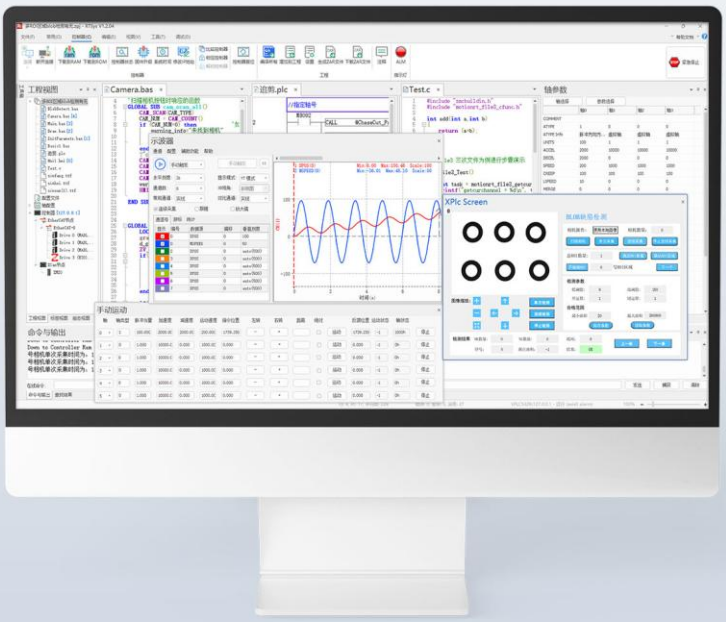
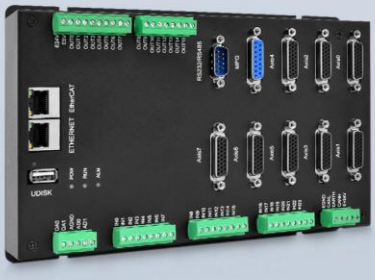
Zmotion®
正运动技术

做更好用的运动控制，智造美好生活
Better Motion Control, Smarter Life

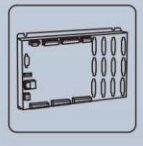
RTSys User Manual

V1.5.2

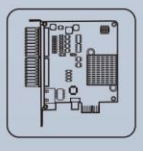




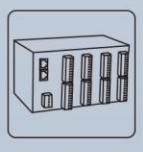
Vision Motion Controller



Motion Controller



Motion Control Card



IO Expansion Module

HMI

Copyright statement

ZMC controller software involved in details as well as the introduction and routines of each instruction, please refer to ZBASIC software manual.

Information contained in this manual is only for reference. Due to improvements in design and functions and other aspects, Zmotion Technology reserves the final interpretation! Subject to change without notice!

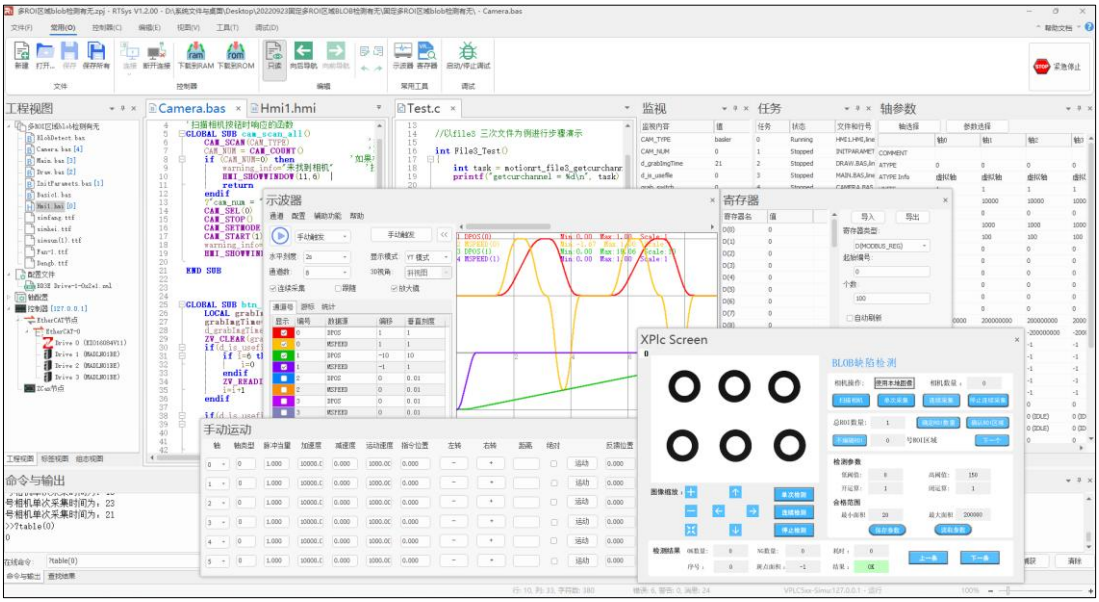
Pay attention to safety when debug the machine! Be sure to design effective safety devices in the machine and add error handling procedures in software. Zmotion has no obligation or responsibility for the loss.

Contents

Chapter I RTSys Overview

RTSys is one development software that integrates motion control and machine vision functions developed by Zmotion. It supports ZBaisc (RTBasic), ZPLC (RTPlc), ZHMI (RTHmi) and ZVision (RTVision) machine vision secondary development. Also, there are many functions, it supports hybrid programming, real-time simulation, online tracking, diagnosis and debugging, etc. What's more, for vision function, vision positioning, vision measurement, vision identification and vision detection of intellectual equipment can be realized rapidly, and complex motion control systems can be developed. RTSys software development page refers to below.

You can connect PC to controller through serial port or Ethernet, and program written by RTSys software can be downloaded into Zmotion motion controller directly to run offline or simulate on PC.



RTSys software supports 4 programming methods, Basic, PLC (ladder of diagram), HMI configuration and C language. In RTSys software, please note, for Basic language, it supports multiple Basic tasks operation, for PLC, it only can use one PLC task, for HMI, it only supports two HMI tasks. In addition, these languages' tasks can run at the same time.

