The Manual of Inverter Host PC Debugging Software

VEDA-in CST

Content

1. System prerequisites

1. It is the prerequisites to install NET3.5 before clients use this software at the first time. Otherwise the software won't be opened. If we use Win 7 systems or above, it doesn't need to be installed.

2. The system currently supports RS485 communication. The baud rate range is between 1200 and 57600.

3. The system currently supports CAN communication.

4. The system supports the debugging of RD05/11/30 host PC.

2. Main interface

Main interface includes some functions such as Menu bar, Toolbar, Function display area, Message bar and Status bar.

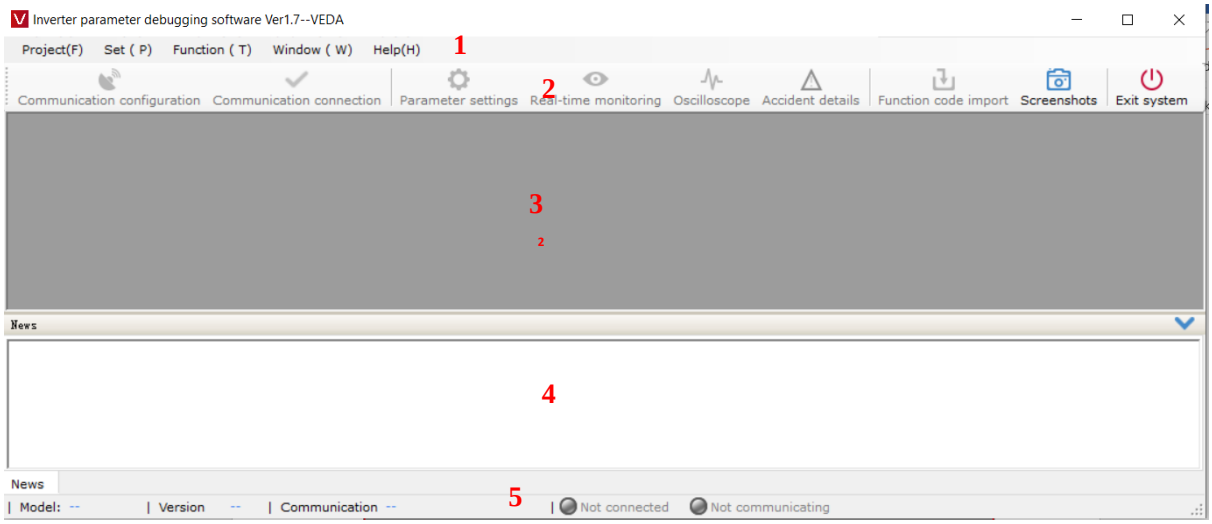


Figure 1

①—Menu bar

Menu bar includes Projects, Settings, Tools, Windows and Help.

- Project: New, Open, Close, Recent Projects (10 newly built projects were provided, it can be opened directly) and Empty All.
- Setting: Communication Configuration, System Configuration:
- Tool: Parameter Reading and Writing, Parameter Monitoring, Digital Oscilloscope, Maintenance.
- Window: Stacked Display, Horizontal Display, Vertical Display.
- Help: Instructions, Update Online, About Us.

②—Toolbar

Toolbar includes Communication Configuration, Communication Connection, Parameter Reading and Writing, Monitoring Parameter, Digital Oscilloscope, Maintenance, Screenshot and Quit.

③—Function display area

Function display area is form container that provides Parameter reading and writing, Monitoring parameter, Digital oscilloscope, Maintenance and so on.

④—Message bar

Message bar includes Message and Control panel.

- Message: Provide all kinds of messages of the operation functions. Blue font means normal condition, red font means abnormal condition.
- Control panel: Provides shortcuts control function.
- Command window: It will be seen under the development mode in order to send and receive instruction test.

⑤—Status bar

Status bar includes current communication and parameters status. There is a hint at the right side of status bar, if new version has been updated.

3. Functions

3.1. Project

The project includes New, Open, Close, Recent projects and so on.

3.1.1. New

Steps [Take RD11 as an example]:

1. Click Menu bar [Project] -> [New]. As shown in figure 2 below.

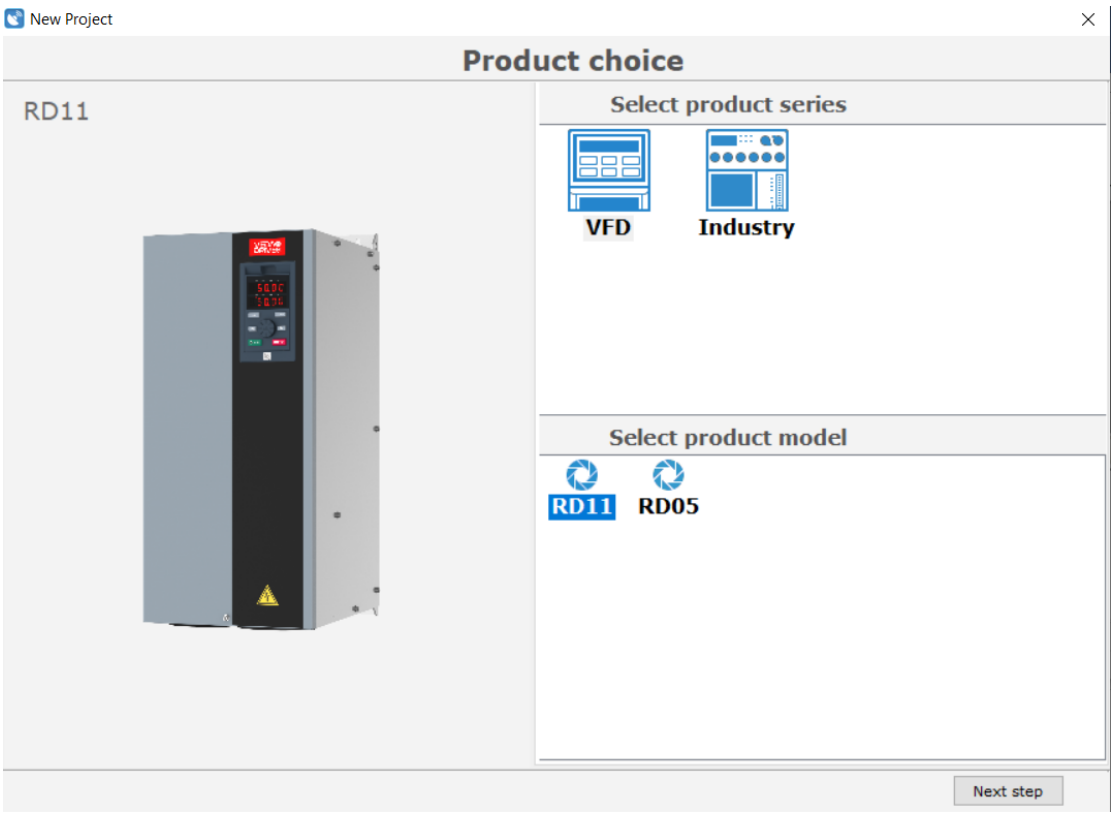


Figure 2

2. Select [Inverter] as products series -> [RD11] -> [Next]. Enter the communication configuration. Item name is required and the communication type: RS485. Other communication parameters should be set according to the actual configuration. As shown in the figure 3 below:

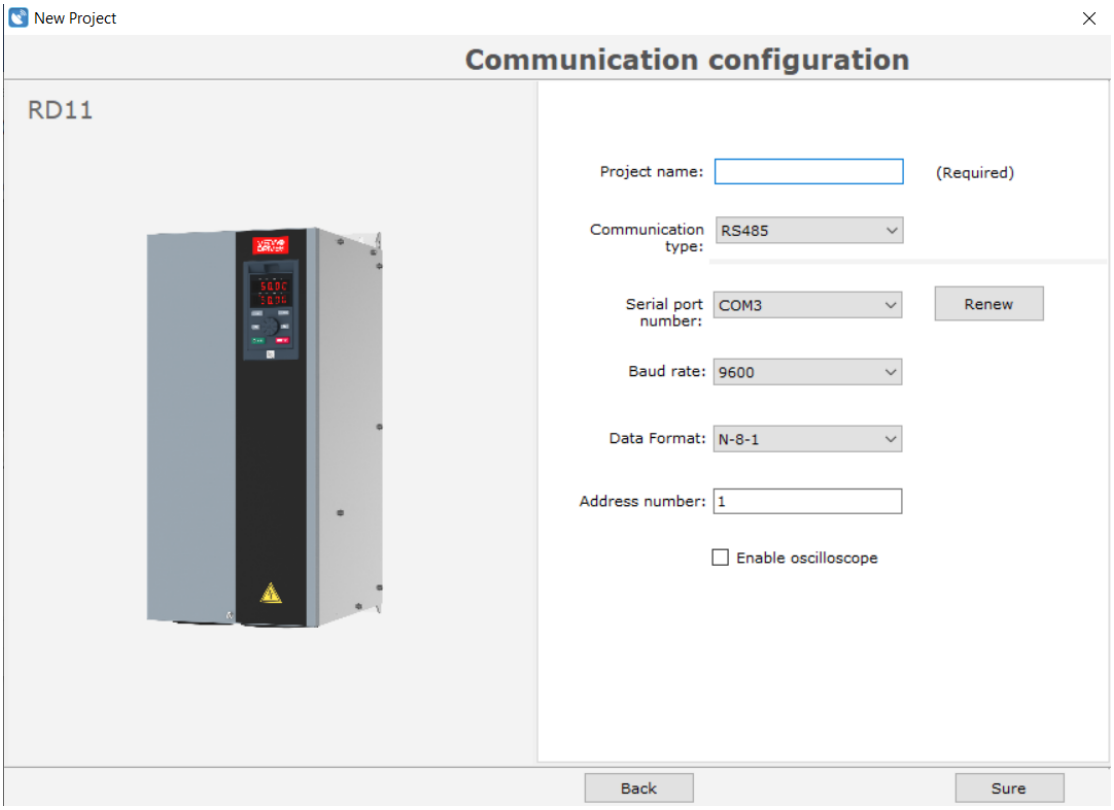


Figure 3

3. Click [OK]. System will save the related information in current project. System runs the communication connection automatically when the device is connected.

3.1.2. Open

Open: Open the existent project file

Steps:

1. Click menu bar [Project] -> [Open] -> [Select current system directory folder **Project**] -> [Select folder 1] -> [Select **1.config**]

3.1.3. Close

Close: Close the current project.

Steps:

1. Click menu bar [Project] -> [Close].

3.1.4. Recent project

Recent project: Listed 10 recently used project files.

Steps:

1. Click menu bar [Project] -> [Recent project] ->[1]

3.1.5. Empty all

Empty all project functions: In order to empty all created projects.

