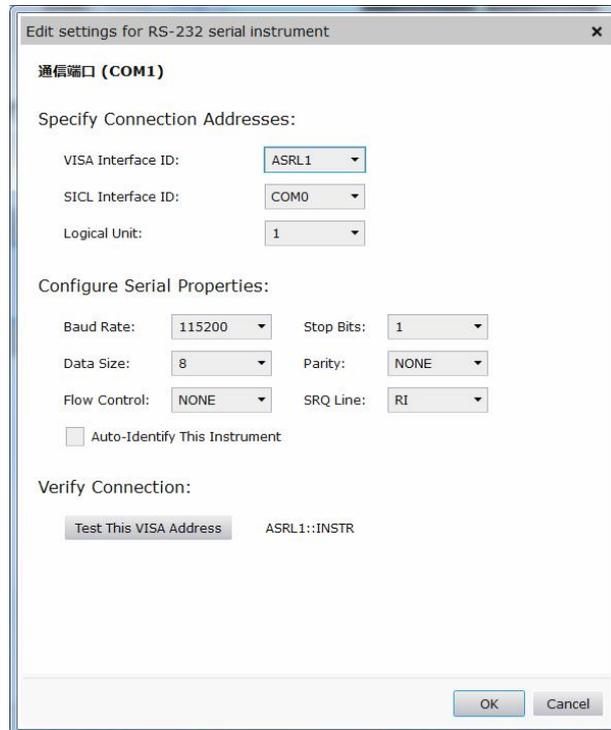
Open IO software and select the recognized device.



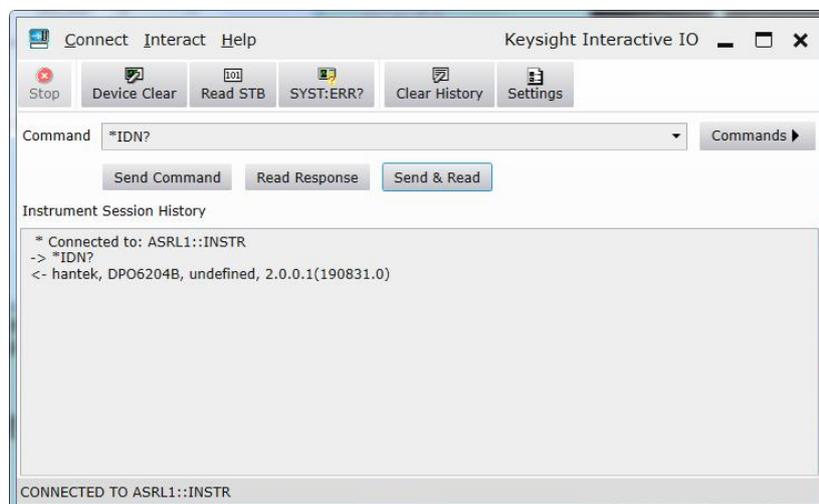
Open the upper computer software and select the corresponding connection mode.

## 20.3.Via serial port control

The network port end of the serial port line is connected to the network port of the back panel of the oscilloscope, and the serial port end is connected to the computer. Open IO program, select "add device" and set serial port properties, baud rate is 115200.



The list of "My instrument" COM of IO software shows connected devices. Click "interactive IO" to send any instruction, and the computer communicates with the device.



Open the upper computer software and select the corresponding connection mode.

## 21. Common failures and Troubleshooting

### Troubleshooting

#### 1. The oscilloscope doesn't turn on after pressing the power key:

- 1) Check the power cord on the side of the oscilloscope to verify it has been connected properly;
- 2) Check the power on/off button to ensure it has been pushed;
- 3) Then restart the oscilloscope.

If it still does not work correctly, please contact the local distributor or contact Hantek technology support department directly.

#### 2. There is no display of waveform on the screen when the oscilloscope is turned on:

- 1) Check the probe to assure its proper connection to the input BNC;
- 2) Check the channel switch (CH1-CH4 menu buttons) to make sure it has been turned on;
- 3) Check the input signal to verify it has been connected to the probe correctly;
- 4) Affirm that all measured circuits have signals to output;
- 5) Turn up the magnitude for DC signals with large magnitude;
- 6) In addition, you may press the Auto Measure button to perform an automatic detection of signals at first.