RTSys supports Chinese and English interfaces by default, please check "12.5 Switch Languages" for details. You also can add other languages.

RTSys software supports online simulation debugging, and it has simulator "ZMC Simulator" and configuration program simulation tool "xplc screen".

- RTSys Doesn't Support Down to Compatible, that is, Please Don't Open RTSys File in ZDevelop. (ZDevelop is the original version)
- For HMI Function, Controller's Firmware Must Support HMI Function.

1.1. System Requirements

To operate RTSys, PC should meet below requirements:

Item	Minimum Requirements	Recommended
CPU	Pentium level processor, main frequency: 1GHz	Intel i5 level processor, main frequency: 2.9GHz
Graphics	/	OpenGL version can't be less than 1.5
Memory	1GB	4GB
Remain Hard Disk	1GB	4GB
Operation System	Windows7	Windows10
Display	1024x768/24-bit true color	1920x1080/8-bit RGB
Communication	RS232	RS232/USB/Ethernet (can be converted by HUB)

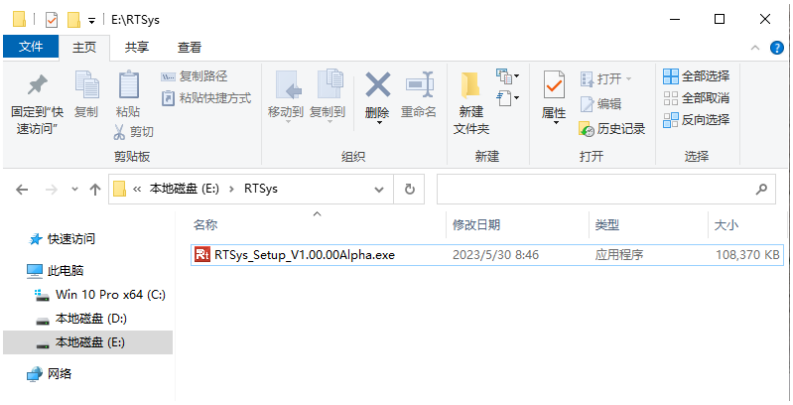
1.2. How to Install / Uninstall / Upgrade RTSys

1.2.1. How to Install RTSys

Installation Steps:

Step 1:

Get RTSys package from Zmotion website (zmotionglobal.com), then download it to PC, it can be saved in any place. Find it, then compress it, below .exe program will be obtained.



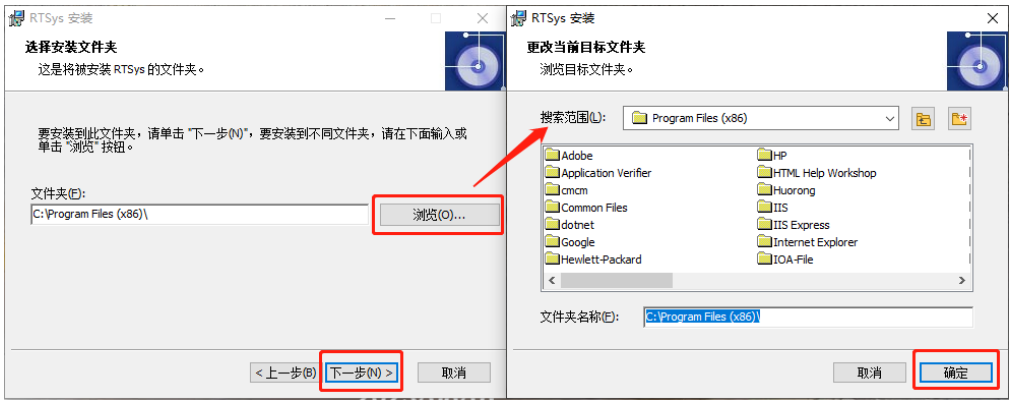
Note: the version No. in the image is not fixed, because it will update. The newest version can be got from website or contact with us.

Step 2:

Double click .exe, then below window will appear, click “next”.

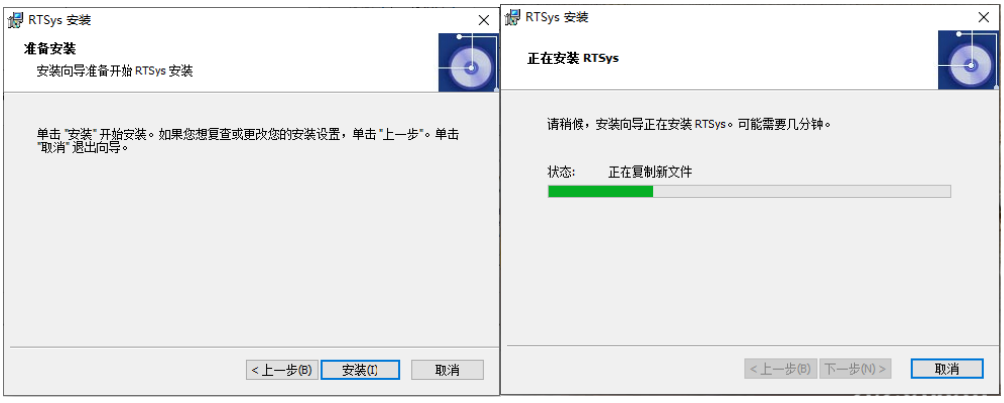
Step 3:

Click “browse”, then select software installation path. Then, click OK, and Next.



Step 4:

Click “Install”, right shows it is installing.