Steps:

1. Click menu bar [Project] -> [Empty all].

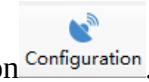
3.2. Configuration

3.2.1. Communication Configuration

Steps:

1. Start communication configuration: (Provide two types)

Method 1: Click menu bar [Configuration] -> [Communication configuration].

Method 2: Click toolbar's icon 

.

2. Communication configuration is shown in the figure 4 as below. Click [OK] after updating the communication parameters.

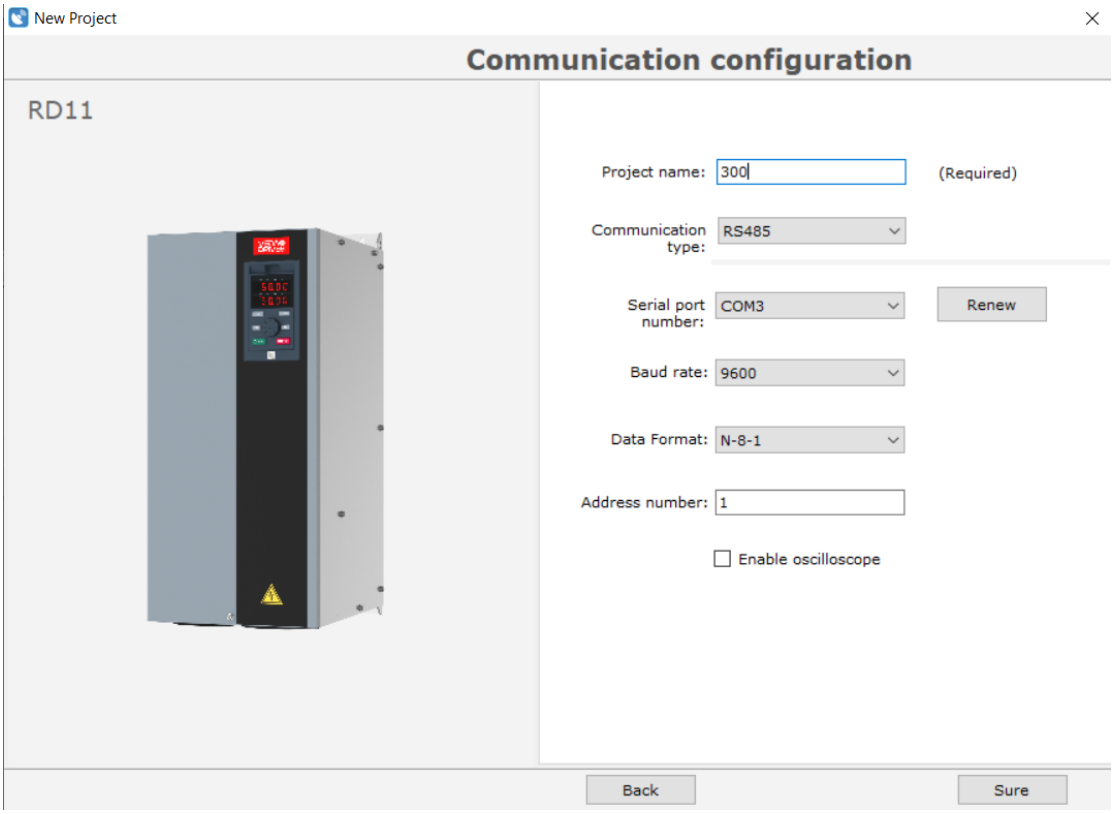


Figure 4

3.2.2. System configuration

Steps:

1. Click menu bar [Configuration] -> [System configuration].
2. The system configuration is shown in figure 5 as below:

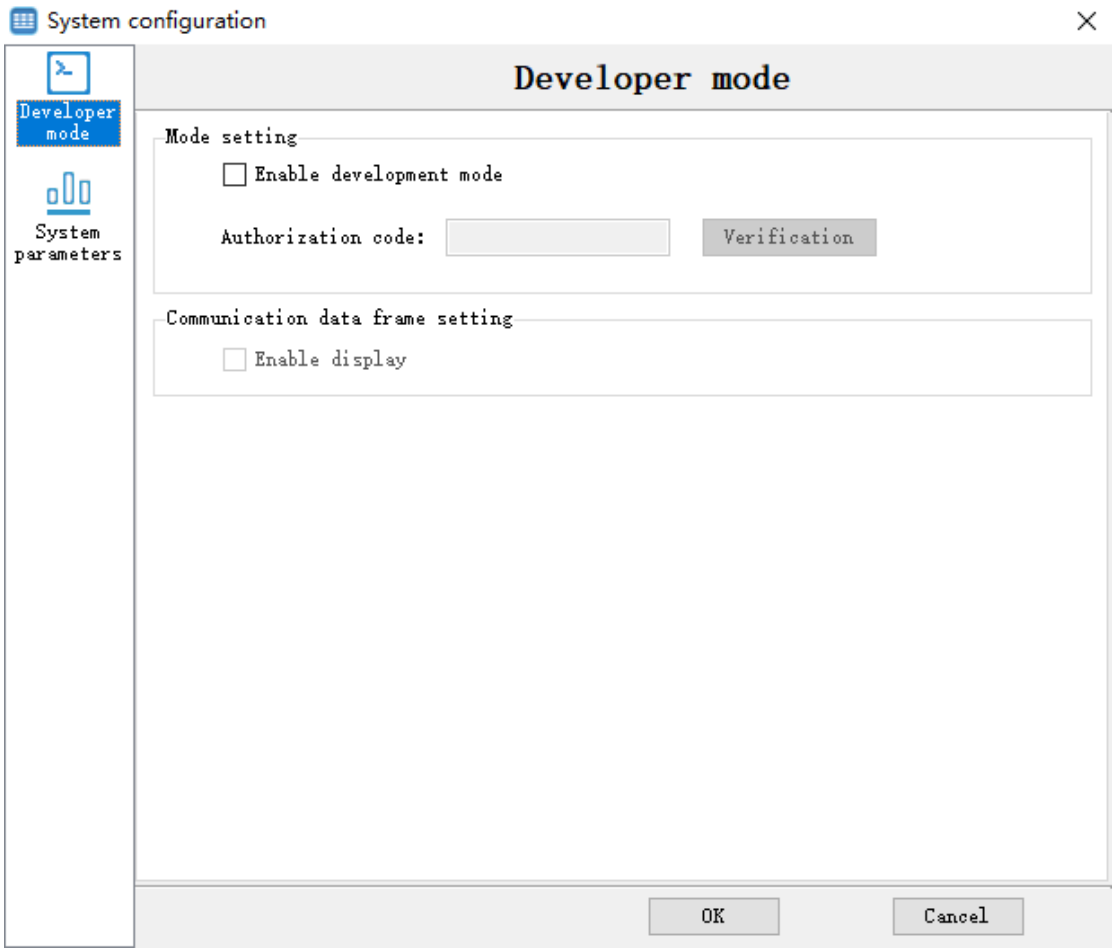


Figure 5

3. Click [Development mode], selection [Start development mode], input correct authorization code, Click [Verify]. Click [OK] if it's finished, some functions could only be seen under development mode.
4. The setting of communication data frame is used in development to display the data of communication.
5. Click [System parameters] and you could choose the default settings to be shown at starting time.

Notes: The instruction of function under development mode can be referred in attachment 1.

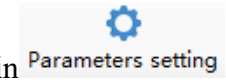
3.3. Parameters reading and writing

The parameters reading and writing includes code reading and writing, leading-in, leading-out.

Steps:

1. Start the interface of parameters reading and writing:(Provide two types)

Method 1: Click menu bar [Tools] -> [Parameters setting]

Method 2: Click toolbar's coin 

.

2. The interface of system parameters reading and writing are shown figure 6 as below:

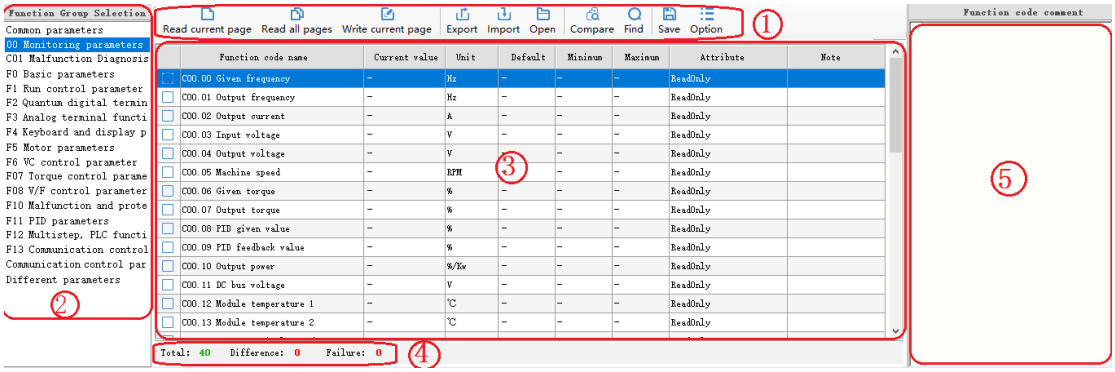


Figure 6

① —Toolbar

Toolbars include reading function code of current page, reading function code of all pages, writing code of current page, lead-out function code in batches, lead-in function code in batches, open function code, comparing function code, option and so on.

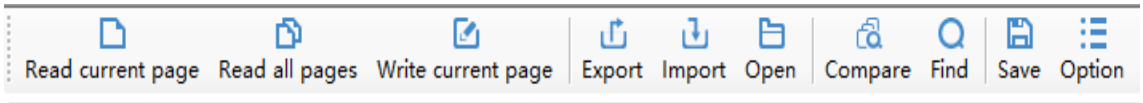


Figure 7

② —Functions

Display different function name in table, add some parameters in common use and different parameters group. It is convenient to classify the function codes. Corresponding function code will be shown in the right side by clicking the function name.

③ —Function code

Function code is the specific function, including the Current status, name, current value, units, default, max value, min value and related attribute information and if you click one certain line, the relevant function code annotations will be shown on the right side.

Statements:

: Ready : Wait : Communication fault

: Cancel : Succeed : Abnormal communication (Data return 0x86/0x83)

④ —Statistics of consequences

The total statistics include the number of current function codes, the number of actual value and different default, the number of abnormal communication.

Notes: Clicking the total number of statistics, you can automatically screen the current corresponding function code.

⑤ —Notes

Notes about related function codes:

3.3.1. Code reading

Function codes can be read singly or in batches.

Steps:

1. Single reading: Selection one function code by clicking right-hand button [Reading current page] -> [Rows selected]

2. Reading in batches: (Provide three modes)

Mode 1: Selecting one function group page. Click toolbar icon Read current page to read current page.

Mode 2: Clicking toolbar icon Read all pages to read all the current pages. At the same time the pop-up dialog box displays the progress of the read function code in the form of progress bar, as shown in Figure 8:

Figure 8

Mode 3: Select one function group page. Select [reading current page] -> [All] by right-hand button.