If there is still no display of waveform, please contact Hantek technology support department in time.

#### 3. If the waveform of the input signal is distorted seriously:

- 1) Check the probe to assure its proper connection to the channel BNC;
- 2) Check the probe to assure its good connection to the measured object;
- 3) Check the probe to verify it has been well calibrated. Otherwise, refer to the content about calibration described in this manual.

#### 4. If the waveform is rolling continuously on the screen but can not be triggered:

- 1) Check the trigger source to make sure it consistent with the signal input channel;
- 2) Check the trigger level to assure its correct adjustment. You may push the TRIGGER

LEVEL knob to reset the trigger level back to the center of the signal;

3) Check the trigger mode to confirm it is a right choice for the input signal. The default trigger mode is edge trigger. However, it is not suitable for all kinds of input signals.

## Technical Specifications

All specifications apply to the DPO6000/MPO6000 series oscilloscopes, to see the details in the last part of this chapter. Before checking an oscilloscope see if it complies with these specifications, make sure it meets the following conditions:

- The oscilloscope must have been operating continuously for twenty minutes under the specified operating temperature.
- The Self Calibration operation must be performed through the Utility menu if the operating temperature changes by more than 5°C.
- The oscilloscope must be within the factory calibration interval.

All specifications are guaranteed unless noted “typical”.

### Oscilloscope Specifications

#### Acquisition

Acquisition Methods	Real-time sample
Real-time sample rate	1 GSa/s(single channel) 500 MSa/s(two channels) 250 MSa/s(three/four channels);
Peak detection	Analog channel 4ns <b>Note:</b> digital channels don't support
Average mode	Analog channel All channels reach N time samples at the same time,N can be selected from 2、 4、 8、 16、 32、 64、 128、 256、 512 and 1024. <b>Note:</b> digital channels don't support
High resolution	Up to 12bit <b>Note:</b> digital channels don't support
Minimum test pulse width	8ns
Memory depth	Single channel 64M/128M[EDU] Two channels 32M/64M[EDU] Three,four channels 16M/32M[EDU] Five,six,seven,eight channels 8M/16M[EDU]

Input

Channel quantity	4 analog channels <b>Note:</b> data channels can't be opened
	3 analog channels <b>Note:</b> digital channel LA1/LA2/LA3/LA4/LA1LA2/LA3/LA4
	2 analog channels <b>Note:</b> digital channels infinitize
	1 analog channel <b>Note:</b> digital channels infinitize
	0 analog channel <b>Note:</b> digital channels infinitize
	Input coupling
Input impedance,DC coupling	Analog channel 25pF±3pF,1MΩ±2% Digital channel (300KΩ±2%) ,(8 pF±3 pF)
Supported probe attenuation factor	Analog channel 1X、 10X、 100X、 1000X
Voltage classes	300V CAT II
Maximum input voltage	Analog channel 300VRMS (10X) Digital channel -25V~25V

## Horizontal

Waveform interpolation	(sin x)/x
Maximum record length	Single channel maximum 128M(EDU series)/64M Two channels maximum 64M(EDU series)/32M three/four channels maximum 32M(EDU series)/16M five/six channels maximum 16M
Horizontal scale range	DSO6084 DSO6104      DSO6204

	2ns/div~100s/div 1,2,5 step by step
Time base mode	Y-T、X-Y、Roll
X-Y number	Channel 1,2 1 XY channel、channel3 4 1 XY channel
Zero offset	$\pm 0.5 \text{ div} \times \text{minimum time base gear}$
Sample Rate and Delay Time Accuracy	$\pm 25 \text{ ppm}$
Clock drifting	$\leq \pm 5 \text{ ppm/year}$
Delta Time Measurement	single, acquisition mode $\pm (1 \text{ sample interval} + 100 \text{ ppm} \times \text{reading} + 0.6 \text{ ns})$
Accuracy (Full Bandwidth)	$> 16 \text{ times averages}$ $\pm (1 \text{ sample interval} + 100 \text{ ppm} \times \text{reading} + 0.4 \text{ ns})$ Sample interval = sec/div $\div 200$