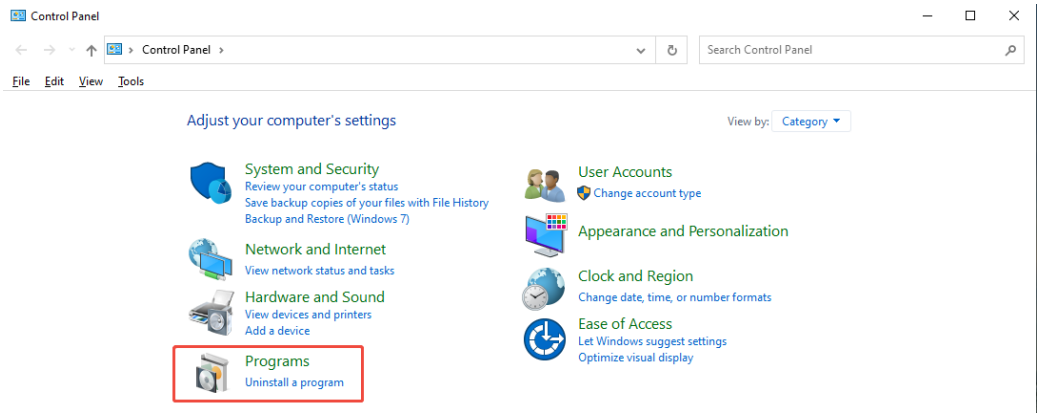
Step 5:

Below window will pop up if successfully, click “Finish”.

1.2.2. How to Uninstall RTSys

Step 1:

Open system “control panel”, then select “Program” – “Uninstall Program”.



Step 2:

Find “RTSys” software, right-click “uninstall”, then click OK.

Notes:

- While uninstalling, please close all corresponding RTSys interfaces.
- Not recommend to use the third-party software to do uninstallation! Please use system to uninstall!

1.2.3. How to Update RTSys

RTSys supports updating online directly. There are two methods to check new version.

➢ **According to "Update" Hint:**

When there is new version, when you open the RTSys, it will show "update" window, at this time, you can update or not.

--click "update now"--

If you click "no reminder", no this kind of hint any more when you open it next time.

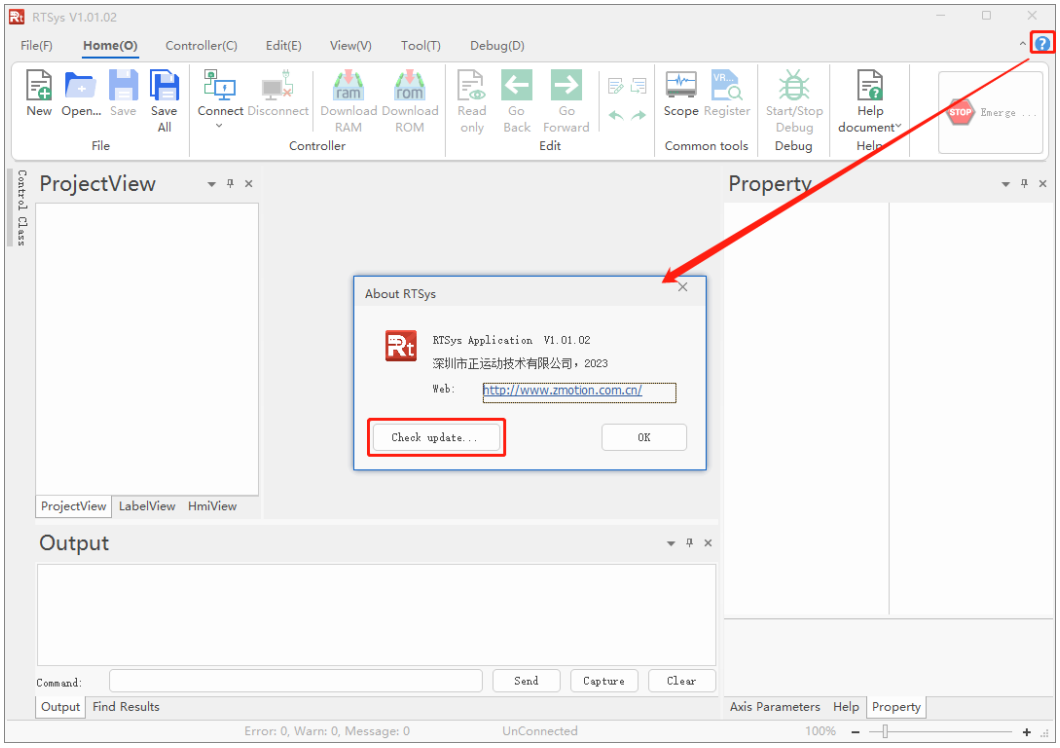
--click "next"--

Then, it starts to update. After that, do above "how to install" again.

➢ **Update Manually**

You can check the version manually, in website or in RTSys.

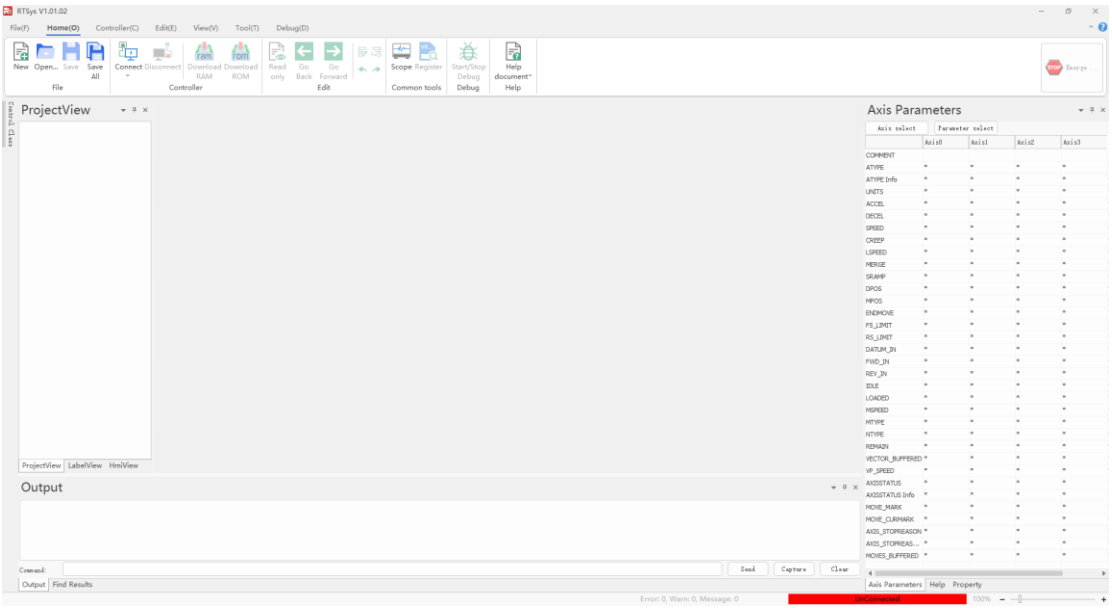
--click "?" in the right upper corner, and click "check update…"--