3.3.2. Function code leading-in/out

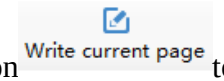
Function code can be written singly, restore default values, write in batches and write into EEPROM.

Steps:

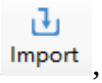
1. Writing in a single: Select one certain function code; click the column twice that is corresponding to the current value. Then it may enter into the editing mode automatically. System will send the writing order automatically after clicking other rows or press the enter button of PC.

2. Restore default: Select one certain function code and select [Restore default] -> [Rows selected] or [All], which could restore the default of the selected function code.

3. Writing in batches: (Provide two modes)

Mode 1: Click toolbar icon 

to write current page.

Mode 2: Click toolbar icon 

, pop up function interface of leading-in in batches. As shown figure 9 as below:

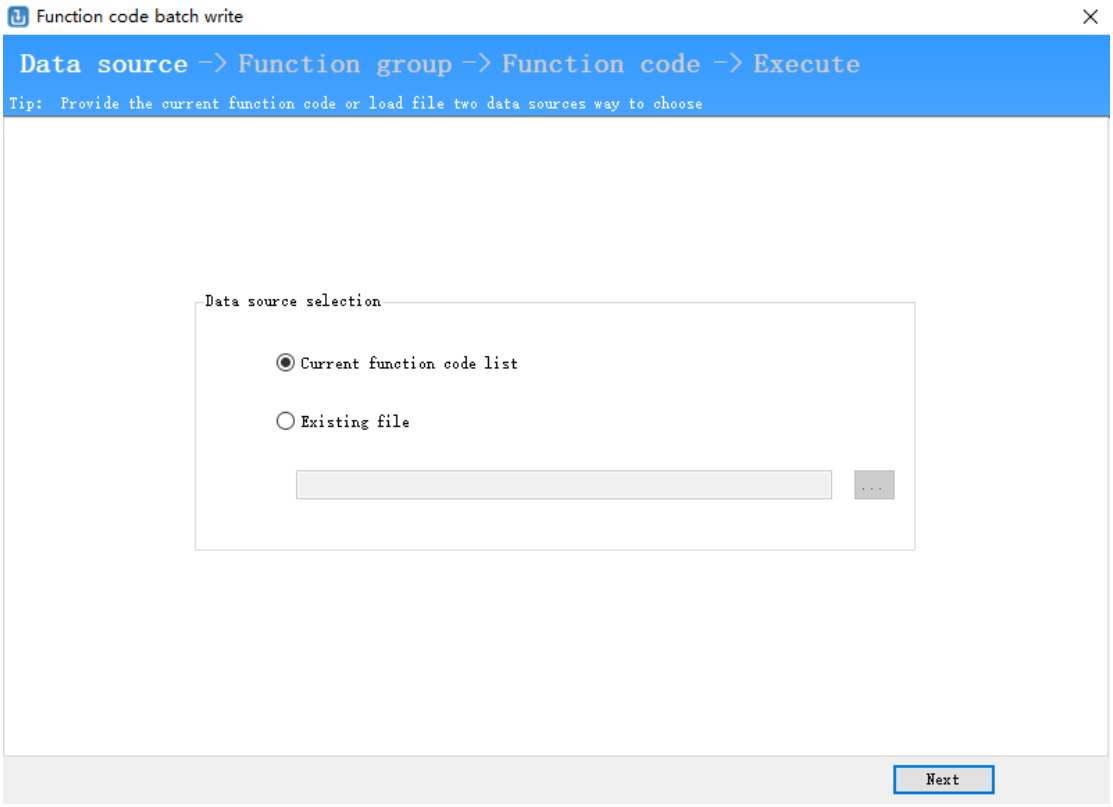


Figure 9

Take steps to finish the lead-in in batches:

- Data source selection [current function code list] -> [Next], as shown in figure 10 as below:

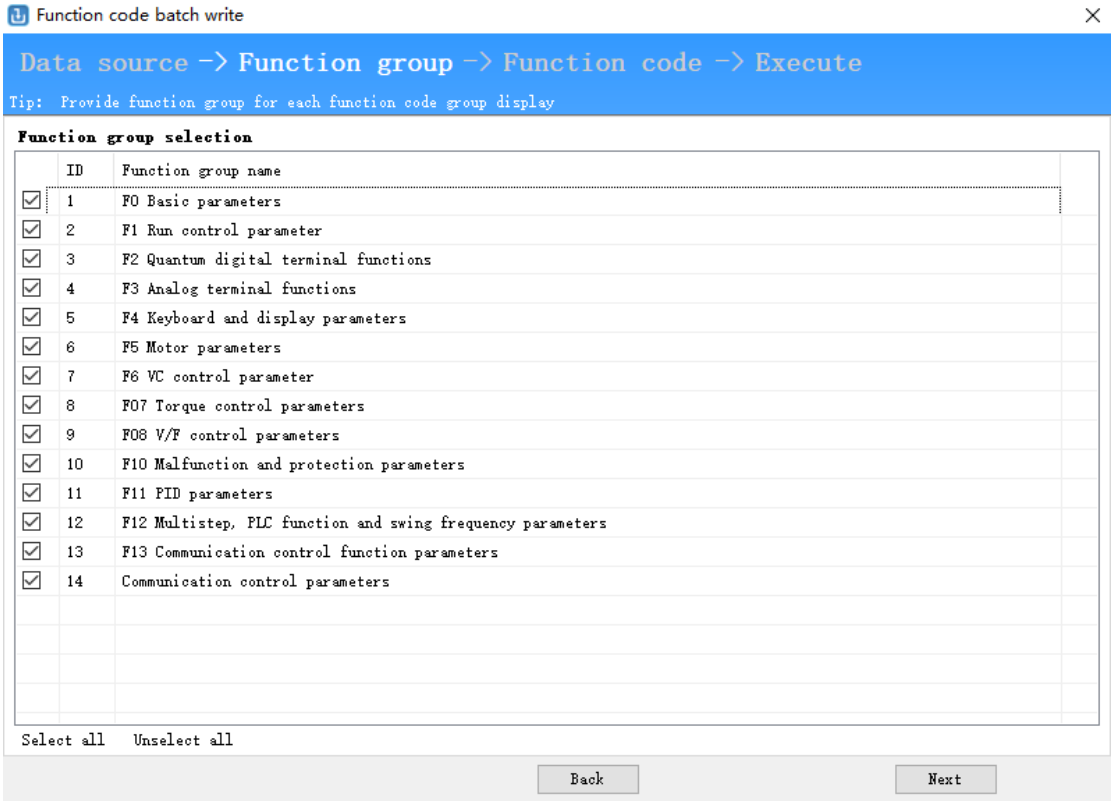


Figure 10

➢ [Function group selection] -> [Next], as shown in figure 11 as below:

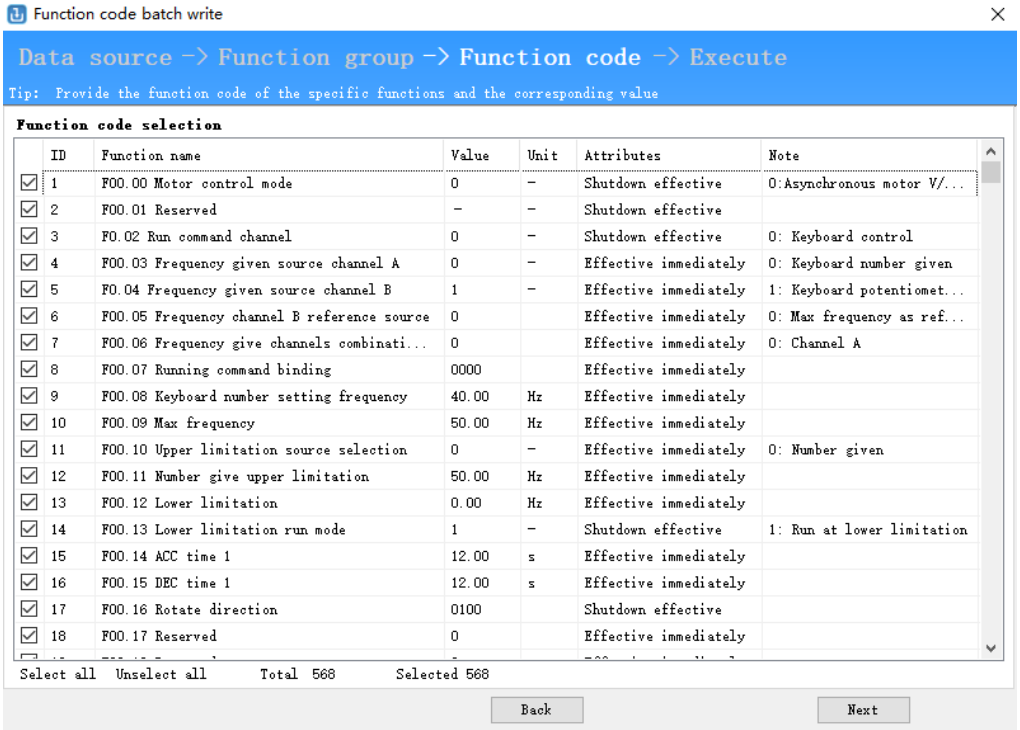


Figure 11

➢ Enter action interface, click [Start writing]. If any abnormal after finishing, fault sheet will be listed. Click [Done] to quit. As shown in figure 12 as below:

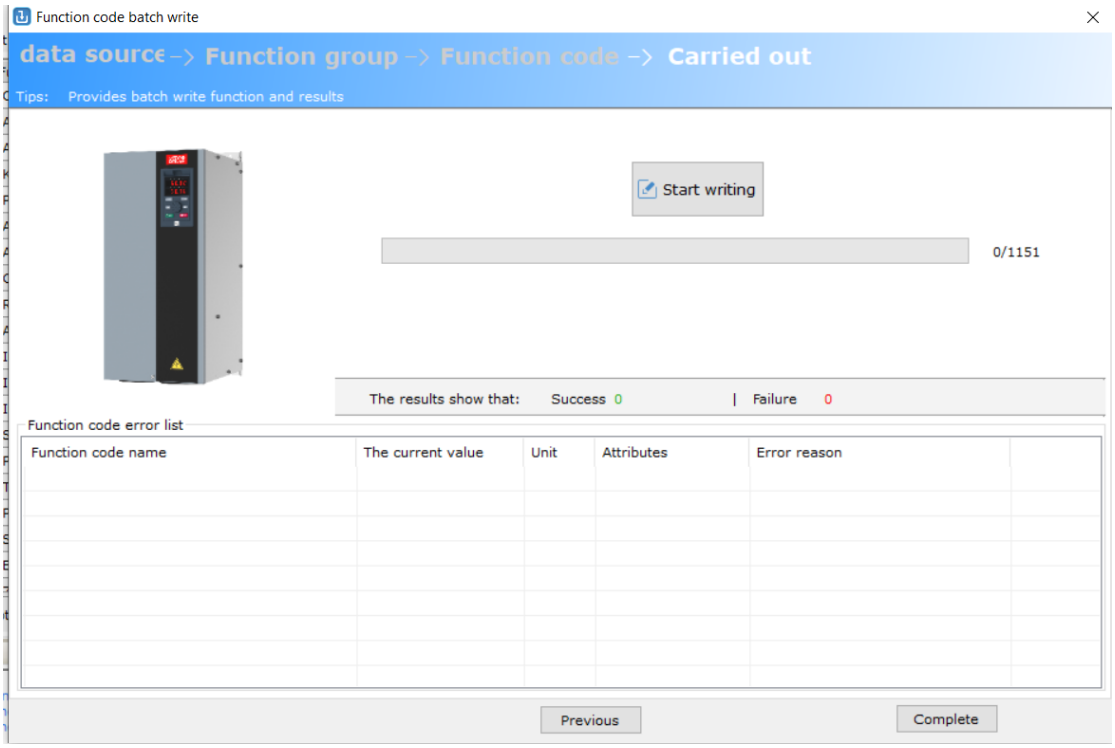


Figure 12

3.3.3. Function code leading-out

This function can lead-out all current codes;