## Vertical

Bandwidth(-3db)	DPO6084	DPO6104	DPO6204
	80MHz	100MHz	200MHz
Vertical resolution	Analog channel 8bit Digital channel 1bit		
Vertical scale range	Input BNC position is $500 \mu\text{V/div} \sim 10\text{V/div}$		
Position range	$500 \mu\text{V/div}$ to $120\text{mV/div}$ , $\pm 1\text{V}$ $122\text{mV/div}$ to $1.2\text{V/div}$ , $\pm 10\text{V}$ $1.22\text{V/div}$ to $10\text{V/div}$ , $\pm 50\text{V}$		
Optional analog bandwidth limitation	Typical 20MHz		
Bass response(-3db)	In BNC position is $\leq 10\text{Hz}$		
Rising time in BNC position, typical	DSO6084	DSO6104	DSO6204
	$\leq 4.4\text{ns}$	$< 3.5\text{ns}$	$\leq 1.8\text{ns}$
Vertical gain accuracy	In "normal" or "average" acquisition mode, the accuracy of $10\text{V/div}$ to $10\text{mV/div}$ is $\pm 3\%$ . In "normal" or "average" acquisition mode, the accuracy of $5\text{mV/div}$ to $500\mu\text{V/div}$ is $\pm 4\%$ .		

DC offset accuracy	$\pm 0.1 \text{ div} \pm 2 \text{ mV} \pm 1\% \text{ offset value}$
The isolation of channels	DC maximum bandwidth: >40 dB

**Note:** Bandwidth reduced to 6MHz when using a 1X probe.

Vertical (digital channels)(used for MPO6000DMPO/6000EDU and DPO6000B/DPO6000C after buying digital probes LP104)

Threshold value	4 channels in 1 group adjustable threshold value
Threshold option	TTL(1.4 V) 5.0 V CMOS(+2.5 V) 3.3 V CMOS(+1.65 V) 2.5 V CMOS(+1.25 V) 1.8 V CMOS(+0.9 V) ECL(-1.3 V) PECL(+3.7 V) LVDS(+1.2 V) 0V User-defined
Threshold range	$\pm 7.0\text{V}$ , 10mV step by step
Threshold accuracy	$\pm(100\text{mV} + 3\% \text{ threshold setting})$
Dynamic range	$\pm 5.0\text{V} + \text{threshold}$
Minimum voltage swing	500 mVpp
Vertical resolution	1 bit

#### Trigger

Trigger level range	$\pm 5$ divisions from the center of the screen	
Trigger mode	auto, general, single	
Level	CH1~CH4	$\pm 4$ divisions from the center of the screen.
Holdoff range	8ns~10s	
Trigger level accuracy	CH1~CH4	0.2 div×volts/div within $\pm 4$ divisions from the center of the screen
<b>Edge trigger</b>		
Slope	Rising edge, falling edge, rising or falling edge	
Signal source	CH1~CH4, D1.0~D1.3, D2.0~D2.3,	

	D3.0~D3.3, D4.0~D4.3
<b>Pulse width trigger</b>	
Polarity	Positive polarity,negative polarity
Condition(When)	<, >, !=, =
Signal source	CH1~CH4, D1.0~D1.3, D2.0~D2.3, D3.0~D3.3, D4.0~D4.3
Pulse width range	8ns ~ 10s
<b>Video trigger</b>	
Signal standard	NTSC, PAL
Signal source	CH1~CH4
Synchronization	Scanning line、line number、odd field、even field、all field
<b>Slope trigger</b>	
Slope	rise,fall
condition(When)	<, >, !=, =
Signal source	CH1 ~ CH4
Time range	8ns ~ 10s
<b>Overtime trigger</b>	
Signal source	CH1~CH4, D1.0~D1.3, D2.0~D2.3, D3.0~D3.3, D4.0~D4.3
Polarity	Positive polarity,negative polarity
Time range	8ns ~ 10s
<b>Window trigger</b>	

