Virtual Instruments User Guide

HK LRF Limited

Web：www.HKLRF.com

Contents

I. Overview ..... - 4 -

1. Basic knowledge ..... - 4 -
2. Product models and specification ..... - 5 -
3. Supported protocols ..... - 6 -

II. Brief introduction to Kingst VIS ..... - 7 -

1. How to install software ..... - 7 -
2. Brief introduction to GUI ..... - 7 -
3. Multi-language switching ..... - 8 -
4. Brief introduction to the demo function ..... - 8 -

III. Connect the device ..... - 10 -

1. Connect the device to PC ..... - 10 -
2. Connect the device to system under test ..... - 10 -
3. Multipoint grounding to increase accuracy ..... - 11 -

IV. Details of operations ..... - 12 -

1. Sampling depth and sampling rate settings ..... - 12 -
2. Trigger condition settings ..... - 13 -
3. Acquisition signal ..... - 14 -
4. Waveform check and operations ..... - 15 -
5. Waveform measurement ..... - 16 -
6. Analyzers ..... - 18 -
7. Sample mode select ..... - 20 -
8. Channels settings ..... - 21 -
9. Save the settings and data ..... - 24 -
10. Export the data ..... - 26 -
11. Threshold voltage setting ..... - 27 -
12. PWM and Trigger Position Settings ..... - 27 -
13. The “Options...” in the main menu ..... - 28 -

V. Settings for standard protocols ..... - 31 -

1. UART/232/485 ..... - 31 -
2. I2C ..... - 32 -
3. SPI ..... - 33 -
4. CAN ..... - 33 -

5. Parallel ..... - 34 -
6. 1-Wire ..... - 35 -
7. DMX-512 ..... - 36 -
8. UNI/O ..... - 36 -
9. User-defined protocol analyzer ..... - 36 -

VI. FAQs ..... - 37 -

1. Driver installation fail with the device connected to computer ..... - 37 -
2. Identification fail or work unstably with the device connected ..... - 37 -
3. Signal glitches appear on individual channels ..... - 37 -
4. The actual sample time is less than expected ..... - 37 -
5. Automatic update failed ..... - 38 -

VII. Contact us ..... - 39 -

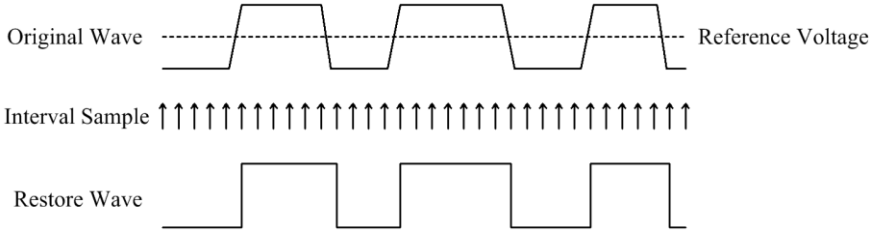
I. Overview

1. Basic knowledge

Logic analyzer is the instrument that collects and displays the digital signal from the system under test. It is mainly used for timing judgement an analysis. Unlike the oscilloscope with many voltage grades, It has only two grades(Logic “1” and Logic “0”). After the reference voltage is set, the logic analyzer could decide from the test signal that the signal above the reference voltage is logic 1, and the signal below is logic 0. The digital waveform is formed with 1 and 0. Compared with the oscilloscope, when testing and measuring the digital systems like MCU, ARM, FPGA and DSP, the logic analyzer could provide better accuracy, much more data and more complicated measuring methods.

For example, if you are sampling a signal with the logic analyzer, the sample rate of which is 1MHz, and the reference voltage(threshold voltage) is set to 1.5V, the logic analyzer would compare the current voltage with 1.5V. The signal above 1.5V would be high level(logic 1), and the signal below 1.5V would be low level(logic 0). Thus we get a sample point, and then we could link all these points(logic 1 and logic 0) to get a waveform, in which the user could see and analyze the timing of the signal, logic errors, the relation between each other, etc.

The figure below shows how the logic analyzer samples the signal:



According to the difference of hardware design, logic analyzer can be divided into independent type (or single type) or virtual type which needs to be combined with computer. The logic analyzer products introduced by Kingst are all virtual logic analyzers. The hardware of the device capture the signals according to the set conditions, and then uploads the sampling data to the computer software KingstVIS. KingstVIS can restore the original data to digital waveform and display it on the computer screen, decode the data according to the standard protocols supported, and other analysis and measurement functions.

2. Product models and specification

Product model	LA1010	LA1016	LA2016	LA5016	LA5032
Number of channels	16	16	16	16	32
Max sampling rate	100M@3CH, 50M@6CH[1] 32M@9CH, 16M@16CH	100M	200M	500M	500M
Measurement bandwidth	20MHz	20MHz	40MHz	80MHz	80MHz
Min detectable pulse width	20ns	20ns	12.5ns	6.25ns	6.25ns
Hardware memory size	--[2]	1Gbits	1Gbits	2Gbits	4Gbits
Hardware sampling depth	--[2]	50MSa	50MSa	100MSa	100MSa
Max compression depth	10GSa				
Input voltage range	-50V～+50V				
Input impedance	220KΩ, 12pF				
Threshold voltage	Range: -4V～+4V, Step: 0.01V				
Port type	USB2.0				
Standby current	100mA	130mA	130mA	200mA	350mA
Max active current	200mA	260mA	280mA	400mA	480mA

Note:

(1) LA1010 has no internal memory, so the sampling data needs to be uploaded to the computer in real time. Limited by the bandwidth of USB, only 3 channels can be enabled when working at 100M sampling rate, 6 channels at 50M, 9 channels at 32M and 16 channels at 16M.

(2) LA1010 has no internal memory, so the sampling data needs to be uploaded and compressed in real time. The maximum sampling depth that can be achieved depends on the available memory capacity of the computer. Each signal jumping edge of each channel will increase the memory consumption of 5 bytes..

3. Supported protocols

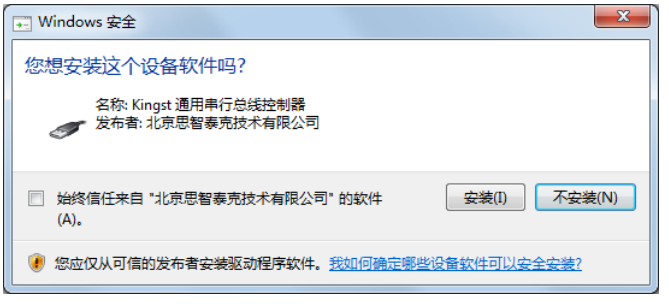
	LA1010	LA1016	LA2016	LA5016	LA5032
1-Wire	●	●	●	●	●
Atmel SWI	●	●	●	●	●
CAN	●	●	●	●	●
CAN-FD	x	x	x	●	●
DMX-512	●	●	●	●	●
HDMI CEC	●	●	●	●	●
I2C	●	●	●	●	●
I2C-EEPROM	●	●	●	●	●
I2S/PCM	●	●	●	●	●
IR-NEC	●	●	●	●	●
IR-JVC	●	●	●	●	●
IR-RC5	●	●	●	●	●
IR-SIRC	●	●	●	●	●
ISO7816	x	x	x	●	●
JTAG	●	●	●	●	●
LCD1602/12864	●	●	●	●	●
LIN	●	●	●	●	●
Manchester	●	●	●	●	●
MDIO	●	●	●	●	●
MIDI	●	●	●	●	●
MIPI-DSI LP	x	x	x	x	●
Modbus	●	●	●	●	●
Parallel	●	●	●	●	●
Parallel 8080	●	●	●	●	●
Parallel 6800	●	●	●	●	●
PS/2 Keyboard	●	●	●	●	●
QSPI	x	x	x	●	●
RGB-WS28xx	●	●	●	●	●
SDIO	●	●	●	●	●
SMBus	●	●	●	●	●
SPI	●	●	●	●	●
SWD	●	●	●	●	●
UART/RS232/485	●	●	●	●	●
UNI/O	●	●	●	●	●
USB LS/FS	●	●	●	●	●
USB-PD	x	x	x	●	●

II. Brief introduction to Kingst VIS

1. How to install software

Kingst virtual instruments software Kingst VIS could be found in the attached CD-ROM, or downloaded from the web page: https://www.HKLRF.com And the software package is something like KingstVIS_Setup_v3.x.x.exe (v3.x.x represents the version).

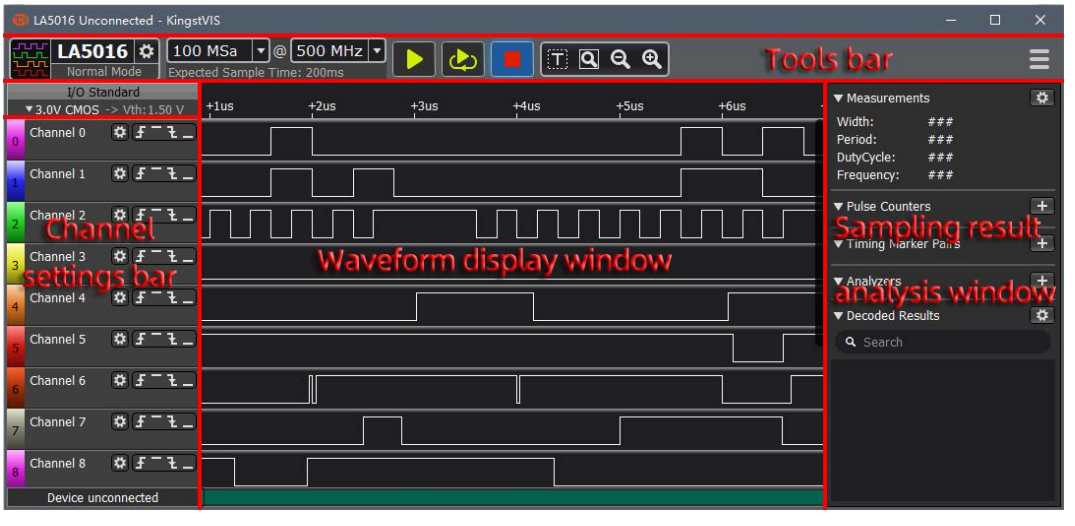
Double click the package file to execute the installation program. The procedure is similar with the common software in windows, and there are instructions that you could follow in every step. In the last step, you should install the driver program of hardware device, and you will see the dialog as the figure below(there could be differences between different OS). Please select “Install” to complete the procedure.