Steps:

1. Click toolbar icon Export, pop-up the dialog box, click [OK] to complete.

3.3.4. Code saving

It can save the whole codes to the current configuration files;

Steps:

1. Click toolbar icon Save.

3.3.5. Function code comparison

Code comparison is done through two different historical data files, or a historical

data file is compared with the internal parameters of the online driver to find different functional codes.

Steps:

1. Click toolbar icon Compare, pop-up the dialog box, as shown in figure 13 as below:

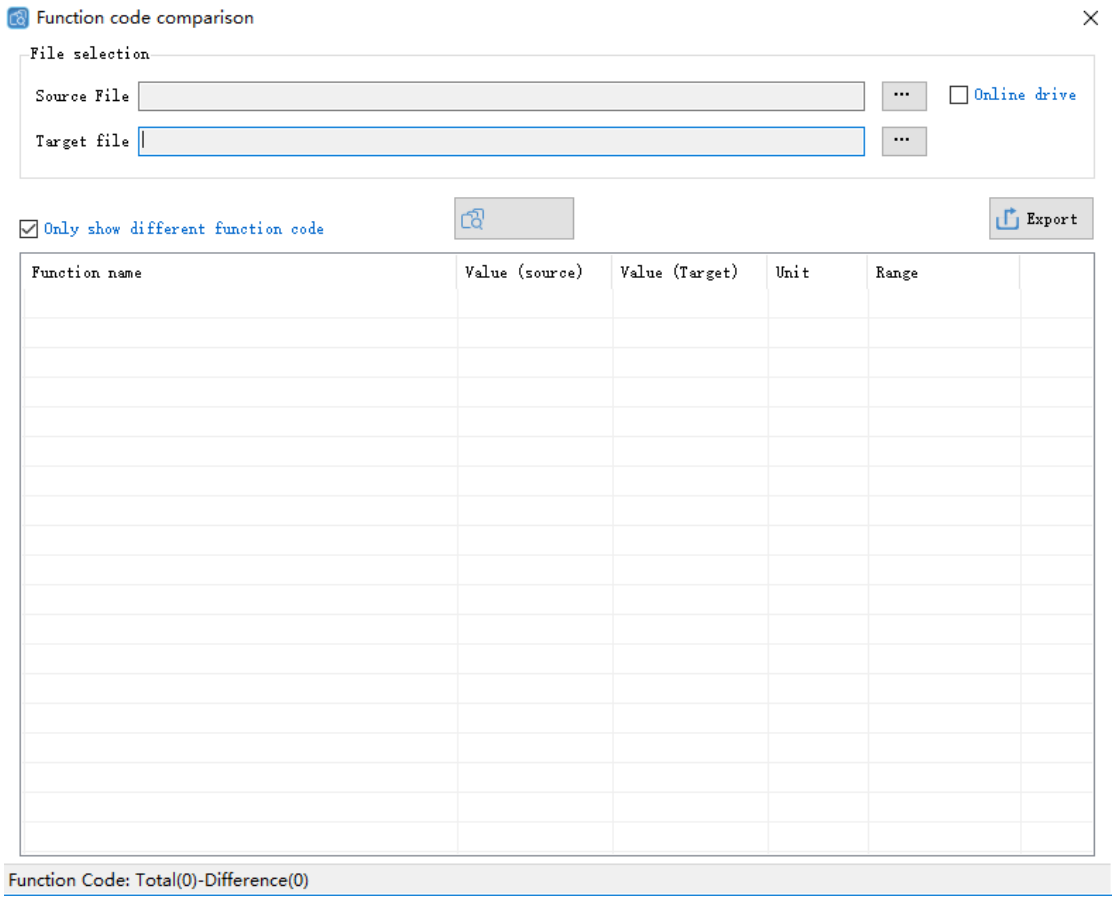


Figure 13

2. Click comparison button after selecting content.

3. When selecting "online driver", you will be prompted to read all the function codes to ensure the punctuality and correctness of the comparison.

4. The list displays the different function codes by default, and can display all the function codes.

3.3.6. Function code Look-up

Function code look-up provides a vague lookup function, and queries the related function codes to facilitate the user to operate on a specific function.

Steps:

1. Click toolbar Find, pop-up dialog box, as shown in figure 14 as below:

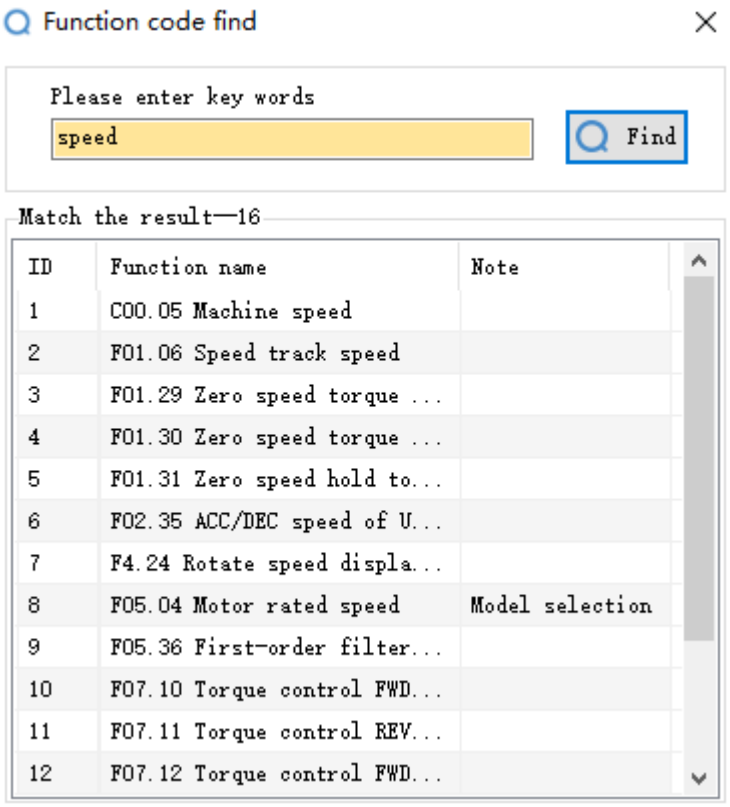


Figure 14

2. Enter the key words, such as "speed", enter or click the "look-up" button, list all relevant speed key function codes, support function code name and note look-up.

3.4. Real Time Monitoring

Real-time monitoring provides real-time view of monitoring parameters and I/0 state, as well as current error information.

Steps:

1. Start the real-time monitoring interface (two modes):

Method 1: Click toolbar [Tools] -> [Real-time monitoring].

Method 2: Click toolbar icon 

.

2. Display the real-time monitoring interface, as shown in Figure 15 below:

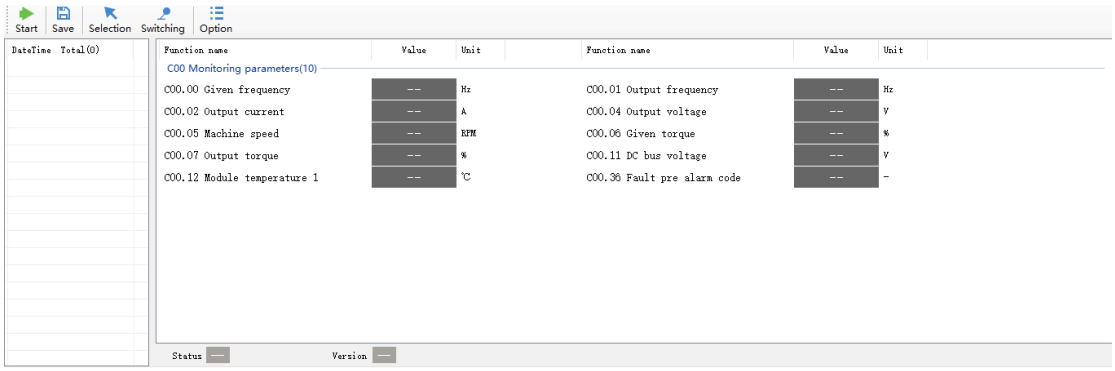
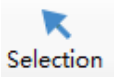


Figure 15

3. Start and stop real-time monitoring: click the toolbar icon 

, the right list in the interface is recorded time, only the latest 500 records can be set through options.