Signal source	CH1~CH4,LA1~LA4
<b>Pattern trigger</b>	
Pattern	0:low level; 1:high level; X:ignore;
Level(signal source)	CH1~CH4
<b>Interval trigger</b>	
Slope	rise,fall
condition(When)	<, >, !=, =
Signal source	CH1~CH4, D1.0~D1.3, D2.0~D2.3, D3.0~D3.3, D4.0~D4.3
Time range	8ns ~ 10s
<b>Delay trigger</b>	
Edge type	Rising edge,falling edge
Signal source	CH1~CH4
condition(When)	<, >, !=, =
Time range	8ns ~ 10s
<b>Set up hold trigger</b>	
Edge type	Rising edge,falling edge
Signal source	CH1~CH4
condition(When)	<, >, !=, =
Time range	8ns ~ 10s
<b>Runt trigger</b>	
Polarity	Positive polarity,negative polarity
Condition(When)	<, >, !=, =
Signal source	CH1~CH4
Time range	8ns ~ 10s
<b>UART trigger</b>	

condition(When)	start、 stop、 data、 odd-even check、 reception error
Signal source(RX/TX)	CH1~CH4, D1.0~D1.3, D2.0~D2.3, D3.0~D3.3, D4.0~D4.3
Data format	Hex(hexadecimal)
Data length	1 byte
Data bit width	5 bit, 6 bit, 7 bit, 8 bit
Odd-even check	none、 odd、 even
Free level	high、 low
Baud rate (optional)	110/300/600/1200/2400/4800/9600/14400/19200/38400/57600/ 115200/230400/380400/460400 bit/s
Baud rate(user-defined)	300bit/s~334000bit/s
<b>LIN trigger</b>	
condition(When)	Interval field、 synchronization field、 ID field、 synchronization error、 identifier、 ID and data
Signal source	CH1~CH4, D1.0~D1.3, D2.0~D2.3, D3.0~D3.3, D4.0~D4.3
Data format	Hex(hexadecimal)
Baud rate(optional)	110/300/600/1200/2400/4800/9600/14400/19200/38400/57600/ 115200/230400/380400/460400 bit/s
Baud rate(user-defined)	300bit/s~334000bit/s
<b>CAN trigger</b>	

condition(When)	Start bit、remote frame ID、data frame ID、frame ID、remote frame data、data frame data、wrong frame、all errors、answer error、overload frame
Signal source	CH1~CH4
Data format	Hex(hexadecimal)
Baud rate(optional)	10000, 20000, 33300, 500000, 62500, 83300, 100000, 125000, 250000, 500000, 800000, 1000000
Baud rate(user-defined)	5kbit/s~1Mbit/s
<b>SPI trigger</b>	
Signal source	CH1~CH4, D1.0~D1.3, D2.0~D2.3, D3.0~D3.3, D4.0~D4.3
Data format	Hex(hexadecimal)
Data bit width	4, 8, 16, 24, 32
<b>IIC trigger</b>	
Signal source (SDA/SCL)	CH1~CH4, D1.0~D1.3, D2.0~D2.3, D3.0~D3.3, D4.0~D4.3
Data format	Hex(hexadecimal)
Data index	0~7
When(condition)	Start bit、stop bit、no response、address、data、restart
<b>Measurement</b>	
cursor	Voltage difference between cursors $\Delta V$
	Time difference between cursors $\Delta T$

	Reciprocal of $\Delta T$ , in Hertz ( $1/\Delta T$ )	
Auto measurement	frequency, period, mean, peak-to-peak, RMS, minimum, maximum, rising time, falling time, + width, - width, base, top, middle, amplitude, overshoot, preshoot, rising edge phase difference, falling edge phase difference, + duty, - duty, period mean, PRMS, FOVshoot, ROVshoot, BWIDTH, FRF, FFR, LRR, LRF, LFR, LFF	
DVM	Data source	CH1, CH2, CH3, CH4
	Measurement type	DC effective value
		AC effective value DC
Frequency meter	hardware 6 bits frequency meter	

#### Display

Display type	7" TFT diagonal liquid crystal
Display resolution	800(horizontal)*480(vertical) pixels
Display colour	16 million colours (24 bits true colour)
Persistence time	minimum, 1 s, 5 s, 10 s, 30 s, infinite
Display type	dot, vector
Display mode	Color temperature, gray scale
Display brightness	adjustable
Grid type	adjustable
Grid brightness	adjustable

#### Interface

Standard interface	USB Host, USB Device, LAN, EDU signal, WIFI Aux(trigger output/PassFail) – only EDU with this interface
Optional interface	AUX UART HDMI

Arbitrary waveform generator (for oscilloscopes with signal source channels)