After the install procedure is complete, a shortcut would be created in the start menu and desktop, and then the Kingst VIS software could be accessed with this shortcut.

2. Brief introduction to GUI

When the software is started, you will get a screen similar to the figure below. The detailed information of the software would be introduced in Chapter 4, and here is only a brief introduction.



As shown in the figure above, the software interface could be divided in several parts:

①. Tools bar: top of the GUI, including common settings of the current device and the main menu button of the top right corner.

②. Threshold voltage: top left of the GUI, you could select or customize the threshold voltage value through the combo box .

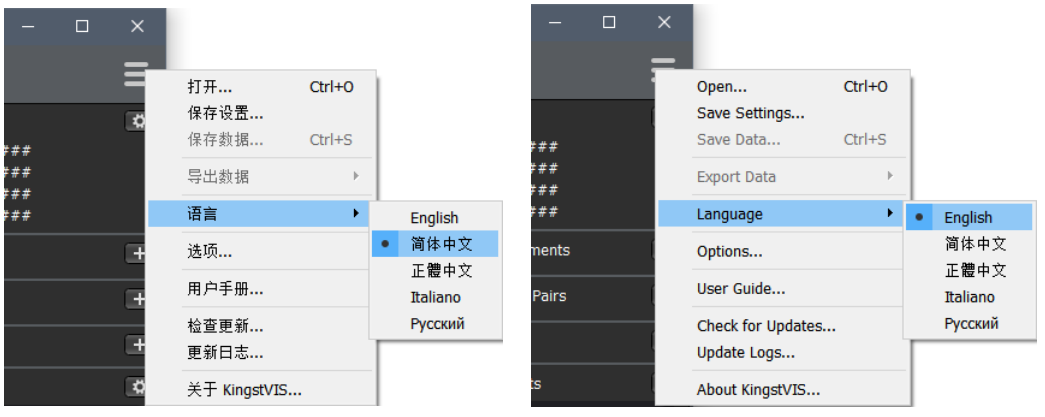
③. Channel settings bar: left of the GUI, is the number and name of current measurement channels.

④.Waveform display windows: middle of the GUI, the topmost is the timeline. The sampled waveform displays in the middle and a scroll bar is below it.

⑤. Sampling result analysis windows: right of the GUI, the top half shows measurement results that are frequently used, and you can add analyzer decoder and see the results on the bottom half.

3. Multi-language switching

The Kingst VIS software supports multilingual switching. For example, switch between Chinese and English, click the main menu button in the top right corner, move the mouse to the menu item “Language”/“语言”, and select the language, The selection would become effective after the software is restarted.

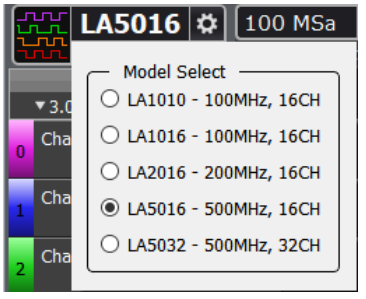


4. Brief introduction to the demo function

The software could provide demo function. You can simulate the functions without the actual hardware, and you could get a good experience of the software through this function.

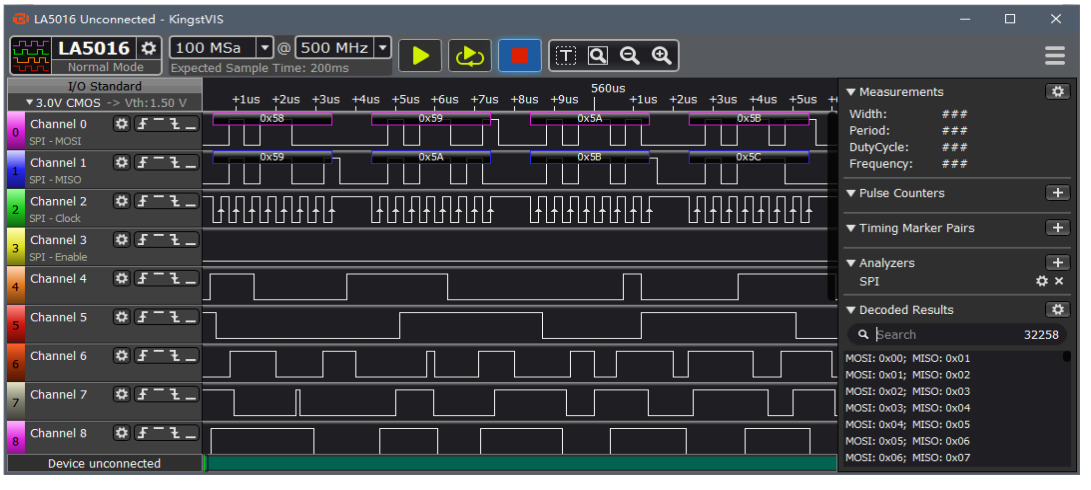
All Kingst Virtual Instruments share the same Kingst VIS software, there is a device control bar at the top left of the software interface, as shown in the figure below.

The icon of left side represents that logic analyzer is the current device, and The middle character indicates the model of the current device. If you click the current device model , as shown in the figure below, you could see all the devices supported. You could select any device that you want. You could evaluate and experience the device that you selected.



After the button on the toolbar is pressed, the software would simulate the work process as the actual device exists. And the software would eventually display some square wave or pulse signals. One of the key functions of the logic analyzer is that, it could analyze the signal that complies with some standard protocol. Here we take SPI as an example, to give a brief introduction. Find the "Analyzers" on the right side of GUI, as shown in the figure below.

After press the button on the right, and select "SPI" from the popped menu. We will get a analyzer settings dialog, the settings on which are default. Next we can press the menu on the toolbar, and we will get the SPI signal waveform on the channels 0-3, with the analyzer result. And other channels still show random square wave or pulse signal(as the figure below shows).



By clicking the left and right mouse button or scrolling the scroll wheel, you can enlarge or reduce the waveform, and hold down the left mouse button to drag the waveform. More details are covered in following chapters.

III. Connect the device

1. Connect the device to PC

After the software is installed, the logic analyzer can be connected to PC through the attached USB cable (In case of desktop computer, use the USB port behind the tower box). Then the computer would report that new hardware has been found. In Windows XP, there would appear a driver installation dialog, and just select to install automatically. In Windows 7/8/10, a dialog would appear in the lower right corner of the screen. Then the install process would start automatically, and we just need to wait for a while. After the install process is completed, a new device called "Kingst Instrument - Logic Analyzer" would appear in "Device Manager->Universal Serial Bus".

After the device is connected to PC and the driver has been installed, the device would be connected automatically when we open the software. When the connection is complete, as shown in the figure below, the device bar on the top left side of GUI would display current device type.

The icon on the left side means that the current device is a logic analyzer. And through the two buttons on the right side, you could select other devices or change the settings of current device. The detailed information would be introduced in chapter 4.

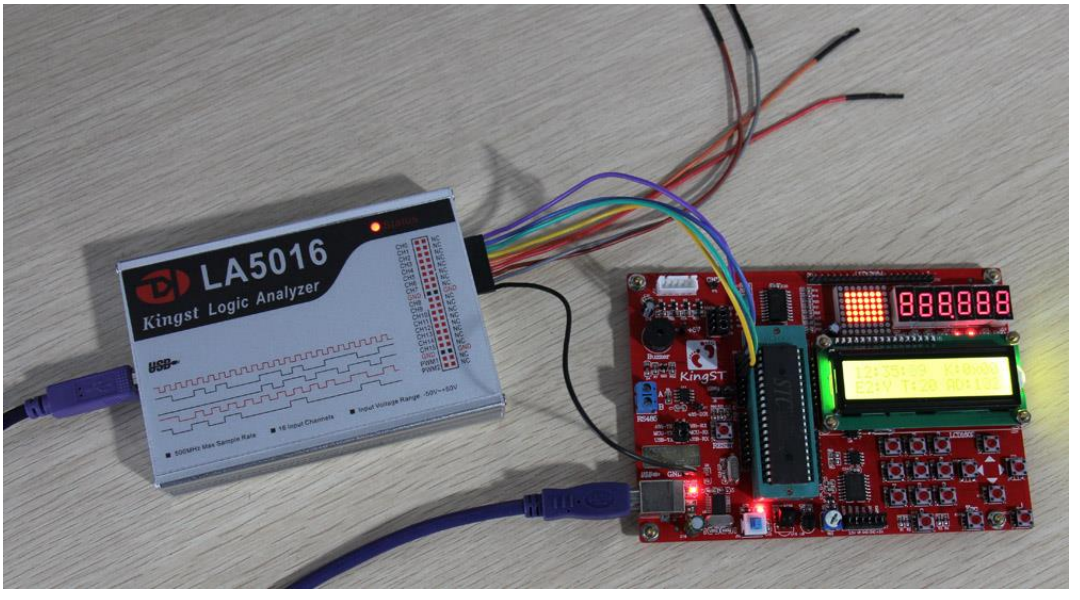
Besides, the status bar of the bottom left side shows the connection status of current device. "Device connected" means the device hardware has been connected to PC successfully and ready to work, while "Device unconnected" means the device hardware is not connected to the USB port of PC, or because there is something wrong, the device could not work normally.

2. Connect the device to system under test

Please note that the logic analyzer and the computer share the same ground, so the voltage between the GND of system under test and the GND of the computer should be zero. Especially when the system is connected to the force electricity, please make sure the isolation is made.

When the logic analyzer is connected with the system under test, you should first connect the GND channel to the system under test, and then connect the signal channels. There are 16 channels in the logic analyzer. It means that up to 16 digital signals could be tested simultaneously. If the number of existing signal is less than 16, the channels could be selected at will. The channel numbers of the software correspond to that of the hardware device.

In addition, when measuring the signal with high speed, the measuring lines of logic analyzer should be near the signal of system under test. The cable between the logic analyzer and the system under test should not be long, because long cables would result in heavy inductive effect and signal reflection. Therefore it is recommended that in debug stage of the system some pins should be reserved on the experiment board to make the best of measurement. The connection between the logic analyzer and the system under test is shown in the figure below:



3. Multipoint grounding to increase accuracy

when measuring multiple channels with high frequency signal, the signal current from all channels would flow into the system under test through GND channel, and the inductive effect of the wire in high frequency is strong, so signal current of multiple channels would overlap on GND channel, and as a result of that, the instantaneous voltage difference would be too large to result in the "glitch" on the waveform under test.