4. Parameter selection: click the toolbar icon 

, the pop-up selection interface, as shown in Figure 16 below:

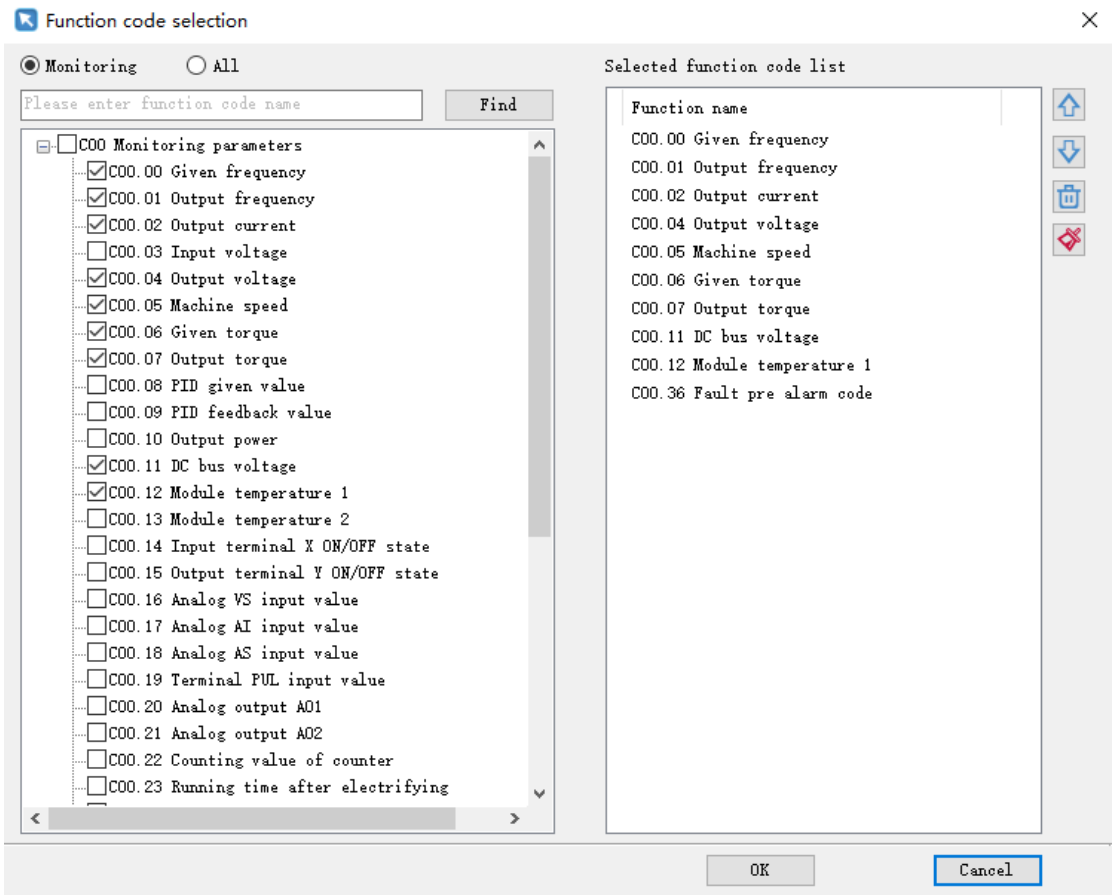


Figure 16

5. Save real time data: click the toolbar icon 

to save real time data in Excel

format.

6. Running state: when the running state is stopped, it displays red font; when the running state fails, it displays yellow font. You could click directly into the failure information query page to know the details.

7. State maintenance: clicking toolbar icon Switching, pop-up the status maintenance interface, providing add and delete function of state variables, which facilitates the custom variables of customers, as shown in the following figure 17:

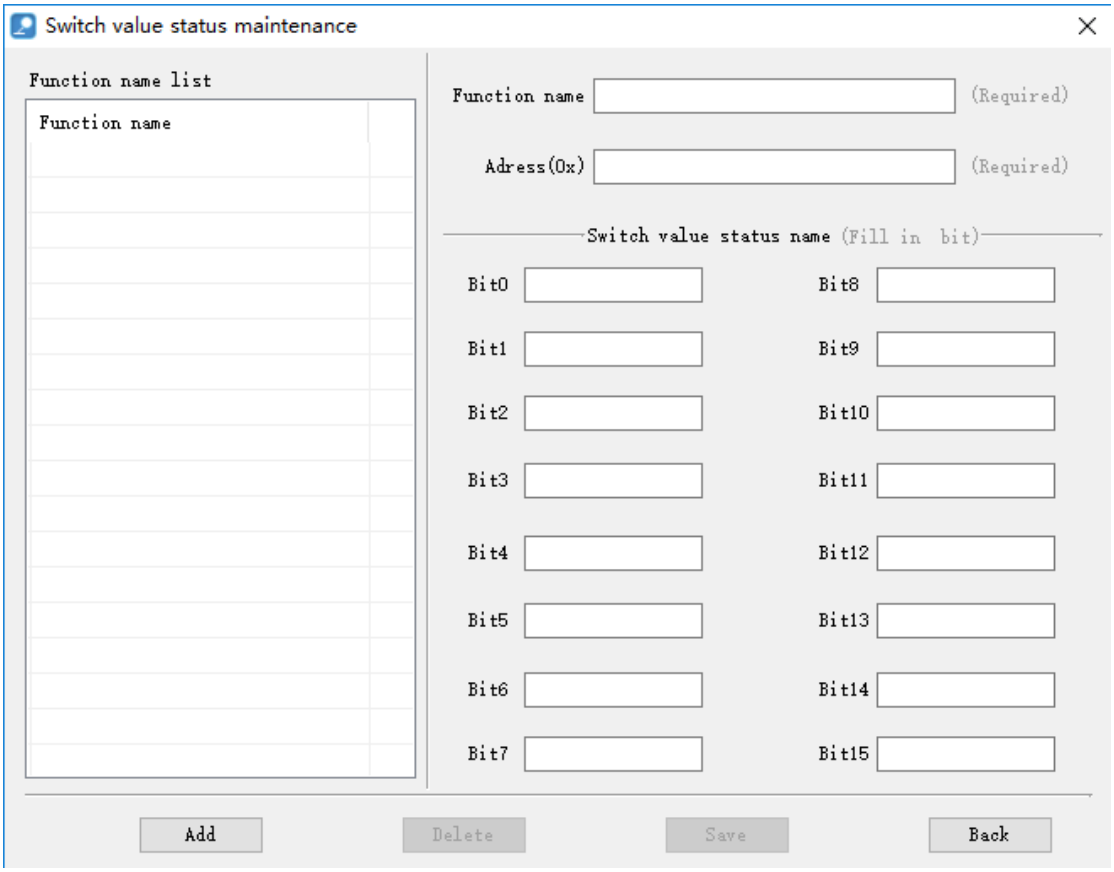


Figure 17

3.5. Error Information

Error maintenance can read all the current errors and provide corresponding troubleshooting measures for reference. At the same time, it also provides error reset, error clearing, error preservation and other functions.

Steps:

1. Start the error maintenance interface (Three modes):

Method 1: Clicking toolbar [Tools] -> [Error information];

Method 2: Clicking toolbar icon Fault info;

Mode 3: When the real-time monitoring interface has an error display, click [Error]

Display the error maintenance interface, as shown in Figure 18 below:

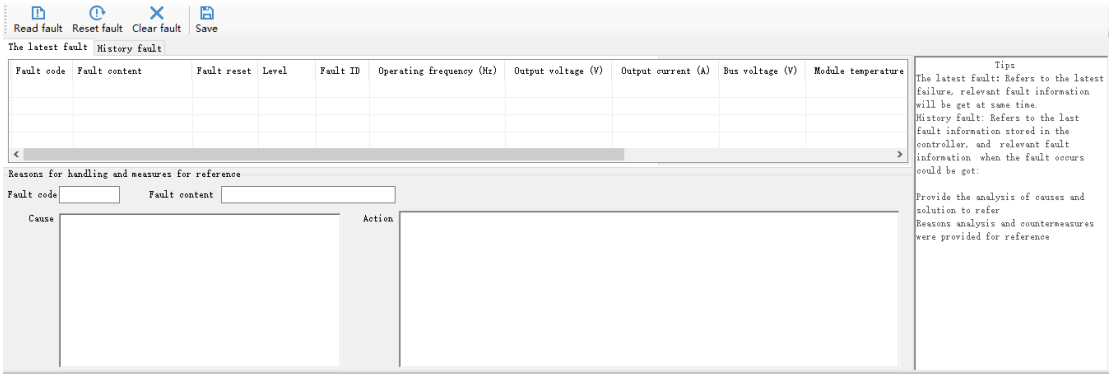
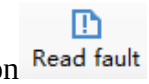
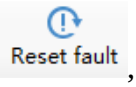


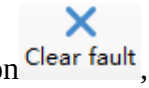
Figure 18

3. Read error: click the toolbar icon 

to read the current error related information (including a history error).

4. Error reset: click the toolbar icon 

, immediately send down the error reset instruction.

5. Error removal: click the toolbar icon 

, immediately send out the error clearance instruction.

6. Error preservation: click the toolbar icon 

to save the error information in Excel format.

7. Error treatment: clicking on one certain error will automatically show the cause of the current error and the corresponding treatment.

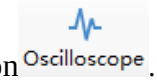
3.6. Digital Oscilloscope

The digital oscilloscope is used to collect data at high speed and display it in a graphic curve, which is used to analyze the data.

Steps:

1. Start the data oscilloscope interface (two ways):

Method 1: Click menu bar [Tools] -> [Oscilloscope].

Method 2: Click toolbar icon 

.

2. Display the data oscilloscope interface, as shown in Figure 19 below:

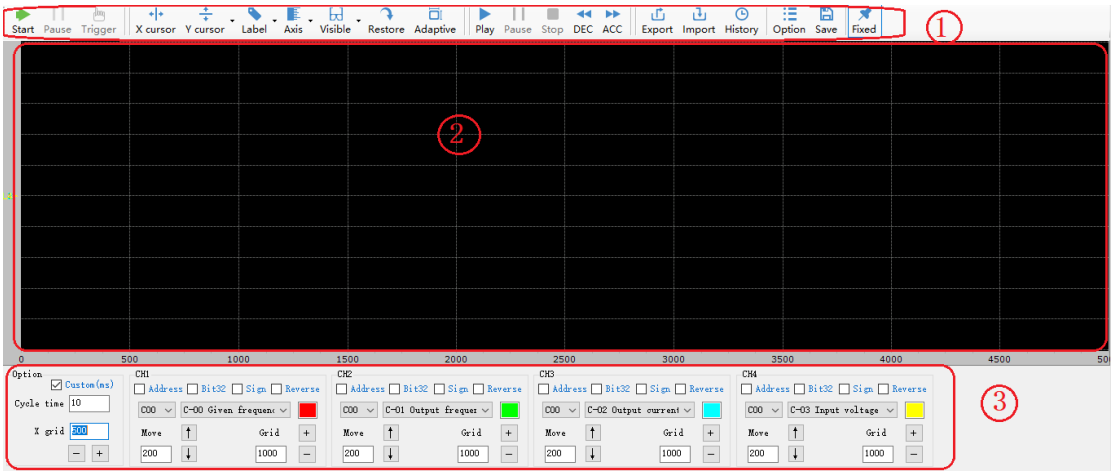


Figure 19

①—Toolbar

Toolbar includes start-up, manual trigger, X axis cursor, Y axis cursor, XY label value, Y axis scale, Y axis curve, one button restore, replay start, replay pause, replay stop, replay deceleration, replay acceleration, data export, data import, history, configuration options, save configuration and other functions.

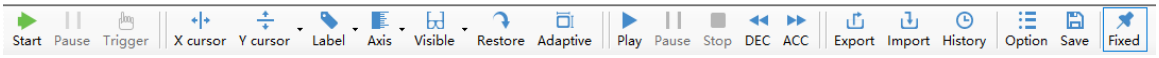


Figure 20

②—Curve display area

Different curves provide visual display.

③—Channel parameter setting

It provides channel related parameter setting functions. Parameter settings include option parameters and channel parameters.