--if no need to update, click "OK", if there is newer version, please click "next", then execute

updating--

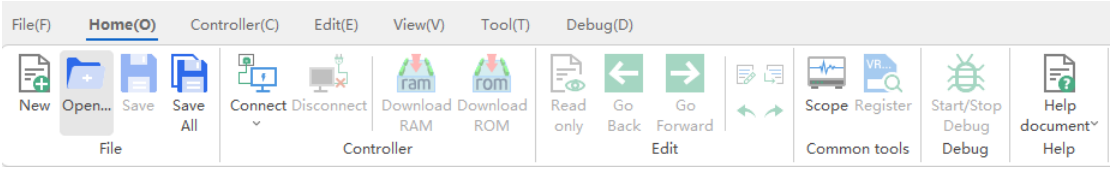
1.3. RTSys Main Interface



Above shows main interface.

Default information: menu bar, project view (label view, Hmi view), axis parameter (help, property), command and output window (result checking).

➢ **Menu Bar**



It mainly includes some basic functions, like, new build file, connect to controller, etc., at the same time, and there are some debugging tools. For details, please refer to behind content in this manual.

➢ **Main Interface Left Part**

It is mainly File / Label / Hmi window. It can be switched by clicking directly. For File, it shows as "ProjectView", you can check file numbers, type, and task No., editing can be achieved by double-click the file. More corresponding settings, please right-click mouse on the above of "ProjectView", it will pop up file setting window. For Label, it shows as "LabelView", all Basic SUB files can be checked. For Hmi, it shows as "HmiView", created windows and components' name in each window of HMI file can be known.

➢ **Main Interface Right Part**

It mainly includes Axis Parameter / Help / Properties window.

For Axis Parameter window, it shows corresponding parameters in motion control, for example, axis No., axis type (ATYPE), pulse amounts (UNITS), speed (SPEED), etc., and some parameters can be modified directly. For Help window, it can show needed commands help documents, double click the command in the file can check in this window quickly. For Properties window, it is used to show and set HMI function' window and components' properties and parameters.

➢ **Main Interface Bottom Part**

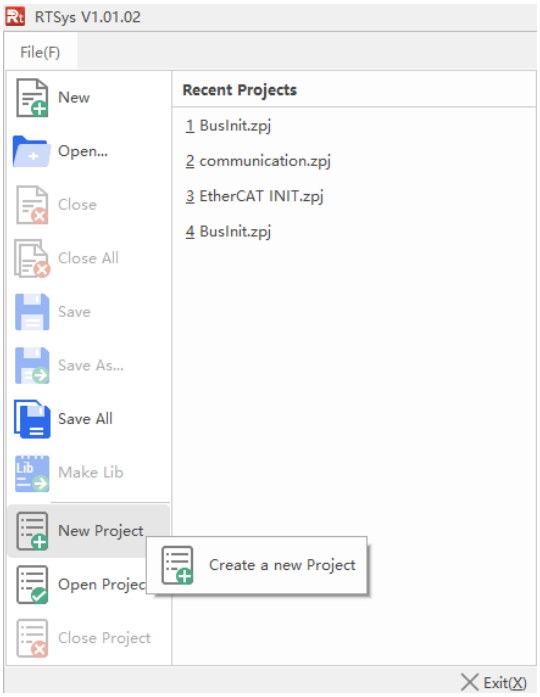
It is used to check and output all kinds of controller parameters, and it can online enter commands. In addition, output information will show file name, line No. and content.

1.4. RTSys Basic Operations

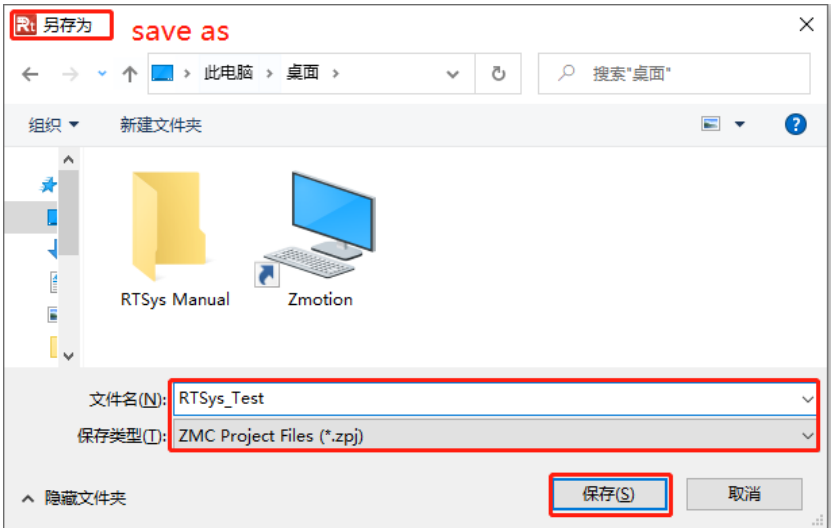
At first, build a new folder in PC to save the project that you will create. Here take RTSys V1.01.02 as example, if you need to update, please refer to "How to Update RTSys".

Basic Processes: build a new project → build a new file → select the file type → add AutoRun Task No. → edit the program → connect to controller → download the program into controller.