To remove these "glitches", we could take the multipoint grounding method. Normally the logic analyzer would provide several GND channels. If we connect these channels to the grounding point of the system under test, the signal current which we have mentioned would be divided into different path, and the "glitch" could also be removed. The multipoint grounding mainly includes:

①. Direct connection——GND channel of the logic analyzer should be connected to the GND wire of the system under test directly.

②. Dispersed connection——the GND channels should be connected to different parts of the system under test. Multiple GND channels should not be connected to one grounding point of the system.

Besides,as a "remedial measure", the glitch filter can also be set by software. For details, please refer to 4.8.2.

IV. Details of operations

After the software installation and device connection are complete, we could sample the signal and analyze it. In this chapter we will see how to use the logic analyzer step by step. Some of these steps need to be configured only once, and in future operations we could just skip them.

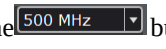
1. Sampling depth and sampling rate settings

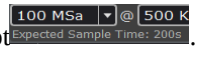
Sampling depth: the number of sample points that is collected in one sampling process. It defines how much data the device could sample. The bigger the depth is, the more data the device could sample for one time.

Sampling rate: also called sampling speed. This is the frequency the device samples, and it equals the number of sample points per second. It defines the time accuracy of the sample result. The higher the sampling rate is, the higher the time accuracy is. The time one sample point takes equals "1 / Sampling rate", that is also one sampling period.

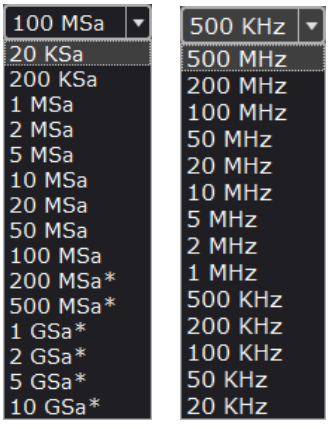
The duration of a sampling process is equal to "sampling depth / sampling rate". Before sampling, you should first evaluate the signal under test, including the maximum frequency, the sampling time, then we could select the sampling rate from the maximum frequency. To do this, the rule we should follow is "Sampling rate must be 5 times more than the maximum frequency of the tested signal, but 10 times would be better". The higher the sampling rate is, the time accuracy is higher. But the sampling rate should not be too high, as in the same sampling depth, the higher sampling rate would result in the shorter sampling time. So when we think about the sampling time needed, the sampling rate should be a little more than minimal requirements.

The first combo box in the left side of Kingst VIS software toolbar is sampling depth, and second one is sampling rate, as shown in the figure below. After the mouse clicks the

button, the list for selecting the sampling depth appears. And the list for selecting the sampling rate appears after clicking the 

button. Select the sampling rate and sampling depth that you want to set, the software calculates the corresponding sample time automatically and gives a prompt 

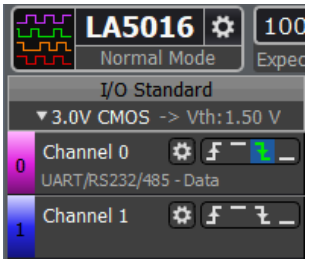
.



2. Trigger condition settings

If we have set the sampling depth to 1M and the sampling rate to 8M, we could sample the data for 125ms. In the default settings with no trigger set, after we press the button, the logic analyzer would begin sampling immediately, and it would stop automatically in 125ms. The waveform of signals under test would display in PC screen. But in real environment, the signal may not be continuous, and the user can not tell when it would occur, such as UART communication. In this way, maybe we could not sample any effective data during the time since we pressed the button. To solve this problem, we could utilize the trigger function.

First we set a certain condition, and then when the signal meets the condition, the data sample would start. This is how trigger works, and the conditions here are trigger conditions, such as jump edge of the signal, high/low level or the combinations of them. The trigger conditions should be set based on the characteristics of the signal to test, for example, in UART communication, for the idle state in which no data are transferred, the signal is high level, and every UART data frame is started by the transfer from idle state to start bit, which is low level, so we should set the falling edge of this channel as the trigger condition. If the channel 0 is used for UART signal, as the below figure shows, we could press the button on the right side of channels settings bar of channel 0, and the button represents “falling edge trigger”.

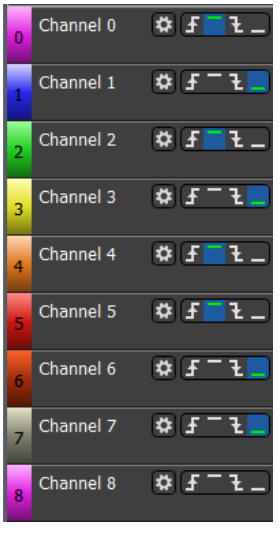


Indicates "rising edge trigger". Indicates "high level trigger".

Indicates "falling edge trigger". Indicates "low level trigger".

After the trigger is set and the button is pressed again, if the trigger condition (falling edge) we have set have not appeared on channel 0, the logic analyzer would stay in wait state until the falling edge arrives. Then the device will sample and save the data, and upload the data to the computer for displaying and analyzing when the process is complete.

Except the edge and level trigger condition for single channel, the logic analyzer also supports the condition combinations of multiple channels, such as levels, one edge and multiple levels. The final condition is the “logical AND” of these conditions, which means the sampling process starts when all conditions are met. In this way, the trigger can be the result of certain parallel data. It could be used in the situation when the master device like MCU accesses the peripheral through the bus and we want to check the data write/read operations of a certain address. For example, if we want to check the data operations of address 0x35, channels 0-7 should be connected to 8 address lines, and the other channels are connected to the data lines. After we set the level combination of channel 0-7 as the trigger condition, the data in this address could be sampled. The trigger settings are shown in the below figure.



3. Acquisition signal

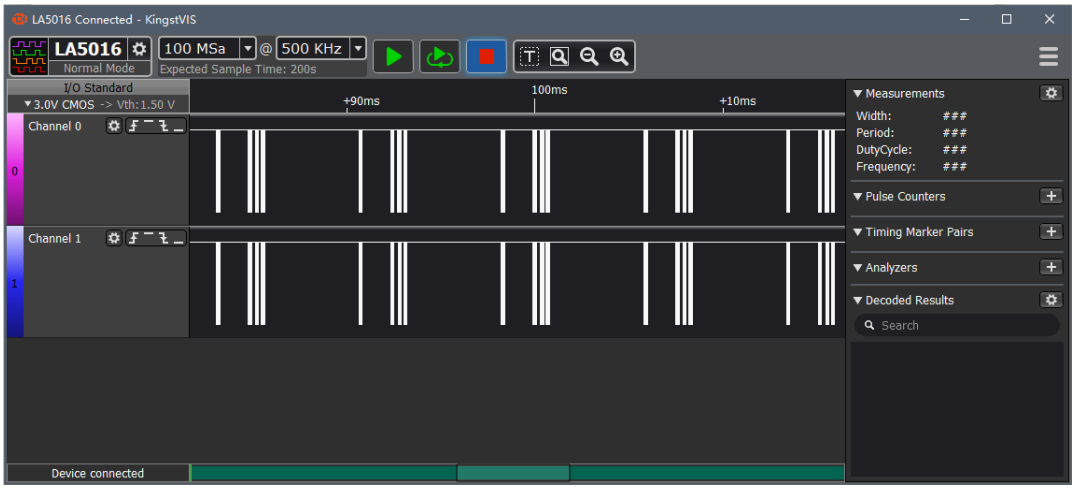
Single acquisition signal:After the basic settings we mentioned above have been made, we could begin sampling the signals that we need. The sampling process is started by pressing the button. The logic analyzer samples the signal since then (trigger conditions should be met if they are set), and stops if it has got the sample points (sampling depth) required. It would upload the data to the computer. The software would restore the waveform and maybe measures or analyzes the data later.

Automatic repeat acquisition:Click the button in the toolbar to start automatic repeat acquisition.When the collection time is reached, the software will automatically starts the next collection, it is not necessary to click the button manually until the sampling needs to be stopped, click the button to stop the sampling immediately.

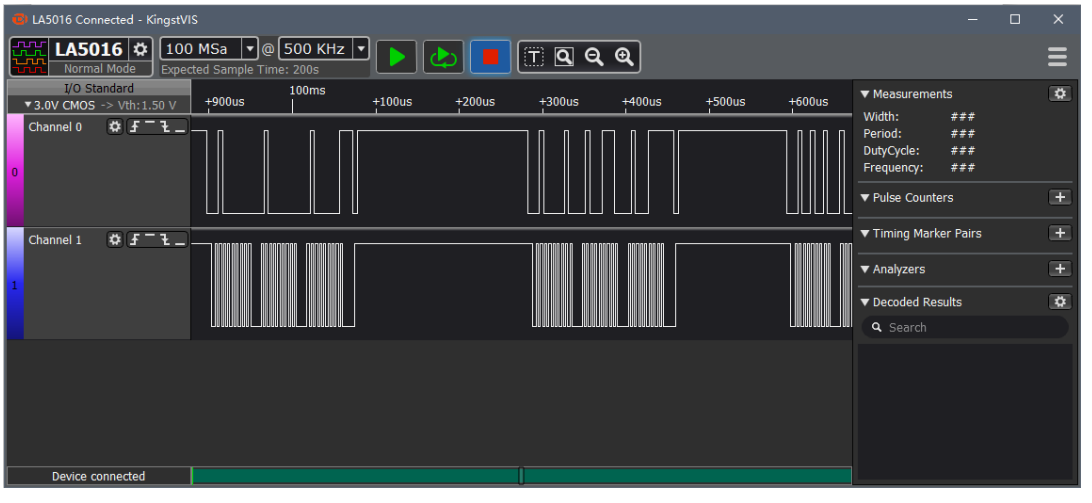
Single acquisition signal, automatic repeat acquisition and stop the sampling buttons. When the background color of the button is blue (when it is on), it indicates that it is in the current operation state, and the icon is display as follows: , and .

4. Waveform check and operations

After the sampling process is complete, the waveform would be displayed in the screen. We will continue this topic with an I2C communication example, and this example is based on KST-51 development board.