3.6.1. Start/Stop

Steps:

1. Start: Click toolbar icon 

, start recording the wave, while the icon status changes to 

, the pause and manually triggered icon change available status.

2. Stop: Click toolbar icon 

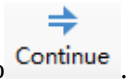
, stop recording the wave, while the icon status change to 

, the pause and manually triggered icon change unavailable status.

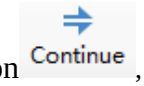
3.6.2. Pause/Continue

Steps:

1. Pause: Click toolbar icon 

, Wave recording is paused, while the icon status changes to 

.

2. Continue: Click toolbar icon 

, Wave recording will continue, while the icon status changes to 

.

3.6.3. Manual trigger

Steps:

1. When the start button status changes to 

, which means recording is starting. Click the toolbar icon 

, the system immediately sends a manual trigger command.

2. After the trigger waveform is received, the waveform will remain in the last state.

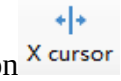
3. If you want to trigger it again, you need to restart recording.

3.6.4. Graphics operations

Graphical operations include X-axis cursor, Y-axis cursor, XY label value, Y-axis scale display / hide, Y axis curve display / hide, curve zoom in / out, curve shift, one key restore, graphic property setting and other functions.

3.6.4.1 X axis cursor

Steps:

1. Click toolbar icon 

. The graph automatically displays the X1 and X2 axes. The X1 and X2 values corresponding to the X-axis cursor are automatically displayed on the right. The points between the two axes, the theoretical time difference, and the actual time difference are as follows:

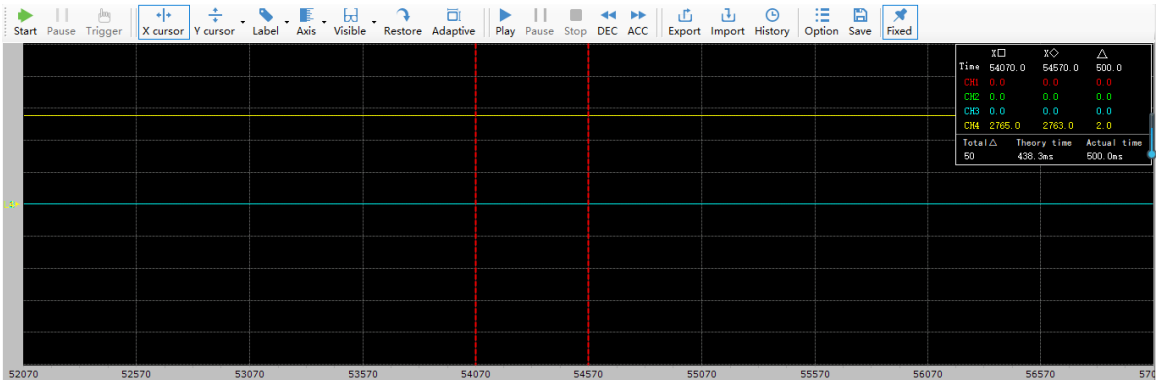


Figure 21

Remarks:

- X □ and X ◇ are the time corresponding to the two cursors on the X axis and the values corresponding to the time of channels 1 to 4; △ is the difference between the two cursors.
- Theoretical time difference: X □ and X ◇ between the points multiplied by the sampling interval; that is the △ value; (in the measurement to the theoretical time difference).
- The actual time difference: X □ and X ◇ is the host computer to receive data recording time, which is provided by the computer system time, measurement only as a reference.
- In the case of stable communication, the theoretical time difference and the actual time difference are basically the same, if there is packet loss, the theoretical time difference is less than the actual time difference.
- Right-click on the measurement results panel, pop up the information menu, you can show / hide the relevant name of the current channel and the drive version number.

	X□	X◇	△
Time	19190.0	19220.0	30.0
CH1	0.0	0.0	0.0
CH2	0.0	0.0	0.0
CH3	0.0	0.0	0.0
CH4	2770.0	2770.0	0.0
Total△	Theory time	Actual time	
3	26.5ms	30.0ms	

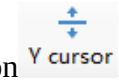
- Fine-tuning the X-axis cursor: Click a certain cursor axis, then press the keyboard left and right keys to achieve fine-tuning function.

3.6.4.2 Y-axis cursor

There are several axes on the Y axis, which can be selected according to the channel. Only one channel can be selected at a time. That is, CH1 represents the axis of curve 1; CH2 represents the axis of curve 2, and it is analogous in turn.

The color of the cursor changes with the color of the curve.

Steps:

1. Click toolbar icon 

. If you select [CH1], the graph automatically displays the corresponding two axes of Y1 and Y2. The color is the same as that of CH1; as shown in figure 22 below:

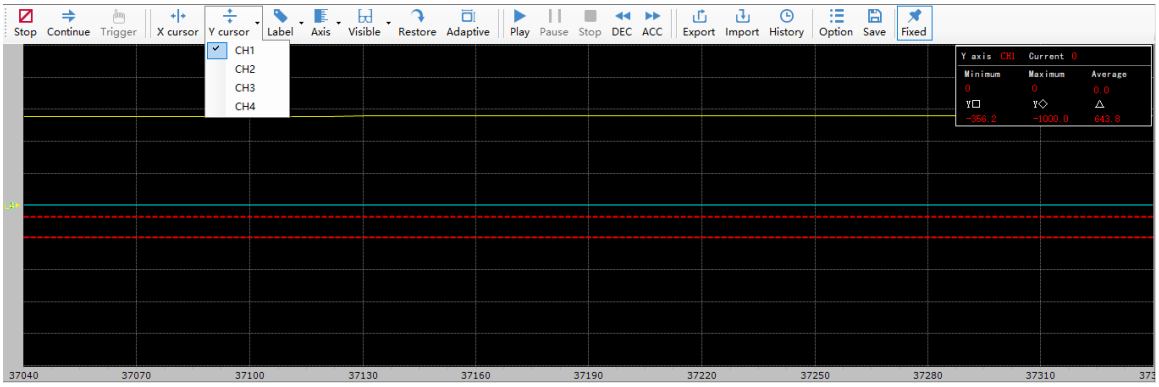
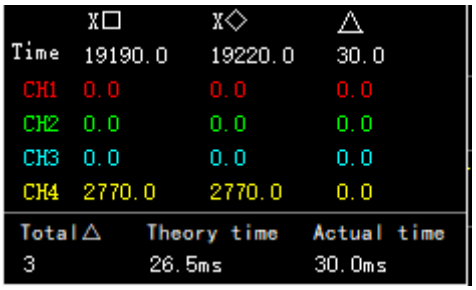


Figure 22

Remarks:

- Display the max-value, min-value, average-value, current-value (These statistics are calculated for the range of viewable areas of the current graph) and the corresponding values of the Y □ and Y ◇, and △ means the difference between these two values.
- Click the right-key on the measurement results panel, pop up version number, such as the information menu, you can show / hide the relevant name of the current channel and the drive:



3.6.4.3 XY tag value

There are several axes on the Y axis, which can be selected according to the channel. Only one channel can be selected at a time. That is, CH1 represents the axis of curve 1, CH2 represents the axis of curve 2, and it is analogous in turn.

Dot represents a point on one certain time; the mouse will be displayed in the form of a label to display the current point of the XY value;

Steps:

Two different display methods:

Method 1: Click toolbar icon 

, if you select [CH1], the graph automatically displays the dot corresponding to curve 1, as shown in Figure 23 below:

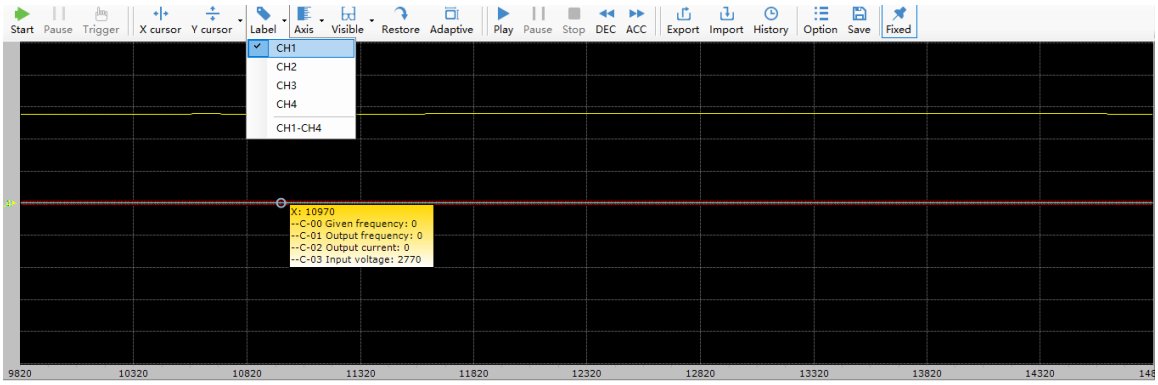


Figure 23

Method 2: Click toolbar icon 

, if you select [CH1-CH4], all the curves will follow the mouse, as shown in Figure 24 below:

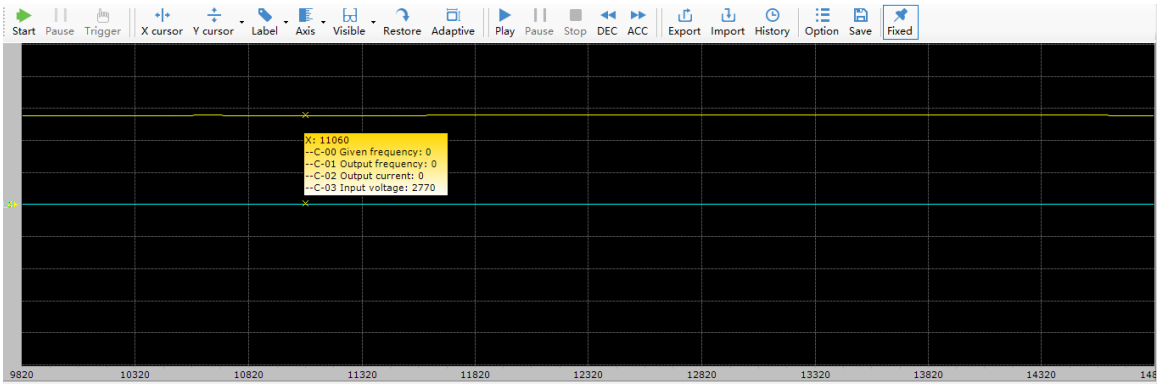


Figure 24

3.6.4.4 Y-axis Scale Display / Hide

There are several coordinate scales on the Y axis, which can be selected according to the channel. Only one channel can be selected at a time. That is, CH1 represents the scale of curve 1; CH2 represents the scale of curve 2, and it is analogous in turn.