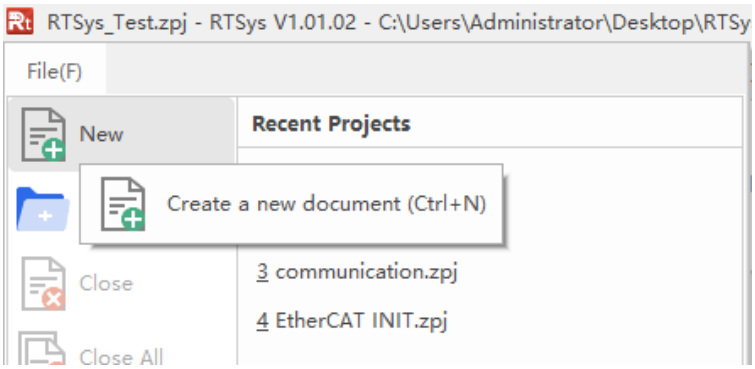
1. Build one New Project: "File" in "Menu" → "New Project".



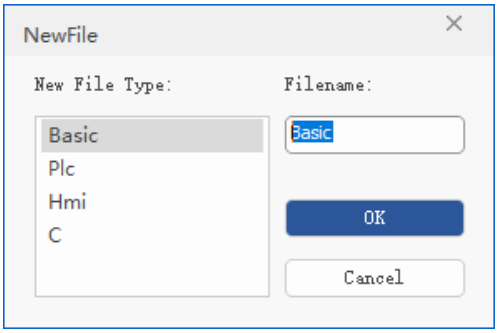
Click “New Project”, then “Save as…” will be jumped, select the folder (that was created just now), and open it, then input folder’s name and save the project, pay attention to the suffix should be “.zpj”.



2. Build New File: File – New

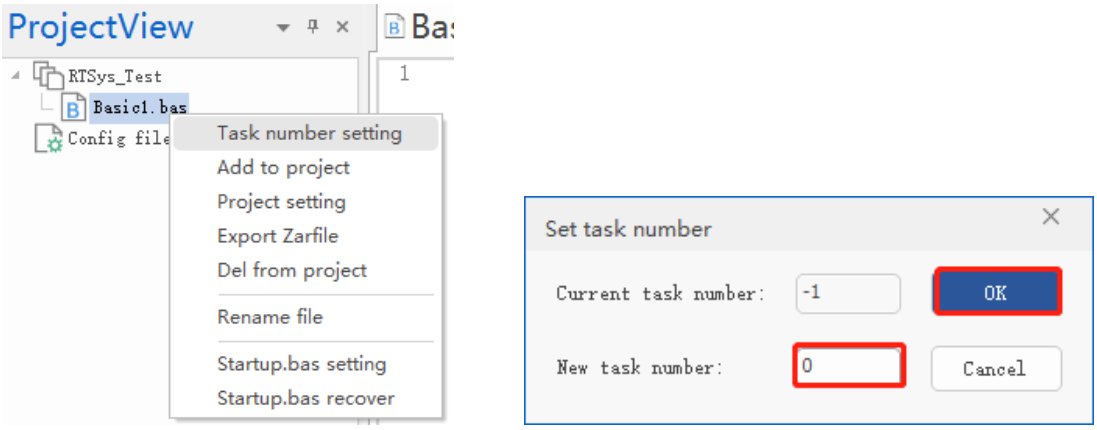


After clicking “New File”, below poping window will appear, which supports Basic/PLC/Hmi hybrid programming. Here selects the “Basic” file type and click “OK”. The name can be modified.



3. Set “File Automatically Run” (AutoRun Task No.)

Like below left image, right-click file, select “task number setting”, then see right image, enter the No. “0”, click OK.

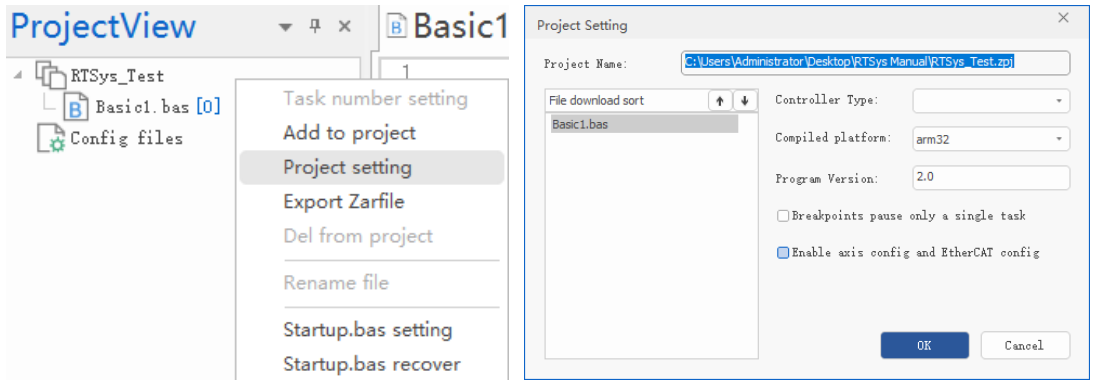


After setting, in this file name right side, task No. will be shown, such as, Basic1.bas[0].

1. Support setting multi-file task No., and one file must be set, otherwise, the program can't be run.
2. The files that set the auto run No. will run at the same time, and the value can be any one, no priority.

4. Open Axis Configuration & EtherCAT Configuration (if needed)

If you need axis configuration and EtherCAT configuration function, it can be opened, please right-click empty part of "ProjectView" – "Project Setting", then check "enable axis config & EtherCAT config", and click OK.