From this figure, we can see the coordinate values of the time axis are too large, and the effective waveform is within a very short period. You can click the left button of the mouse to zoom in the waveform; while the right button would zoom it out. And you can get these done with the mouse wheel too. After the waveform is zoomed up, you could see:



The waveform window supports several mouse operations:

①. press the left button: zoom in the waveform

②. press the right button: zoom out the waveform

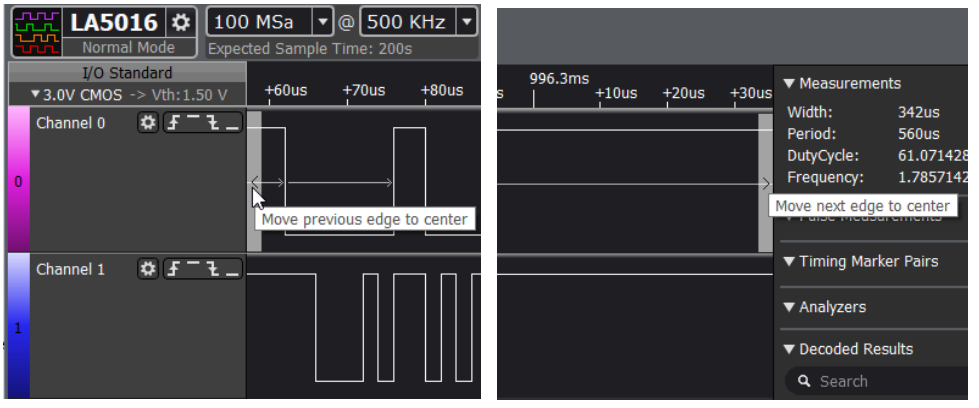
③. wheel up: zoom in the waveform

④. wheel down: zoom out the waveform

⑤. dragging with left button pressed: move the waveform to the left or right

⑥. press the button left side of the channel: Move previous edge to center

⑦. press the button right side of the channel: Move next edge to center



In the toolbar also provides a convenient waveform observation and operation buttons, as shown below.

Click the button to jump to the position of waveform time axis 0.

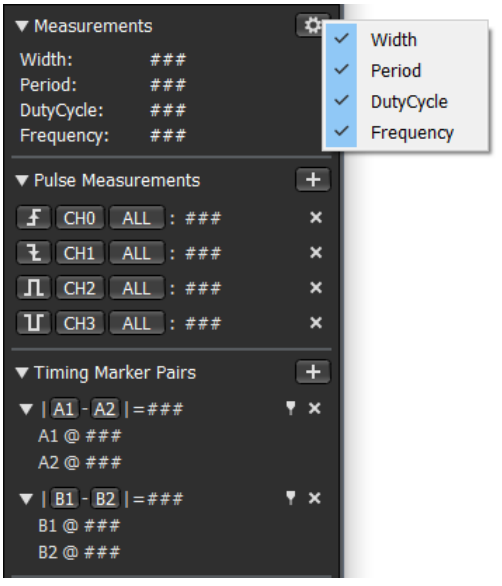
Click the button to zoom the waveform to full screen.

Click the button to zoom out the waveform slightly.

Click the button to zoom in the waveform slightly.

5. Waveform measurement

Besides what we could see directly from the waveform window, we could also check some items from the measurement window on the right side of GUI. If you press the settings menu in the top right corner, you could see all the measurement items. As is shown in the below figure, the users could press the menu to select or deselect certain items.



After some items have been selected, move the mouse inside the waveform window, the measurement results of where the mouse stays would display in the measurement window.

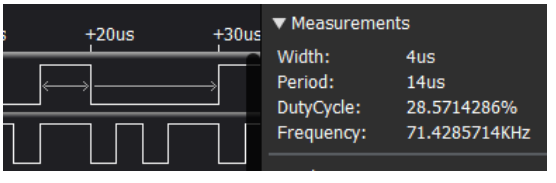
①. width: the width of the pulse where the mouse stays (also called current pulse)

②. period: the period including current pulse and next pulse

③. duty cycle: the width of positive pulse in current period/current period

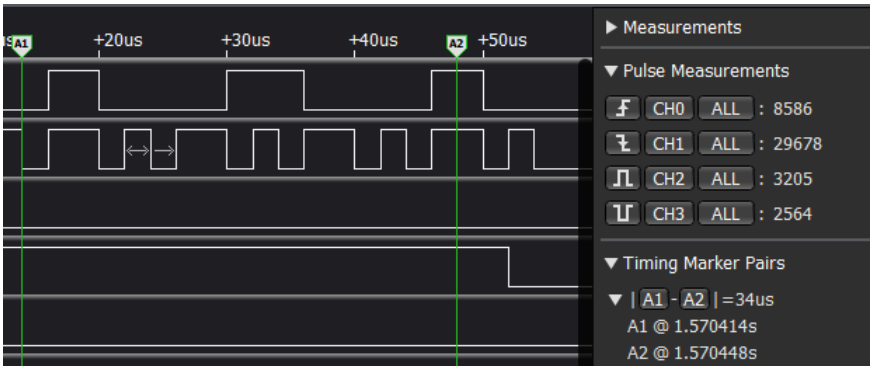
④. frequency: 1/period

Here is an example:



As is shown in the figure above, the mouse lies above a positive pulse of the topmost channel, an arrow will appear above this pulse and the negative pulse next to it. The measurement window will show the following information: the width of this positive pulse is 4us; the period is 14us; the duty cycle of this period is 28.5714286%; the frequency of this period is 71.4285714K.

Add timing makers: if you click the A1 or A2 in the timing marker window with the left button, and move the mouse to the waveform window, the green timing marker line would move with the mouse. And if you press the left button again , this timing marker will stay where you press the button. The corresponding timing marker would disappear by right-clicking while moving with the mouse. When the timing marker is put down, you can right click on the A1 or A2 of the measurement window to close the corresponding timing marker or use the left mouse button to click on the down timing marker, then right-click again to close.

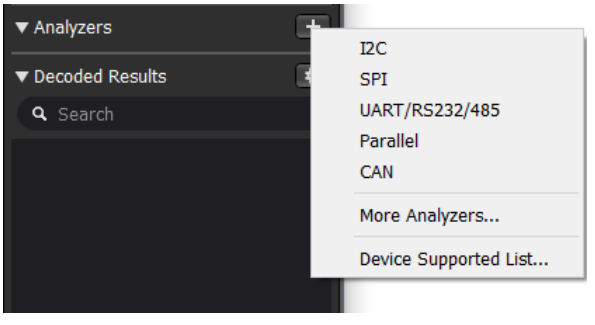


A1 and A2 represent the time where A1 and A2 stay. |A1-A2| is the time difference of these two timing markers, and the positive pulse shows the number of positive pulse between A1 and A2.

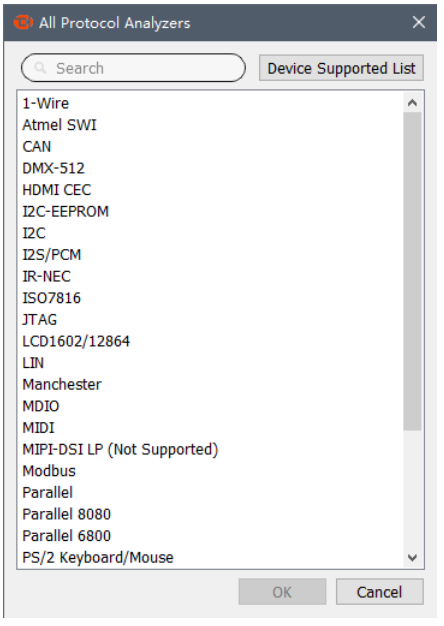
6. Analyzers

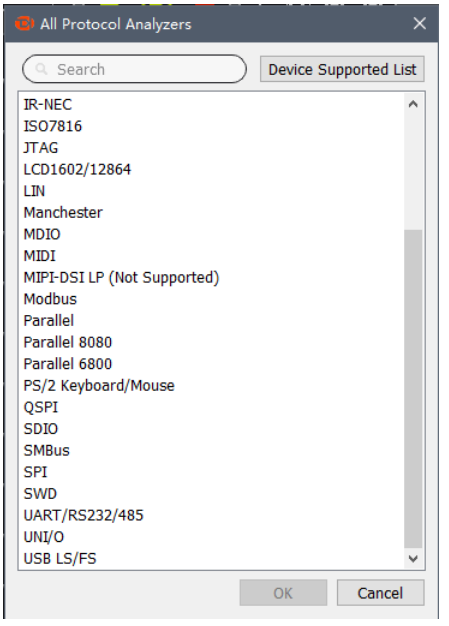
If the signals conforms to standard protocols like UART, I2C, SPI which are supported by Kingst VIS, besides displaying the waveform and some measurement data, the software could analyze the data to get the specific data according to protocol specifications.

Click the button on the right side of the Analyzers settings, and the software will list the most recently used protocols, as shown in the figure below.

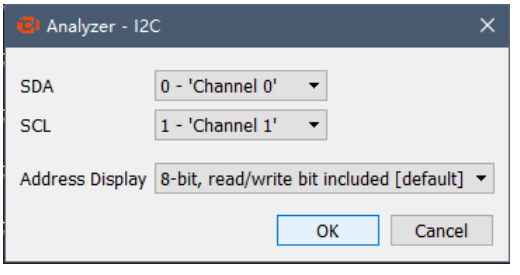


Click the “More Analyzers”, and all the supported analyzers will be listed, as shown in the figure below.