Channel number	2/3 channels	
Sample rate	200MSa/s	
Vertical resolution	12 bits	
Maximum frequency	25 MHz	
Standard waveform	sin、square、pulse、triangular、noise、DC Sinc、index、semi-distortion、Lorentz、dual tone multiple frequency、Gauss、ECG	
Arbitrary waveform	Arb1,Arb2,Arb3,Arb4	
Sin	Frequency range	0.1Hz~25MHz
square/pulse	Frequency range	0.1Hz~10MHz
triangular wave	Frequency range	0.1Hz~1MHz
Sampling wave	Frequency range	0.1Hz~1MHz
Index	Frequency range	0.1Hz~5MHz
Semi-distortion	Frequency range	0.1Hz~1MHz
lorentz	Frequency range	0.1Hz~1MHz
Dual tone multiple frequency	Frequency range	0.1Hz~1MHz
Gauss	Frequency range	0.1Hz~1MHz
ECG	Frequency range	0.1Hz~1MHz
Arbitrary wave	Frequency range	0.1 Hz至10 MHz
	Waveform length	8KSa
Frequency	accuracy	100 ppm(<10 kHz) 50 ppm(>10 kHz)
	resolution	0.1 Hz or 4 bits,take the greater one
Amplitude	Output range	10mV~7Vp-p(high impedance)
		5mV~3.5Vp-p(50Ω)
DC offset	range	±3.5 V,high impedance
		±1.75 V,50 Ω
	resolution	100 μV or 3 bits,take the greater one
	accuracy	2%(1 kHz)
Output impedance	50 Ω	

### General specifications

Probe compensator output		
Output voltage ,typical	about 2Vpp input $\geq$ 1M $\Omega$ load	
frequency、typical	1kHz	
Power supply		
Power supply	100-120VAC <sub>RMS</sub> ( $\pm$ 10%),45Hz to 440Hz,CAT II 120-240VAC <sub>RMS</sub> ( $\pm$ 10%),45Hz to 66Hz,CAT II	
Power consumption	<30W	
Fuse	T,3.15A,250V,5x20mm	
Environment		
Operating temperature	0~50 °C (32~122 °F)	
Storage temperature	-40~+71 °C (-40~159.8 °F)	
Humidity	$\leq$ +104°F ( $\leq$ +40°C): $\leq$ 90% relative humidity	
	106°F ~122°F (+41°C ~50°C): $\leq$ 60% relative humidity	
Cooling method	convection	
Altitude	Operating and non operating	3,000m(10,000 feet)
	Random vibration	0.31 g <sub>RMS</sub> from 50Hz to 500Hz, 10 minutes on each axis
	Non operating	2.46g <sub>RMS</sub> from 5Hz to 500Hz, 10 minutes on each axis
Mechanical shock	Operating	50g,11ms, half-sine wave
Mechanical		
Size	318 x 140 x 150mm(length x width x height)	
Weight	2900g	

## Appendix

### Appendix A: Accessories

This product provides the following accessories, you can get all the accessories from the local distributor.

Standard accessories:

4 passive voltage probes(X1,X10)

1 power cord

1 USB cable

1 LAN cable

BNC to BNC cable

**Note:**DPO6004B series doesn't supply BNC cable,DPO6004 series and MPO6004D series supply 2 BNC to BNC cable,MPO6004EDU series supply 3 BNC to BNC cable.

1 CD

Warranty card

Certificate of qualification

Calibration certificate

**Appendix B: Harmful and Poisonous Substances or Elements**

Component <sup>2</sup>	Harmful and poisonous substances or elements <sup>1</sup>					
	Pb	Hg	Cd	Cr(Vi)	PBB	PBDE
Shell and chassis	X	0	0	X	0	0
Display module	X	X	0	0	0	0
Circuit board	X	0	0	X	0	0
Power supply	X	0	0	X	0	0
Electric wire and cable assembly	X	0	0	0	0	0
Connector	X	0	0	X	0	0
Fasten and installed hardware	X	0	X	X	0	0
Orther accessories (including probes)	X	0	0	X	0	0
Others	0	0	0	0	0	0

'X' means that at least the content of this poisonous and harmful substance in a homogeneous material of this component exceeds the limit specified in the SJ/T 11363-2006 standard.

'0' indicates that the content of this poisonous and harmful substance in all homogeneous materials of this component is refrained under the limit stated in the SJ/T 11363-2006 standard.

This component list contains components approved in the file '*Management Measures*'.