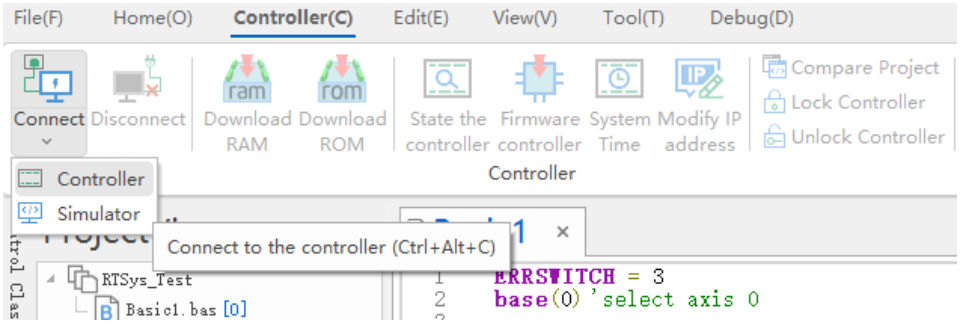


5. Edit Program

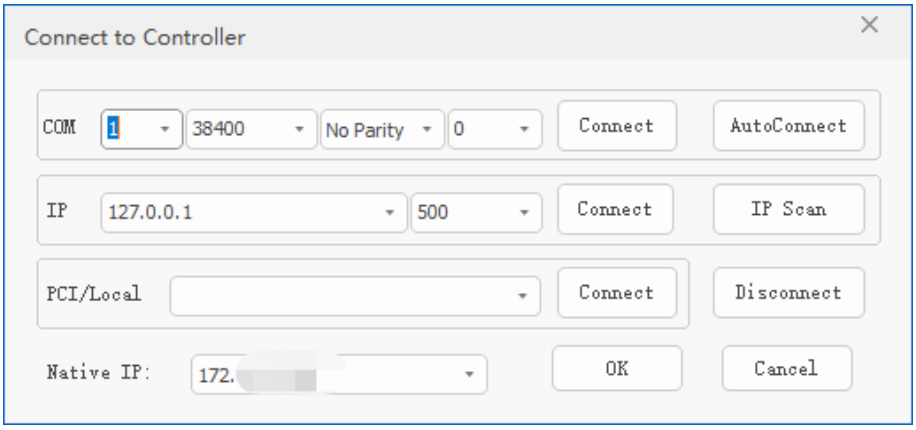
when the program is edited, click "save" the file. Then new built Basic file will be saved automatically into the file in Project zpj.

6. Connect to Controller

After editing the program, then click "Controller – Connect – Controller". If there is no controller, simulator can be used: "Controller – Connect – Simulator", then it can be opened, please note don't close the simulator window.

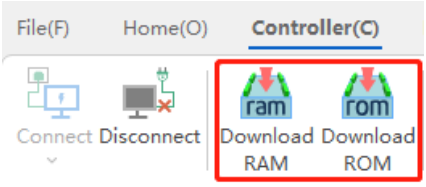


After above steps, "Connect to Controller" window will appear. At this time, you can use serial port or Ethernet. For serial port, please fill in corresponding parameters. For Ethernet, please enter IP address. And remember to click "connect". When connected successfully, in "command & output" window, you will see controller information (for example: Connected to Controller: VPLC516E Version:4.99-20190219). For the connection, please refer to "connect to controller".

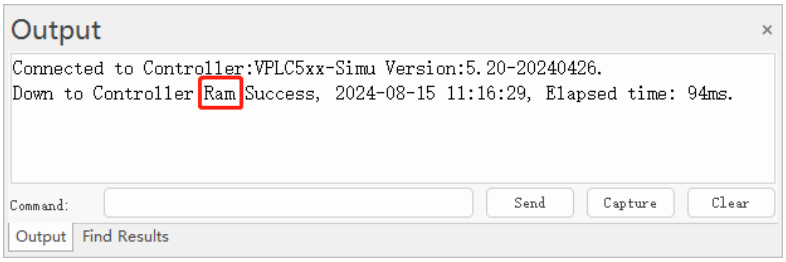


7. Download Program

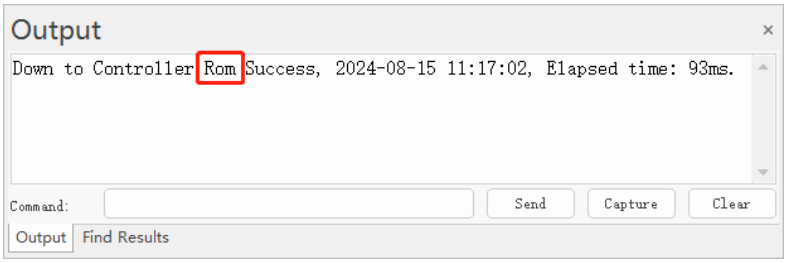
In menu bar, there are two downloading methods, **Download RAM / Download ROM**. After downloaded, also, there will tell you corresponding information in "output" window, then, program will run automatically.



--succeed in downloading RAM--



--succeed in downloading ROM--



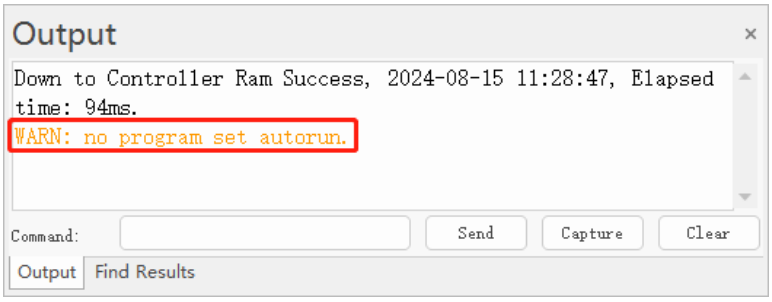
--download RAM VS download ROM--

RAM: not save program when power down

ROM: save program when power down, that is, program will run automatically according to task No., when connect to controller next time.