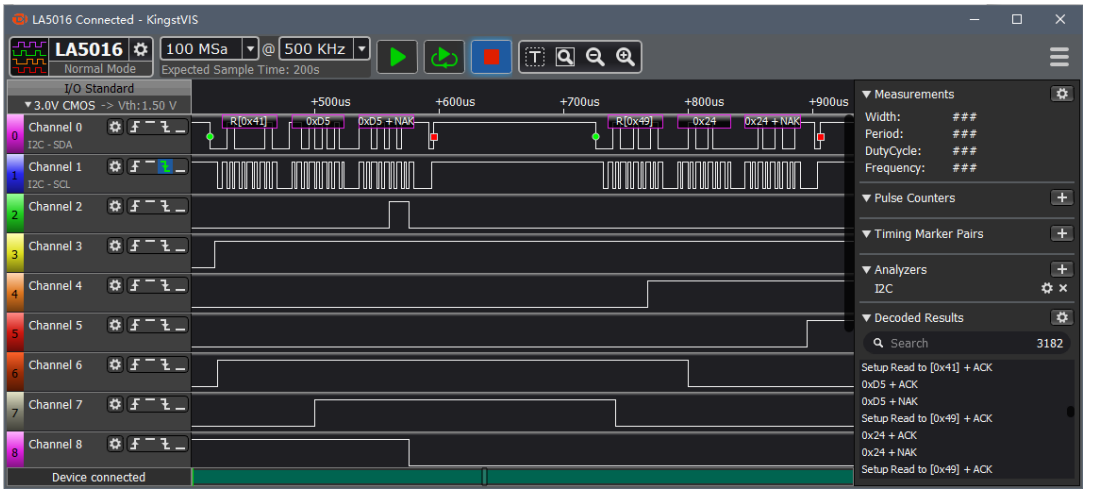




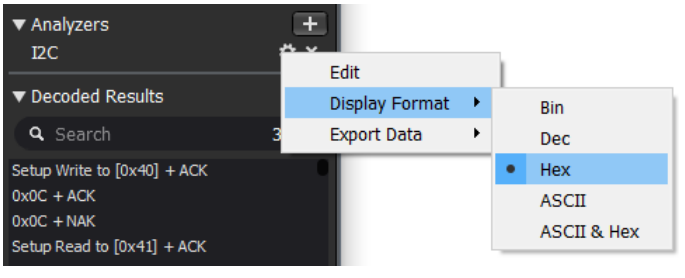
If channels 0 and 1 are used for I2C bus signal, and we have pressed the I2C in the menu, then we will get the settings dialog of I2C analyzers as the figure below:



In the dialog box that is displayed, set SDA to channel 0 and SCL to channel 1. Click “OK”. The corresponding channel names in the protocol resolver are automatically displayed on channel 0 and channel 1. Then the software will analyze channels 0 and 1 according to I2C protocol. After the analyzing process, the SDA channel in the waveform window will show the analyzed data, and the analyzed results will appear in the “Decoded Results” window, where you can find and locate some data. The analyzed data is shown as the figure below:



The data is displayed in hexadecimal by default. If you want to change the format, please press the button in the analyzer bar, and select the format required in the menu “Display Format”. As is shown in the below figure, the software supports Bin, Dec, Hex, ASCII and Hex.



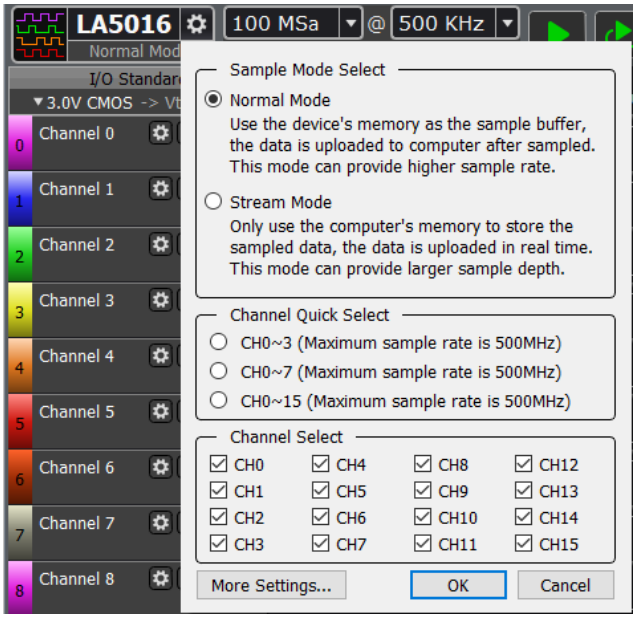
From the menu “Edit”, you can get the analyzer settings dialog to change the settings.

If you want to export and save the analyzed data, you can use “Export Data”(more details in section “Export data”).

If you want to remove an analyzer, you can click ⊠.

7. Sample mode select

Click the ⚙button at the right of the device model to open the settings window of the current device, as shown in the figure below. You can select normal mode or stream mode in the "sample mode select" box. Some models may support only one mode, and there will be no "sample mode select".



①. Normal Mode

This is the default mode with the internal memory model. During the sampling process, the internal memory is used as the sampling buffer, and the data is uploaded to the computer after the sampling is completed. In this mode, the sampling rate is not limited by USB bandwidth, and all channels can be sampled at full speed at the same time.

②.Stream Mode

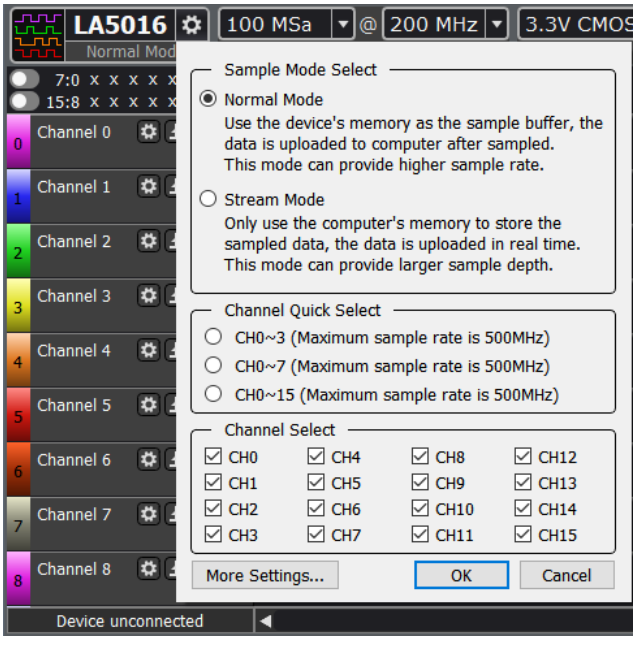
This mode uses computer memory to store sampling data, the data needs to be uploaded to the computer in real time. This is the default mode of LA1010. Limited by USB bandwidth, this mode can not provide full speed sampling under the full channel, but when the computer memory capacity is large enough, it can provide a larger sampling depth to achieve a longer sampling time.

8. Channels settings

①. Channels enable

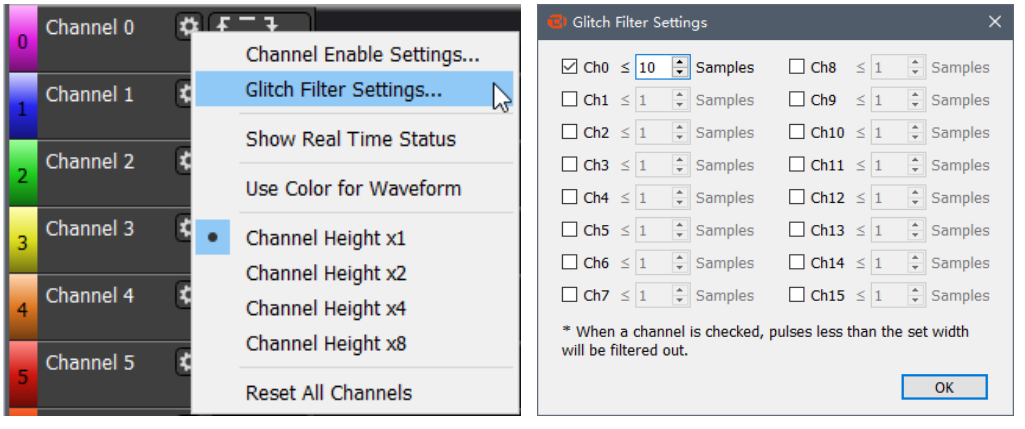
There are usually no less than 8 channels in the logic analyzers, but most of the time, we only use some of the channels. In the previous I2C bus example, only 2 channels are used. To make the GUI simpler and easy to use, we could close the channels that are not used.

Click the button at the right of the device model to open the setting window of the current device, as shown in the figure below. You can enable or disable the corresponding channel in the “Channel Select” setting box. The “Channel Quick Select” box can quickly enable or disable channels 0 ~ 3, 0 ~ 7 and 0 ~ 15.



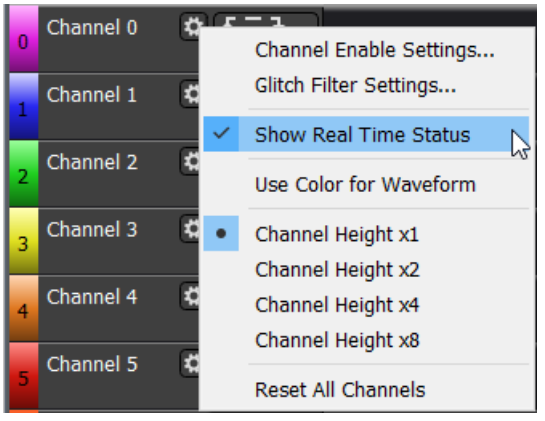
②. Glitch filter

As shown in the following figure, click the setting button on the right of the channel name to set glitch filter through the menu item. Check the channel that needs to be glitch filter, and then set the judgment basis of glitch (that is, the pulse signal less than 'N' samples will be regarded as glitch). After clicking OK, the pulse with the width less than the set value will be filtered out.



③. Show real time status

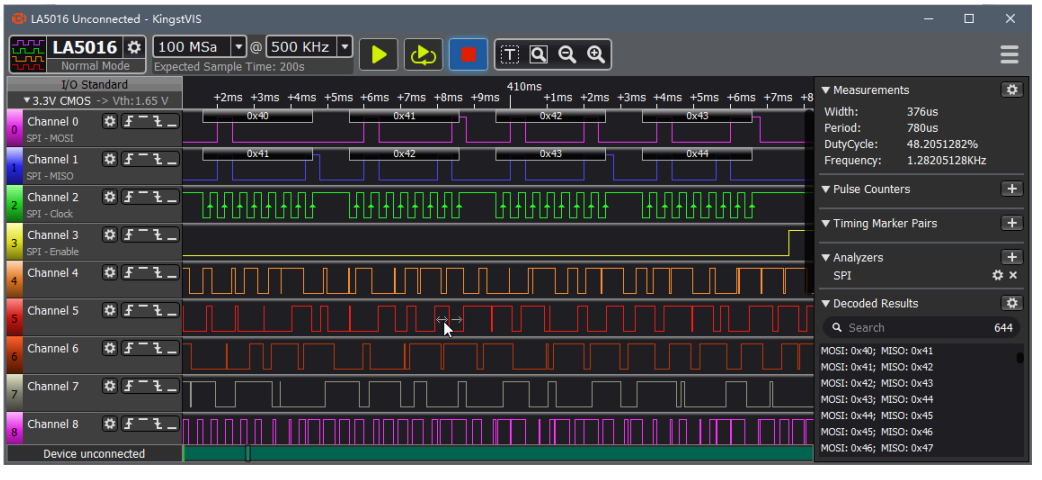
The software can display the real time status of all channels in the channel bar, so that users can directly see the level or rising / falling edge of the channel without sampling. Click the button on the right side of the channel name to pop up the current setting menu, as shown in the figure below, you can select to turn on / off the display of channel real time status.