Y-axis scale is displayed in the left area:

Steps:

1. Click on the toolbar icon 

, such as CH1, the Y axis corresponding to the

curve 1 scale is displayed on the left, as shown in Figure 25:

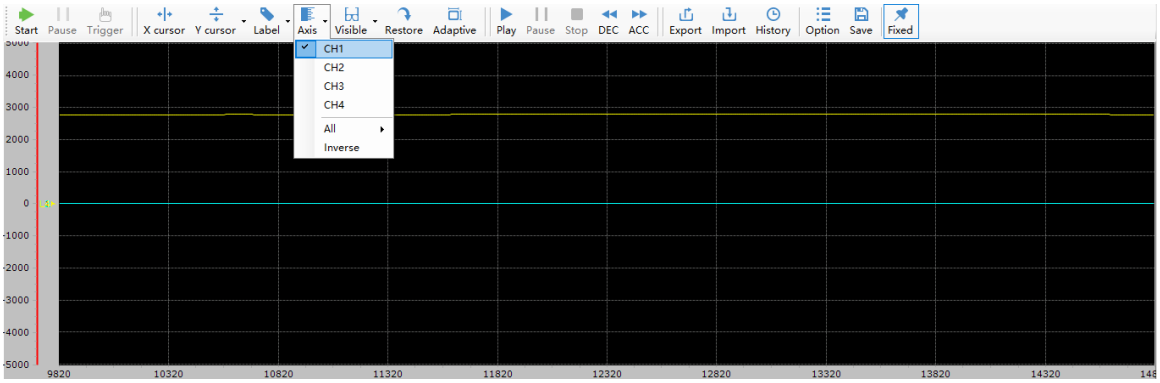


Figure 25

3.6.4.5 Y Axis Curve Display / Hide

The Y axis has multiple curves, which can be selected according to the channel. One can only choose one channel at a time. CH1 represents the curve 1, and the CH2 represents the curve 2, and it is analogous in turn.

Y axis curve is shown by default;

Steps:

1. Clicking on the toolbar icon 

, such as removing CH2, Y axis curve corresponding to the curve 2 will be automatically hidden in the graphics; as shown in Figure 26:

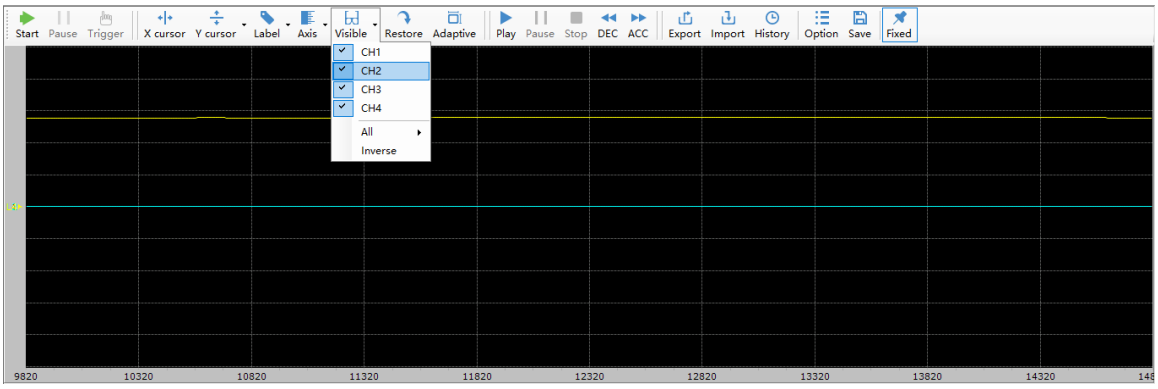
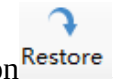


Figure 26

3.6.4.6 One-Key Restore

With the function of the restoring the curve, when the curve is magnified or moved, it can be restored to the original curve shape.

Steps:

1. Click the toolbar icon 

.

3.6.4.7 Curve enlargement / reduction

Steps:

1. Area enlargement: press the left button of the mouse and pull a region from the upper left corner to the lower right corner, that is, the area can be amplified.

2. Area reduction: press the left button of mouse and drag from the lower right corner to the upper right corner, which is to reduction function.

3. Curve X/Y enlargement: click button 

.

4. Curve X/Y reduction: click the button 

.

3.6.4.8 Curve Translation

Steps:

1. Horizontal translation of curve：Press the right button of mouse and move right and left (Note: when you move to the right to 0 point, it cannot be moved right).

2. Single curve vertical translation: press button 
, you should long press and do not release, if continuous translation is needed.

3. All curves are vertical translation at the same time: Click [configuration options] - > [Graphic options] -> Select [Up and down translation by pressing the right button of the mouse] - > [OK].

4. Click logo such as [L1]. It can be dragged up and down to the corresponding curve translation.

3.6.4.9 Curve adaption

Steps:

Curve adaption: Click the toolbar icon 

to realize one key to adjust the size of the curve and display it in the visual area to achieve the purpose of rapid adjustment.

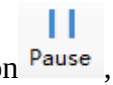
3.6.5. Curve replay

The curve replay provides a function that could replay the historical curves:

including starting, pause, stopping, acceleration and deceleration.
Steps:

1. Start replay: click the toolbar icon 

and the pause and stop icons turn into available states, start the replay icon turns into unavailable state, then operate the start replaying function.

2. Pause replaying: Click the toolbar icon 

, pause turns into unavailable state, start replaying turns into available state, and operate pause replay function.

3. Stop replaying: Click the toolbar icon 

, pause and stop the icons turn into be unavailable, start the replay icon turns into the available state, and operate the stop replaying function.

4. Decelerate replaying: Click the toolbar icon 

to operate the deceleration, and the speed of the drawing will get slow.

5. Accelerate replaying: Click the toolbar icon 

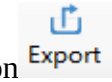
to operate acceleration, and the speed of drawing will be quickened.

3.6.6. Data import/export

According to the current graphics, data and pictures can be exported at the same time. The data supports TXT, XML, EXCEL and HTML exportation in various formats. Besides, the exported data can be imported to view.
Steps:

1. Data import: Click toolbar icon 

, pop up the open dialog box, and find the existing data file.

2. Export data: Click the toolbar icon 

, save dialog box to the specified path. When the file does not exist, the system will automatically create a folder, which consists of 5 different extension files of the same name. Do not change the name of the folder and the file!

3.6.7. History

Record the historical curve of the current importation, it can increase, delete operation, can quickly find the current history curves.

3.6.8. Configuration option

3.6.8.1 Rigger conditions

Steps:

1. Click the icon Option -> [Trigger condition], switch to the trigger condition page, as shown in Figure 27.

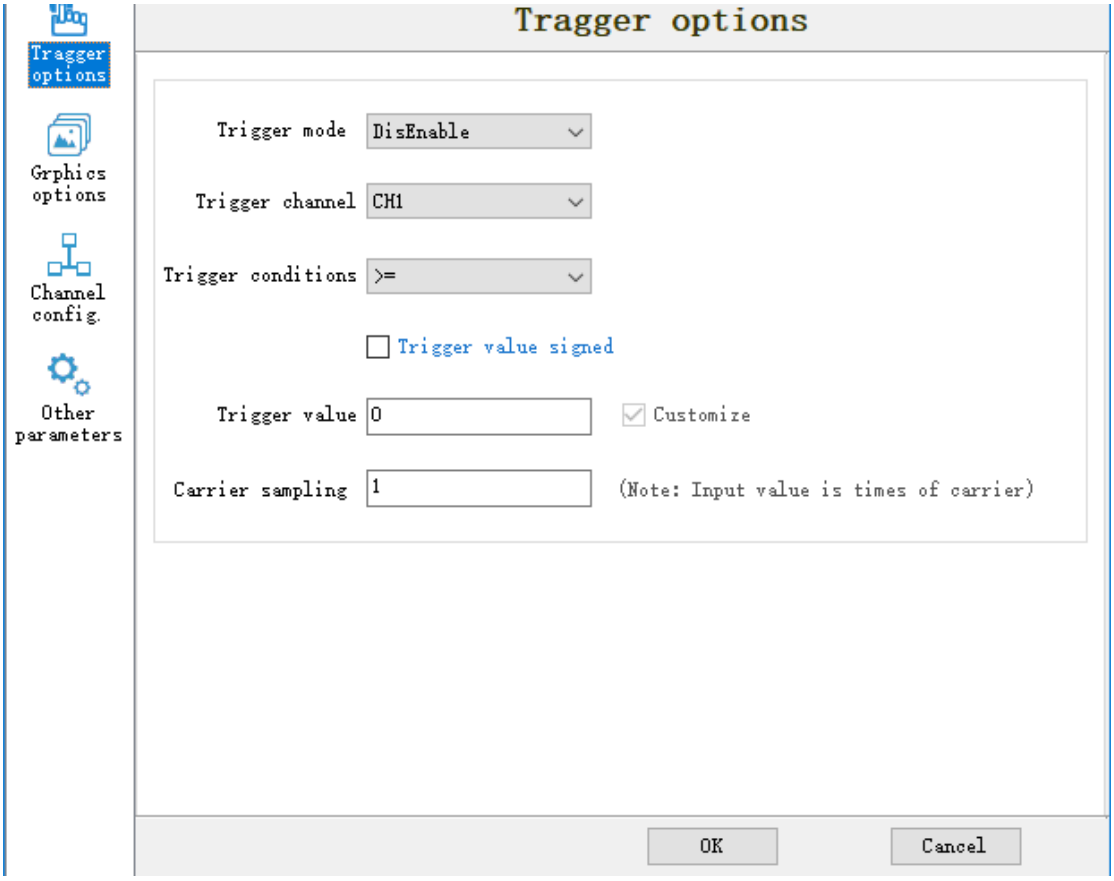


Figure 27

Note: After setting the trigger condition parameter, the terminal will receive the trigger condition, and it will judge automatically according to the condition.