NOTES:

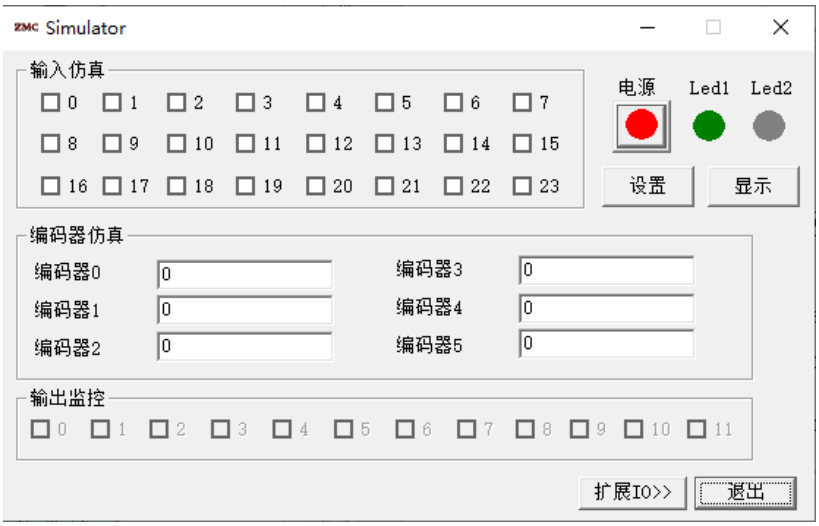
- When open the project, please select the zpj file. If only the Bas file is opened, program can't be downloaded into controller.
- ZMC00x series controllers don't support Download RAM.
- When project is not built, only .bas file can't be downloaded into controller.
- AutoRun No., 0 means the task No., that is, it runs as task 0. Task No. doesn't have priority.
- If there is only one file, AutoRun No. must be set. If there are several files, one No. must be set. If all files of whole project are not set the task No., when downloading into controller, system will give the indication: WARN: no program set autorun.



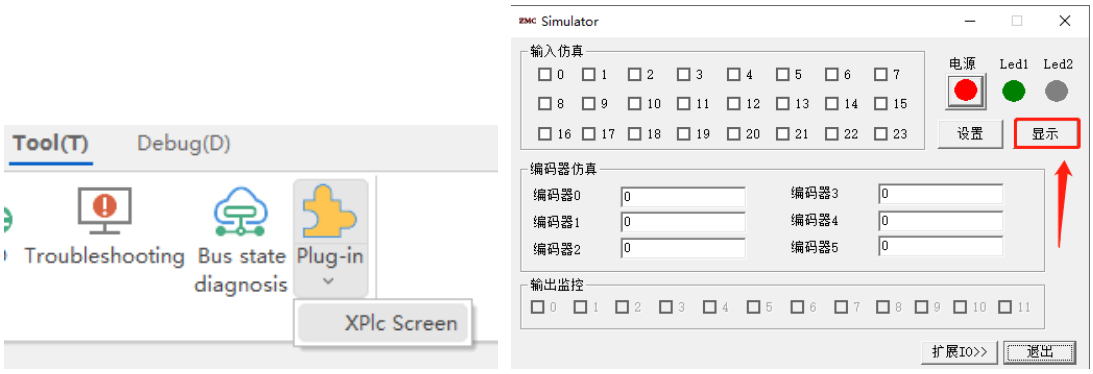
1.5. Offline Simulation

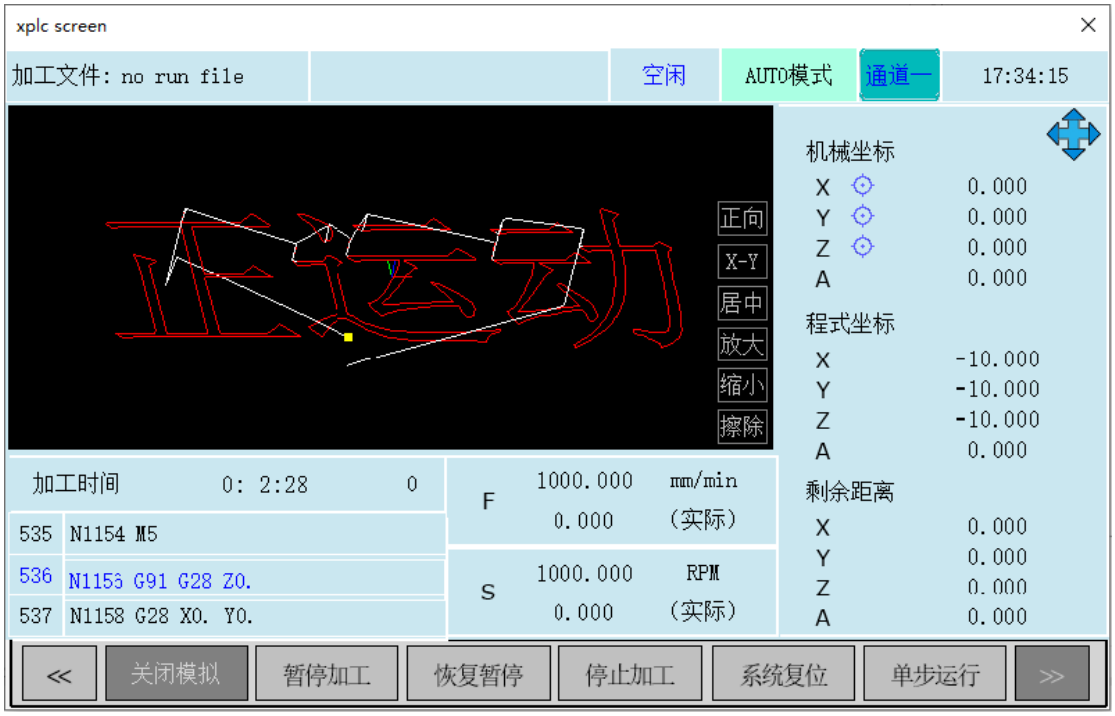
As mentioned above, RTSys supports online simulation debugging, because it is with simulator "ZMC Simulator" and configuration program simulation plug "xplc screen".

Even there is no controller, simulator can be used to test the program in advance. It only need to click "Controller – Simulator".



Xplc screen is used to show HMI interface, click “xplc screen” directly or click “显示” in the simulator window.





1.6. Command & Output

In “online command” and “output” window, much information can be checked, like, all kinds of parameters, axis motion, program running result, errors, etc. And in the program, output content can be obtained by entering corresponding commands (?, PRINT, WARN, ERROR, TRACE, etc.). Also, if you want TRACE, WARN, ERROR to be output or not, ERRSWITCH can decide it.