The specific effect is shown in the following figure: when there is signal change on the channel, the green light will be on; when there is no signal change on the channel, the low level will be displayed as “0”, and the high level will be displayed as “1”; when the device is not connected, no information will be show.

④.Use Color for Waveform

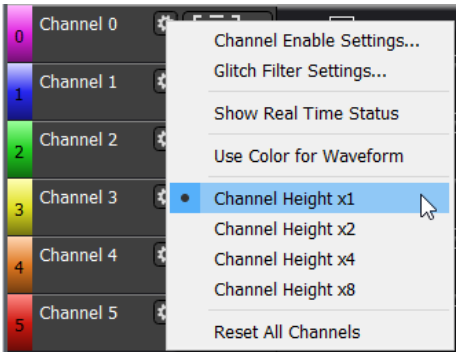
By selecting the “Use Color for Waveform” option in the menu, the waveform line drawn in the software waveform display window can use the color of this channel, as shown in the following figure.



⑤. Set the height of channels

The height of all channels would be set to the same value by default, but if you want to enable or

highlight only some channels, you could set the height of these channels separately. As is shown in the below figure, you could click the settings button in the right of the channel name, and it will pop up menu, you could set the height of this channel to 1,2,4 and 8 times of standard size. In this way, these channels will be highlighted.



Click “Channel Enable Settings” in the menu, you could enable or close the appropriate channel in the Settings box.

Click “Reset All Channels” in this menu, you could reset the channels to default state, for example default height, name, triggering condition etc.

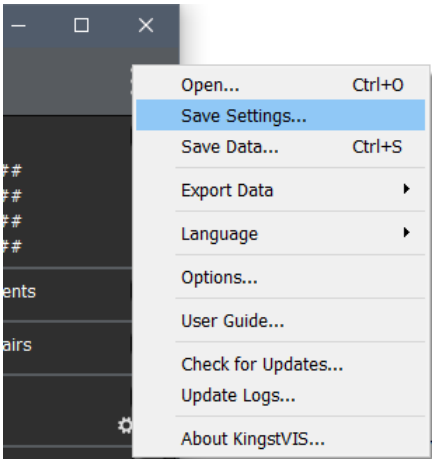
⑥. Edit channel name

If you want change the channel name, as is shown in the below figure, just left click the channel name and input a new name.

9. Save the settings and data

①. Save the settings: you could save the settings into files for later use. And if you want to use these settings, just load them, saving the time to reset the parameters.

To do so, press the main menu button on the top right corner, and select “Save Settings...” in the menu, as is shown the below figure.



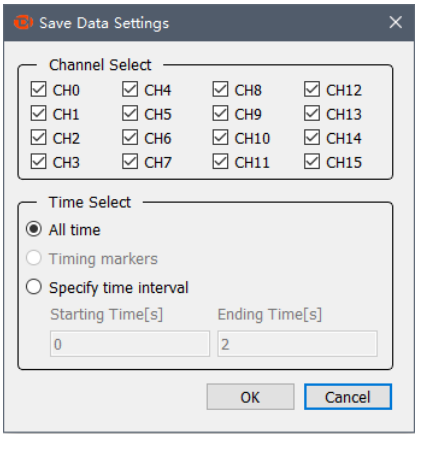
A file saving dialog would appear, and you should specify file path and name, and then press "Save". The channel settings and analyzer settings will be save into this file. If you want to use the same settings, just press "Open…" in the main menu, locate the file in the file dialog and open it.

The extension name of the settings file is "kvset".

②. Save the data: when the device has completed one sampling process, you can save the data and settings together for future use.

You could click the main menu button in the top right corner of the software, and then click "Save data..." in the main menu,and it will pop up a dialog like the one shown below. In the "Channel selection" box of the dialog box, you can select which channels need to be exported, and the default is to select all enabled channels.Then, in the "Time Select" box, you can select the data for which period of time to export, including "All time", and the "Specified time interval" that can be manually set at the start and end time, and the "Between timing markers A1 and A2". After setting, click "OK", select the path and file name to be saved, then you can save the current sampling data and settings. Use "Open..." to open the saved data again.

The extension name of the data file is "kvdat".

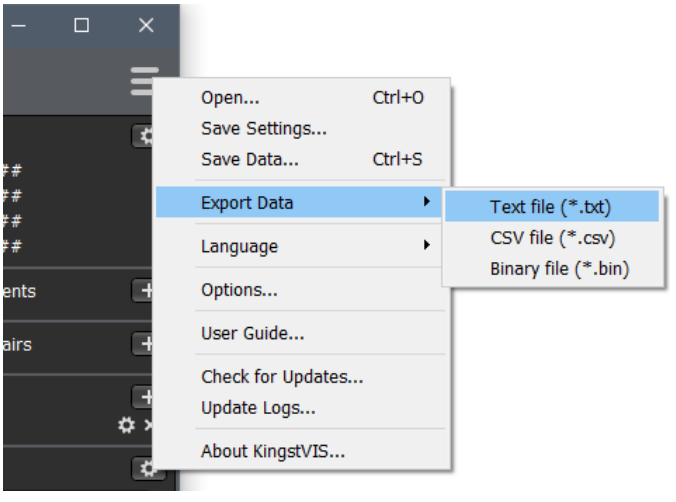


10. Export the data

Kingst VIS supports two data export functions: export original sample data and export the data analyzed by analyzers.

①. Export sample data

As is shown in the below figure, exporting original sample data is done by “Export data…” in the main menu.

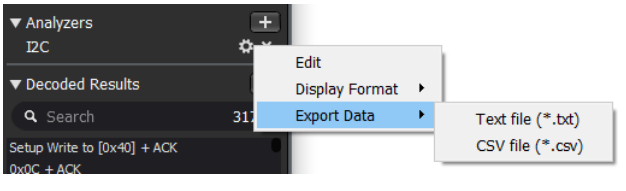


You could export the data as txt, csv, bin. The file types could be opened and edited with other software, for example, open csv with Excel to find and organize data, and import bin with Matlab for further analysis.

②. Export analyzed data

If a analyzer has been added, and some data has been successfully analyzed, you could export and save the analyzed data of this analyzer.

On the right side of the analyzer, you could see a settings button, and if you press it and select “Export Data” in the popped menu, the analyzed data could be exported as txt or csv file, which could be opened with notepad or Excel for checking and analyzing.



The figure below shows what is the analyzed data is like for UART, I2C and SPI. And in this file, we could find time coordinate, packet sequence number and analyzed data.

UART Protocol analysis

文件(F) 编辑(E) 格式(O) 查看(V) 帮助(H)

Time [s]	Value	Parity Error	Framing E
0.001041666	0x00		
0.003125000	0x01		
0.005208332	0x02		
0.007291666	0x03		
0.009375000	0x04		
0.011458332	0x05		
0.013541666	0x06		
0.015625000	0x07		
0.017708332	0x08		
0.019791666	0x09		
0.021875000	0x0A		
0.023958332	0x0B		
0.026041666	0x0C		
0.028125000	0x0D		
0.030208332	0x0E		
0.032291666	0x0F		
0.034375000	0x10		
0.036458332	0x11		
0.038541666	0x12		
0.040625000	0x13		
0.042708332	0x14		
0.044791666	0x15		
0.046875000	0x16		
0.048958332	0x17		

IIC Protocol analysis

文件(F) 编辑(E) 格式(O) 查看(V) 帮助(H)

Time [s]	Packet ID	Address	Data	Read/
0.001342500	0	0x40	0x0C	Write,ACK
0.001397500	0	0x40	0x0C	Write,NAK
0.001727500	2	0x41	0x2B	Read,ACK
0.001782500	2	0x41	0x2B	Read,NAK
0.002037500	4	0x49	0x1E	Read,ACK
0.002092500	4	0x49	0x1E	Read,NAK
0.007222500	6	0x49	0x29	Read,ACK
0.007277500	6	0x49	0x29	Read,NAK
0.008907500	8	0x40	0x10	Write,ACK
0.008962500	8	0x40	0x10	Write,NAK
0.009292500	10	0x41	0x0F	Read,ACK
0.009347500	10	0x41	0x0F	Read,NAK
0.009602500	12	0x49	0x17	Read,ACK
0.009657500	12	0x49	0x17	Read,NAK
0.014787500	14	0x49	0x36	Read,ACK
0.014842500	14	0x49	0x36	Read,NAK
0.016472500	16	0x40	0x14	Write,ACK
0.016527500	16	0x40	0x14	Write,NAK
0.016857500	18	0x41	0x0E	Read,ACK
0.016912500	18	0x41	0x0E	Read,NAK
0.017167500	20	0x49	0x68	Read,ACK
0.017222500	20	0x49	0x68	Read,NAK
0.022352500	22	0x49	0x5C	Read,ACK
0.022407500	22	0x49	0x5C	Read,NAK

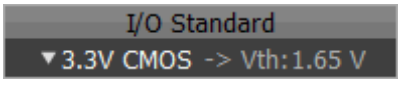
SPI Protocol analysis

文件(F) 编辑(E) 格式(O) 查看(V) 帮助(H)

Time [s]	Packet ID	MOSI	MISO
0.000004800	0	0x00	0x01
0.000008800	0	0x01	0x02
0.000012800	0	0x02	0x03
0.000016800	0	0x03	0x04
0.000029600	1	0x04	0x05
0.000033600	1	0x05	0x06
0.000037600	1	0x06	0x07
0.000041600	1	0x07	0x08
0.000054400	2	0x08	0x09
0.000058400	2	0x09	0x0A
0.000062400	2	0x0A	0x0B
0.000066400	2	0x0B	0x0C
0.000079200	3	0x0C	0x0D
0.000083200	3	0x0D	0x0E
0.000087200	3	0x0E	0x0F
0.000091200	3	0x0F	0x10
0.000104000	4	0x10	0x11
0.000108000	4	0x11	0x12
0.000112000	4	0x12	0x13
0.000116000	4	0x13	0x14
0.000128800	5	0x14	0x15
0.000132800	5	0x15	0x16
0.000136800	5	0x16	0x17
0.000140800	5	0x17	0x18

11. Threshold voltage setting

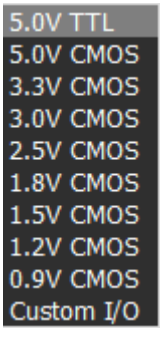
The threshold voltage of most device is adjustable, and you can find the drop-down list of threshold settings in the toolbar, as shown below.