3.6.8.2 Graphic setting

Steps:

1. Click the icon Option -> [Graphic option], switch to the graphic options page, as shown in Figure 28:

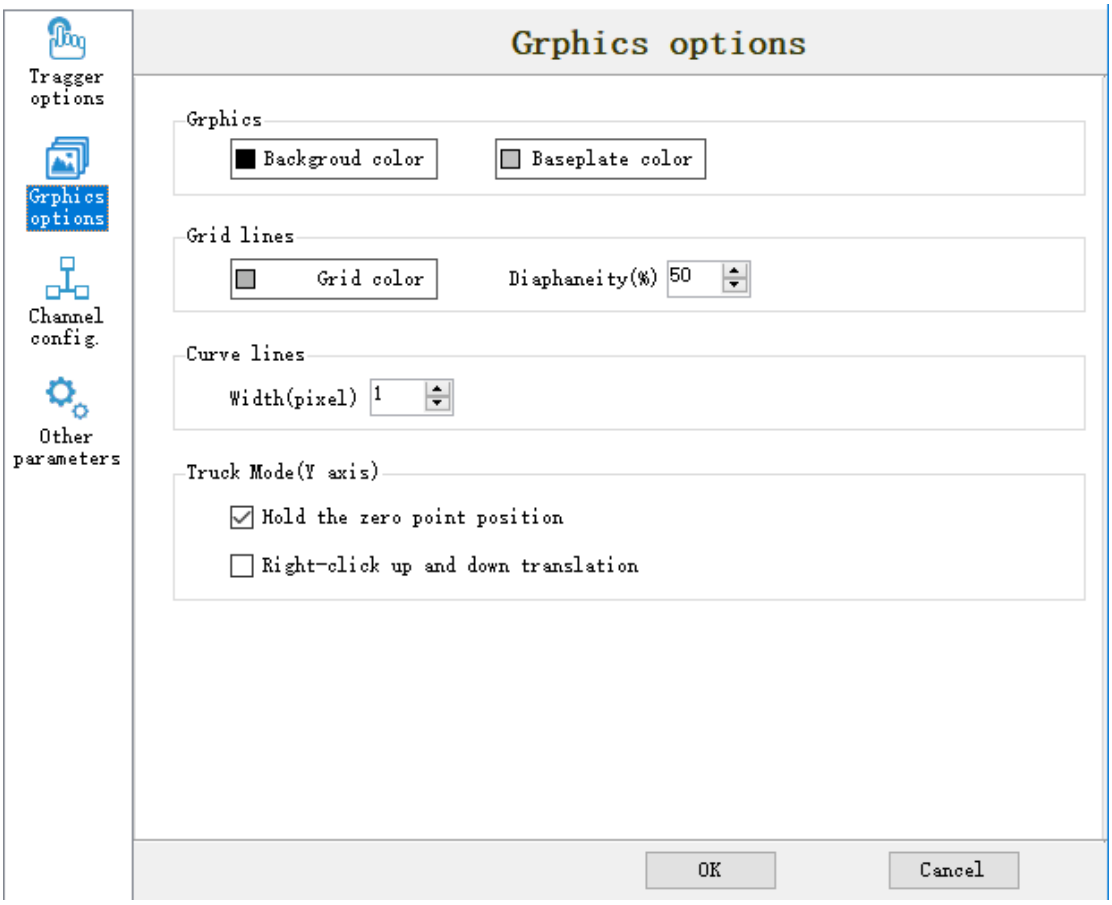


Figure 28

2. Attribute specification:

Background color: The background color of the graphics can be set.

[Floor color]: The background color of the panel can be set.

[Line width]: The line width of the curve can be set in pixels.

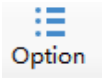
[Lattice line]: Color and transparency can be set.

[Keep 0:00 Position]: When it's in the checked state and the Y-axis curve is panned up and down by pressing the buttons, the 0 position will remain in the visible area of the graph. When the top or bottom end of the pan is shifted, it will not be able to translate; otherwise, 0 position will be unlimited up and down translation; the default check.

【Right-click and up-down】: When the status is checked, the right mouse button can pan up and down, all the curves will be panned at the same time; otherwise, it will not be panned; it's unchecked on the default setting.

3.6.8.3 Channel Configuration

Steps:

1. Click the icon 

-> [Channel Configuration], switch to the channel configuration page, as shown in Figure 29 below:

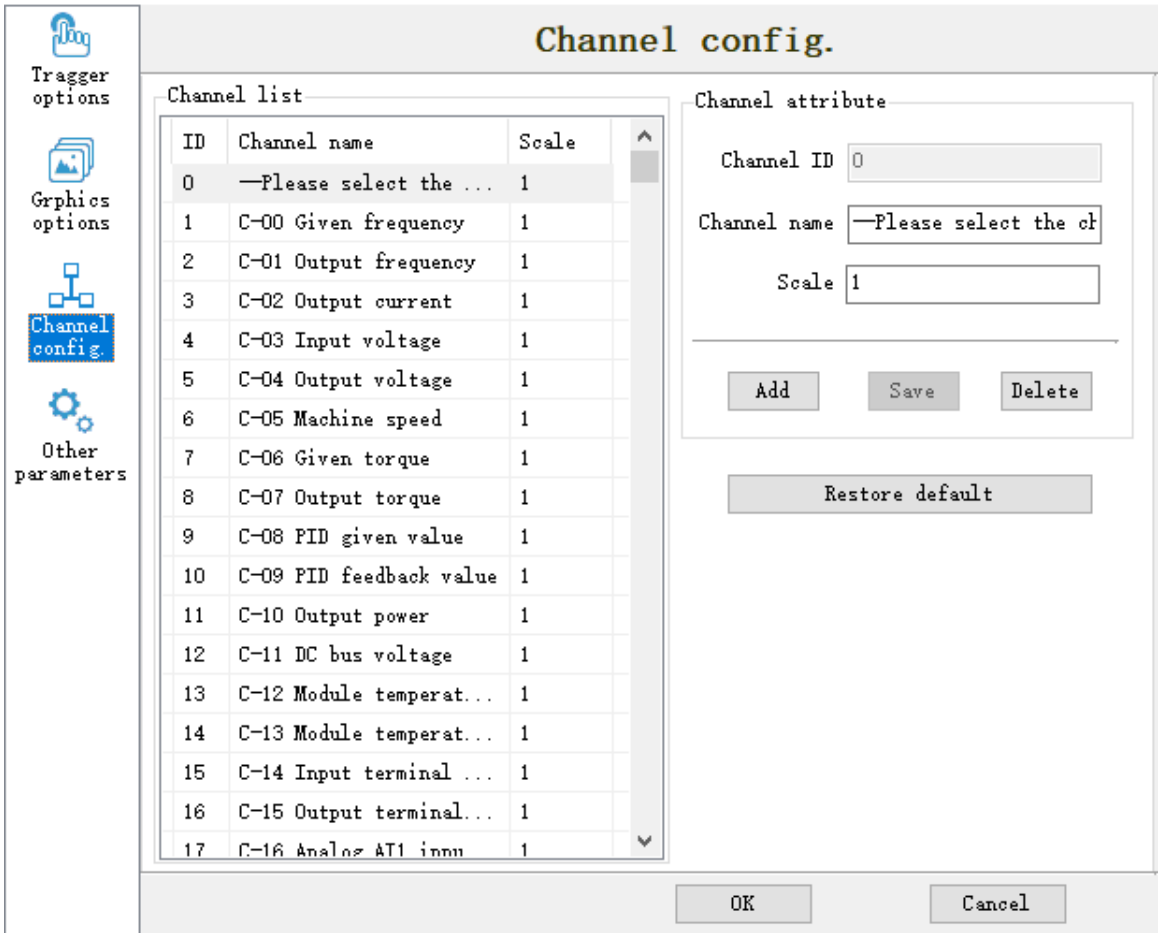


Figure 29

2. Channel change: Select a channel on the left, the right shows the channel property, you can click 【Add】, 【Save】, 【Delete】 button to operate.

3. Restore default value: Click [Restore default value], all channels will default value display.

3.6.9. Channel Parameters Configuration

The channel parameters include the common parameters and the parameters of each channel, supporting simultaneous sampling of up to four channels; as shown in Figure 30:

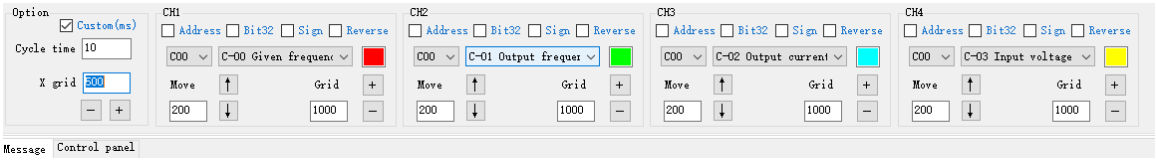


Figure 30

1. General Parameters Specification:

- 【Sampling Interval】: When RS485 communication, the unit is ms, the default is 1, 2, 4, 8, 16, 32, 64. You can also select 【User-define】. If it is CAN, display sample interval changed to carrier multiple.
- 【X-axis grid】: The range is 【1 ~ 10000】, clicking [+] will increase the X-axis grid size, while the compression curve, clicking [−] will reduce the X-axis grid size, while drawing the curve. Channel Parameters Specification: Four channels are provided, that is, four channels correspond to four curves.
- 【Address】: Mainly for providing fixed address to view the data, the address supports decimal and hexadecimal input, enter the address starting with 0x hexadecimal number, such as (0x8825), the longest input can be 4 bytes. In addition, if you do not enter any value after selecting an address, the default channel is not selected.
- 【32 bit】: If selected, this value is 32-bit data.
- 【Signed】: Select this function, the data will be in the range of 【-32768 ~ 32767】, otherwise, the range of 0 ~ 65535 will be unsigned number.
- [Inverted]: Check the inverted option. Drawing curve by data inversion.
- [Channel]: provide the selection of the number of dropdown channels, select a channel, only select the address, and then display the address.
- 【Color】: Provide the color choice of curve.
- 【Grid】: Y-axis coordinates of each grid size can be increased or decreased accordingly and the range is【1 ~ 100000】. Clicking [+] will increase the Y-axis grid size while compressing the curve, click [−] will reduce the Y-axis grid size while drawing the curve.
- 【Pan】: The size of the pan will change with the size of 【Grid】, [↑] means move upwards, and [↓] means downwards. If you press the button [↑] or [↓] a little bit long, it could move continuously.

4. Others

4.1. Control Panel

It provides the operation of the quick send command function, as shown in Figure 31 below:

Figure 31

4.2. Messages

It provides relevant information for inquiring about the operation of the user, the blue font is a normal message; the red font is an abnormal message.

4.3. Window Display

Window display is divided into: cascade, horizontal, vertical display:
1. Cascade: click the menu bar [Window] -> [Cascade display].
2. Level: click the menu bar [Window] -> [Horizontal display].
3. Vertical: click the menu bar [Window] -> [Vertical display].

4.4. Help

4.4.1. Instructions

It provides instructions for using the commissioning software.
Steps:
1. Click the menu bar [Help] -> [Instructions].

4.4.2. Update online

Through the online upgrade function, the current version can be remotely upgraded to the latest version;

Steps:

1. Start online update interface (Two methods):

Method 1: Click the menu bar [Help] -> [Online Upgrade].

Method 2: Click the prompt on the far right of the status bar (You have the new version).

4.4.3. About Us

We provide the software version, copyright and our contact information.

Steps:

You could click menu bar [Help] -> [About Us].

Attachment 1

This attachment contains the development mode of the various functions of the specific instructions.

Features include: Command window.

1.1. Command Window

The main role of the command window is to facilitate developers in the debugging process; you can send and receive commands through a format which is a simple serial debugging tool.

Steps:

1. Message bar Click 【Command window】 to display the current command window.

Remark: When check 【Reply check】 option, the received return data will be automatically checked according to the current check mode and displayed at the end of the data.