After you left-click the

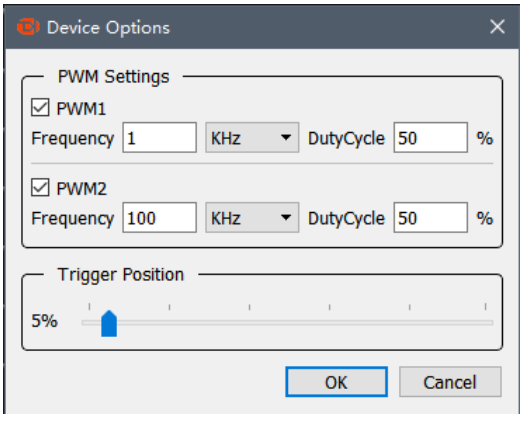
icon with the mouse, a drop-down list appears, as shown below. Then you can select the threshold voltage you want to set, after the selection is completed, it will be displayed below the

icon, as shown in the above figure. In addition, you can also customize the threshold voltage, select “User-defined” in the drop-down list and enter the threshold voltage at “Vth:” .



12. PWM and Trigger Position Settings

Click the button at the right of the device model to open the setting window of the current device, and then click the “More Settings...” button below to open the dialog box as shown below.



PWM generator:There are two PWM waveform generators in the logic analyzer except LA1002, and they can generate square wave whose duty cycle could be adjusted. When the device which support PWM export is connected to the software, by default, PWM1 output 1KHz square wave with 50% duty cycle; PWM2 output 1KHz square wave with 50% duty cycle.

Select on or off PWM1 and PWM2 in the “PWM Settings” box. If it is turned on, we could edit the frequency and duty cycle. After the settings are complete, just press the “OK” button, and the software will generate the PWM signal with the new configuration.

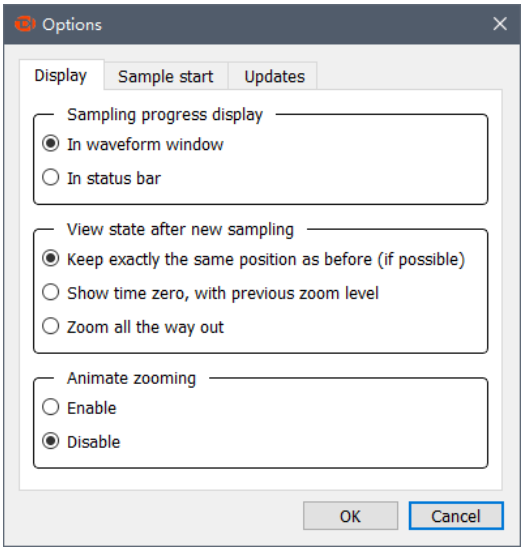
Trigger position setting: The position of the trigger signal during the total sampling period.

For example, in the above figure, the setting value in the “Trigger Position” box is 5%. If the total sampling time is 1s, when the sampling time reaches 0.05s, it is detected whether the set trigger condition is satisfied, and if it satisfies, it continues to sample until the total sampling time, and if it is not satisfied, it waits for the trigger.

Set the trigger position by sliding the slider in the “Trigger Position” box.

13. The “Options...” in the main menu

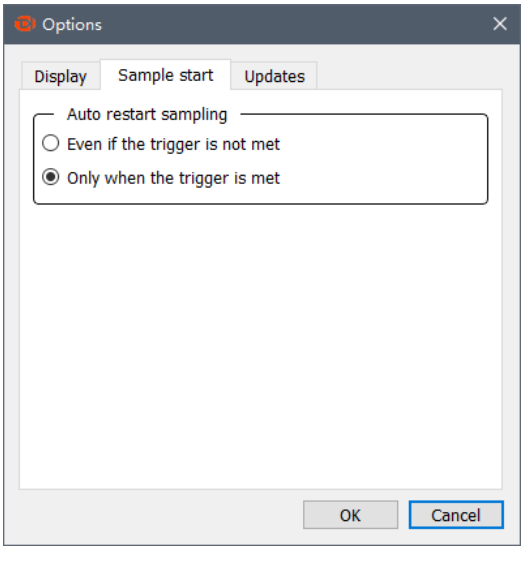
Click the main menu button in the top right of the software, and then click the “Options...” button to bring up the dialog box as shown below.



In the “Sampling progress display” box, select the first option to display the sampling progress in the waveform window; select the second option to display the sampling progress in the status bar in the lower left of the software.

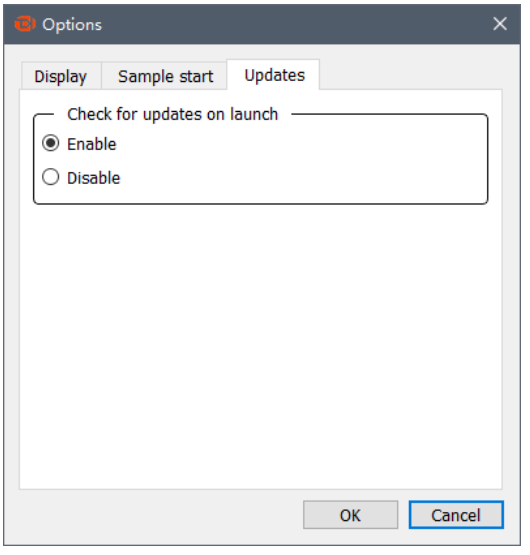
In the “View state after new sampling” box, the first option maintains the previous display scale and the position of the timeline. The second option maintains the previous display scale and moves the time zero point to the current window. The third option causes each sampled waveform to be scaled to display in one screen.

When “Enable” is selected in the “Animate zooming” box, the waveform lines have animated effects when scaled in the software waveform display window. By default, “Enable” is selected.



In the “auto repeat sampling” box, when the first option is selected, whether or not the trigger is

met, the waveform will be repeatedly sampled, that is, the waveform will be continuously refreshed; when the second option is selected, only when the trigger condition is met, the waveform will be repeatedly sampled and refreshed.

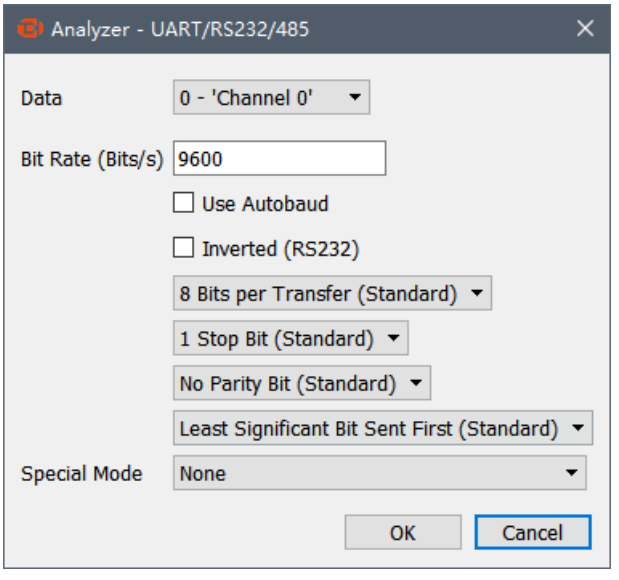


When “Enable” is selected in the “Check for updates on launch” box, the updates is automatically checked when you open the software. If you select "Disable", it is not checked. By default, "Enable" is selected. At the same time, you can manually check for updates, select “Check for updates...” in the main menu to check for software updates.

Ⅴ. Settings for standard protocols

1. UART/232/485

For standard UART, RS232 and RS485, they have the same timing definitions of the physical layer, so they share the same analyzer, and the figure below shows the setting dialog of UART/232/485 analyzer.



1st item, select the channel to use.

2nd item, set the baud rate, and the baud rate here should match that in actual use.

3rd item, use the auto baud, and the software could identify the baud rate automatically. In case of the baud rate is unknown, this option should be enabled. But the accuracy of automatic identification depends on actual signal, and the result could be incorrect.

4th item, select whether or not to invert. Usually, the inversion is only applicable to the standard RS232 level (Because RS232 specifies that the positive level is 0, the negative level is 1, so it is necessary to invert the phase when analyzing.). If it is not inverted, so it is suitable for UART, RS485, etc.

5th item, select the number of data bits, and it is 8 most of the time.

6th item, select the number of stop bits, and there are 3 options: 1, 1.5 and 2.

7th item, set the parity bit, and there are 3 options: no parity, even parity and odd parity..

8th item, set the bit order in data transfer, and the options could be LSB(Least Significant Bit) Sent First and MSB(Most Significant Bit) Sent First.

9th item, set bit 9 as address flag in multiple machine communication or not, and by default, None. When this mode is actually used (seldom used, note that it is different with RS485 multiple machine communication), it could be used for address byte flag.

Please note that if the signal under test is differential signal like RS485 and RS232, there are 3 ways to connect the wire:

①. The GND channel of the logic analyzer is connected to the GND of the test system, and 2 signal channels are connected to RXD and TXD pins of the level shift chip respectively.

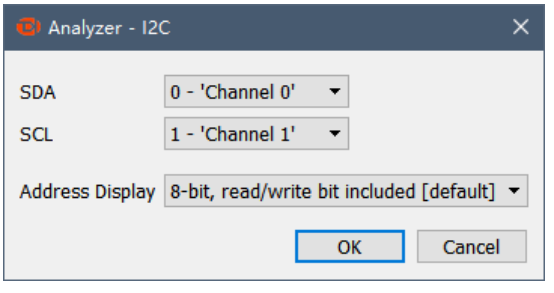
②. The GND channel of the logic analyzer is connected to the GND of the test system, and 1 signal channel is connected to the RS485-A.

③. Connect the RS485 bus to a module that transfer RS485 to TTL, and the GND and a signal channel of the logic analyzer are connected to the GND and signal export end of the system under test.

Most of the time, all these 3 ways could be used to sample the signal, but according to RS485 specification, the voltage which the A-B ends could identify is between 0.2～6V. In complicated situations, such as master with many slaves or long wire, the difference of the bus end could be too little, and the logic analyzer could not identify the signal level correctly with method 2. So method 1 and method 3 are recommended if conditions permit.

2. I2C

The setting dialog of I2C analyzer is shown below:



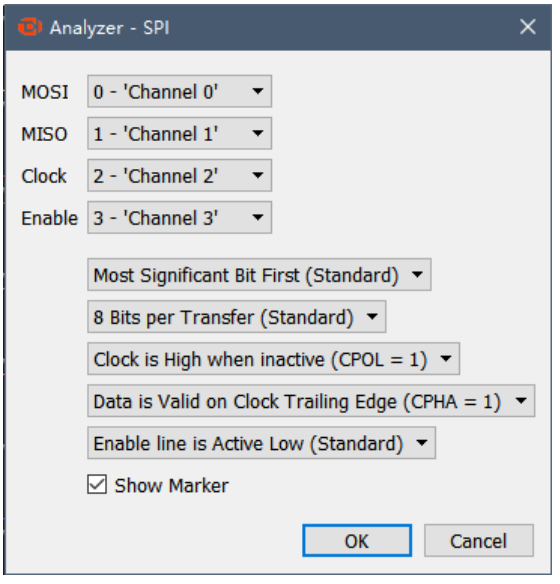
1st item, the channel used for SDA signal (data)

2nd item, the channel used for SCL signal (clock)

3rd item, the way to display the address byte. For I2C protocol, every communication is started with addressing operation, and this byte contains 1-bit read / write flag and device address which is 7-bit wide. And there are three options to display: to display as a whole (8-bit, read / write bit included); to display as a whole but the read / write flag is 0(8-bit, read/write bit set as 0); only display 7-bit address (7-bit, address bits only).

3. SPI

The setting dialog of SPI analyzer is shown below:



1st item, the channel for MOSI signal (master out slave in)

2nd item, the channel for MISO signal (master in slave out)

3rd item, the channel for CLOCK signal (clock)

4th item, the channel for ENABLE signal (enable)

5th item, transmission mode of data bits: MSB(Most Significant Bit) First or LSB(Least Significant Bit) First, usually MSB First.

6th item, data length for one transfer, usually 8 or 16 bits.

7th item, idle state of the clock. CPOL = 0: the clock wire remains low in idle state. CPOL = 1: the clock wire remains high in idle state.

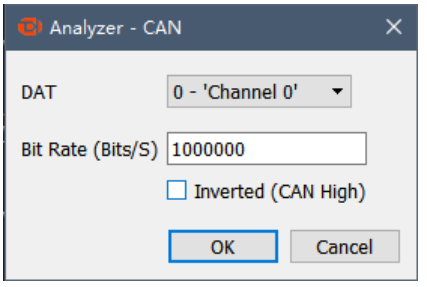
8th item, the edge in which data is latched. CPHA=0: data latched in first clock edge. CPHA=1: data latched in second clock edge.

9th item, the active level of enable signal: active low(Enable line is Active Low) or active high(Enable line is Active High).

10th item, select whether or not to display decode marker.

4. CAN

The setting dialog of CAN analyzer is shown below:



1st item, the channel to use.

2nd item, baud rate in communication.

3rd item, set the logic to invert, need to invert when measuring CAN-High signal line directly.

Please note that the signal from CAN bus is differential, there are 3 ways to measure CAN signal:

①. The GND channel of the logic analyzer is connected to the GND of the test system, and 2 signal channels are connected to RXD and TXD pins of the level shift chip respectively.

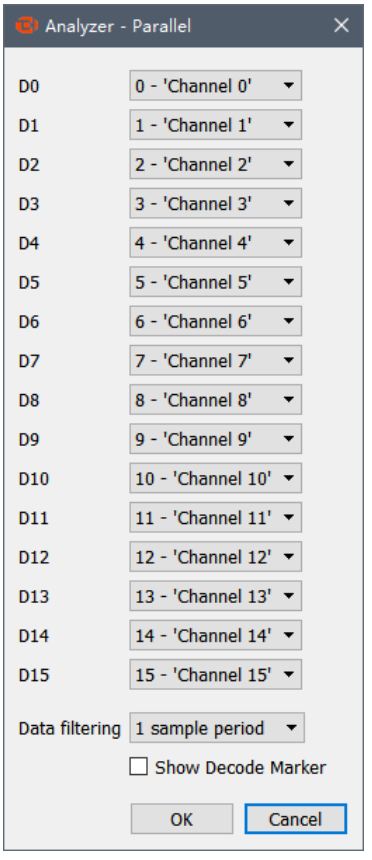
②. The GND channel of the logic analyzer is connected to the L end of CAN bus, and 1 signal channel is connected to the CAN-H, and now you should select “Inverted (CAN High)”.

③. Connect the CAN bus to a module that transfer CAN to TTL, and the GND and a signal channel of the logic analyzer are connected to the GND and signal export end of the system under test.

Most of the time, all these 3 ways could be used to sample the signal, but according to CAN specification, the voltage between the H-L ends is 0V and 2V. In complicated situations, such as master with many slaves or long wire, the difference of the bus end could be too little, and the logic analyzer could not identify the signal level correctly with method 2. In addition, the GND channel of method 2 need to be connected to the CAN-L end, but if other signals need to be tested at the same time, the grounding could be confusing. So method 1 and method 3 are recommended if conditions permit.

5. Parallel

Parallel analyzer, the sampling values of multiple channels at the same time are combined into a 16bits data. When the level of any channel changes, a data is generated. It’s setting window is as follows:



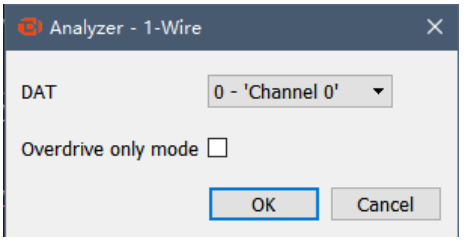
1st-16th items, the channels used for 16 parallel ports.

17th item, select the number of sample periods for data filtering. Because in actual use, it is impossible to achieve complete synchronization for changes of multiple channels, so when data changes, it is possible to generate false intermediate transition values. This option is used to filter out this intermediate status. The signal must be kept unchanged for more than n sampling periods before it is recognized as valid data.

18th item, select whether or not to display decode marker.

6. 1-Wire

The setting dialog of 1-Wire analyzer is shown below:

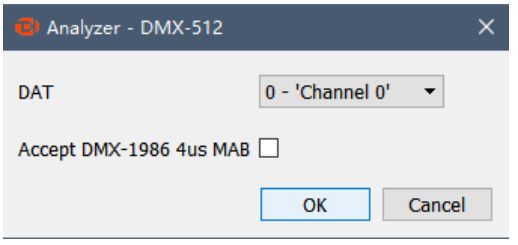


1st item, the channel to use.

2nd item, select this item in high speed mode.

7. DMX-512

The setting dialog of DMX-512 analyzer is shown below:

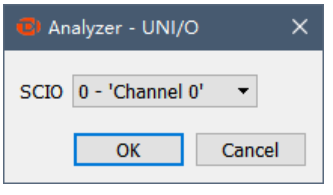


1st item, the channel to use.

2nd item, accept DMX-1986 4us MAB signal or not.

8. UNI/O

The setting dialog of UNI/O analyzer is shown below:



1st item, the channel to use.

Only the above 8 kinds of protocol parsers have been briefly introduced. Other protocols supported by the logical analyzer can be used by you and you can contact us if you have any questions.

9. User-defined protocol analyzer

Besides the standard protocol analyzers in the software, the API functions provided by the software allow users develop their own analyzers. The software API and user manual could be downloaded from the link below:

https://www.HKLRF.com

VI. FAQs

1. Driver installation fail with the device connected to computer

First, Kingst VIS should have been installed before connecting the device to computer. If the software has not been installed, the OS could not find the driver, so the installation would fail. And it would be a good idea to install the software without the hardware connected to the computer.

Second, the device driver is installed automatically by installation program during installing process. If the driver installation is blocked unintentionally during installation process, or it is not installed correctly due to other reasons, when you connect the device for the first time, the driver installation would fail too. In this case, you can find the unknown hardware devices in the device manager, click the right button to install the driver program again. Please select manual installation. The driver program is located in “installation directory \ Driver \ ”, and select the right directory based on your operating system.

2. Identification fail or work unstably with the device connected

When the logic analyzer works at full speed, there would be a lot of the current consumed (maybe more than 500mA). So if the USB port of the computer can not supply sufficient power, the device could be identified incorrectly or work unstably. To solve this, the laptop users could try to switch to the USB port on the other side, and the desktop users must use the USB port behind the tower box. If the USB-HUB is being used, please connect the device USB port directly.

3. Signal glitches appear on individual channels

There are two cases when signal glitches appear: the unused channels are floating, or several signals with high speed are sampled simultaneously.

It is normal that the glitch appears on unused channels, this is because the floating channel wire is like an antenna, and it will transmit weak and alternating signal, and this would result in glitches. We could hide such channels, or keep them and the signal channels at a longer distance. And we should check the grounding of the logic analyzer and the system under test.

And for the glitches which appear when measuring multiple channels with high speed, the reason for that has been explained in the section “Multipoint grounding to increase accuracy”, and the method to handle this is multipoint grounding

4. The actual sample time is less than expected

The logic analyzer is designed with a large-sized memory, to store the sample data temporarily. And through compression algorithm the depth is further extended. With this compression algorithm, the current and last sample data would be compared, if they are not the same, a new sample data

would be generated; otherwise, only the count for last state would be incremented and no new sample data would be generated. And as a result of this, if the signal is discontinuous (for most communication system) or change slowly, the sample depth would be extended greatly. However, if the signal change rapidly the extension effect would not be so obvious. This is why the sample time is less than expected when the sample depth is set above 100M (all the depth items with *) and the signal changes rapidly.

5. Automatic update failed

The software supports auto update, when a new version is released, a message would tell the user to update. However, because of system permission and security policy of the operating system, the auto update could fail. It this happens, the user could download the latest Kingst VIS software from our website, and after uninstalling the software, the user could install the new version.

The download link of the latest Kingst VIS: https://www.HKLRF.com

VII. Contact us

Thanks for choosing our product, if you have any questions, please contact us in the following ways, and we will serve you wholeheartedly.

Web: www.HKLRF.com