ERRSWITCH command is the control switch of TRACE, WARN, ERROR commands, different parameter values are with different output effects:

<table>
<tr><th>Command</th><th>EERSWITCH</th></tr>
<tr><td>Grammar</td><td>ERRSWITCH = switch
switch: switch of debugging & output
<table>
<tr><td>Value</td><td>Description</td></tr>
<tr><td>0</td><td>TRACE, WARN, ERROR commands all don’t output.</td></tr>
<tr><td>1</td><td>Only output ERROR command.</td></tr>
<tr><td>2</td><td>Only output WARN and ERROR commands.</td></tr>
<tr><td>3</td><td>TRACE, WARN, ERROR commands all output.</td></tr>
<tr><td>4</td><td>TRACE, WARN, ERROR commands all output. And watch motion commands.</td></tr>
</table></td></tr>
</table>

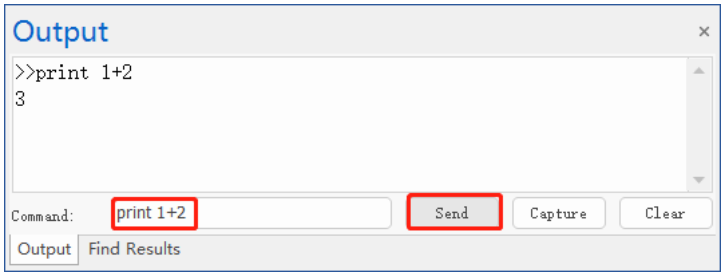
➢ **“online command” window items**

In this window, it supports online inputting the command, it only needs to enter corresponding command or function in the “command” window, then click **“send”**, or press “Enter” button, it will execute the command immediately. Note: please use English character and sign.

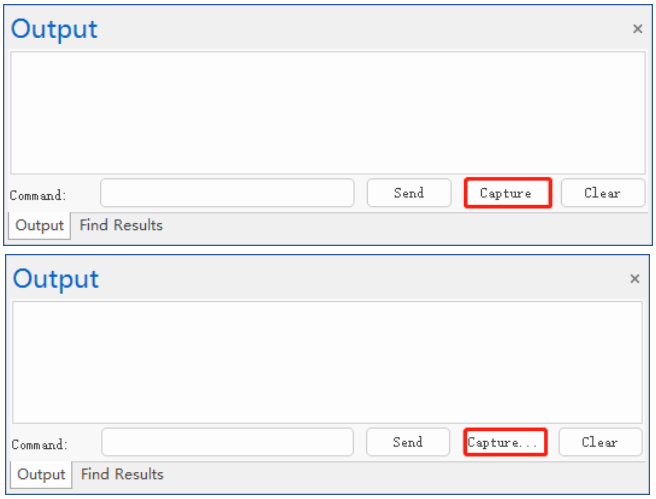
Item	Description
Command	Used to enter commands or functions.

Send	Click “send” to send the edited command / function to the controller, then execute related operation.
Capture	Used to save the printed information as txt format.
Clear	Clear all printed information.
Default Encode	There are 2 encode formats you could choose, ANSI, and UTF-8. When it appears garbled text, try to switch encode formats.

Example: “>>” means the command entered in RTSys “command”, print 1+2 means calculation.



This function can be used after connection (controller / simulator). **“Clear”** can clear all contents in the window. **“Capture”** is to save some content, when you click it, one “save as” window will appear, you can change the file name, then click save, at this time, all following contents output in this window will be saved in your PC and one .txt file will be generated, and please note the state will become “capture…” from “capture”, then it stops until “capture…’ is pressed, in generated .txt file, captured content will be shown.



If there are some **errors**, it will show in this window in **RED font:**

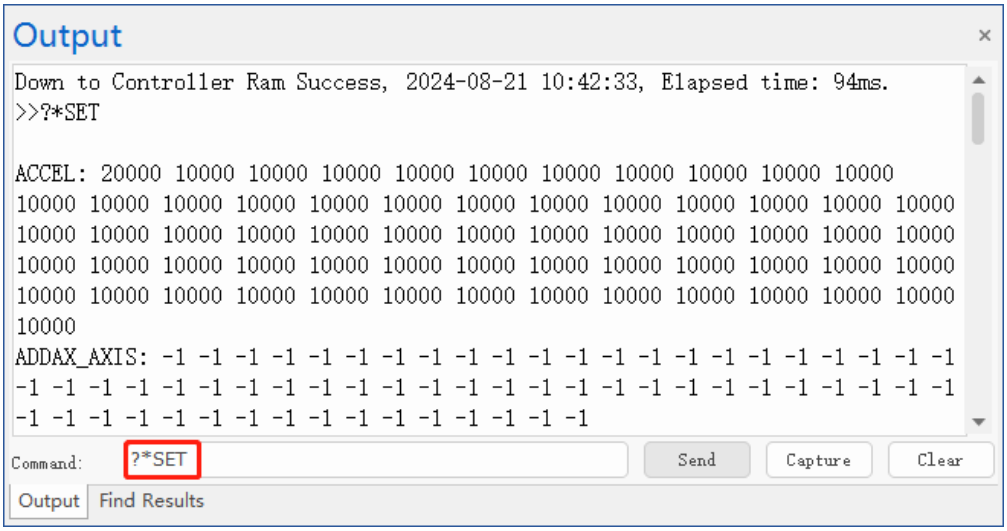
```
Connected to Controller:VPLC532R Version:5.20-20250514.
Down to Controller Ram Success, 2025-11-19 14:35:41, Elapsed time: 141ms.
file:"BASIC1.BAS" line:9 init warn:2033:Dim:ATYP undefined.
file:"BASIC1.BAS" line:9 task:0 stop of error:2029:Label name is invalid.
```

If there are some **warns**, it will show in this window in **ORANGE font:**

```
Down to Controller Ram Success, 2025-11-19 14:37:05, Elapsed time: 157ms.
WARN: no program set autorun.
```

--Commonly Used "PRINT" Commands--

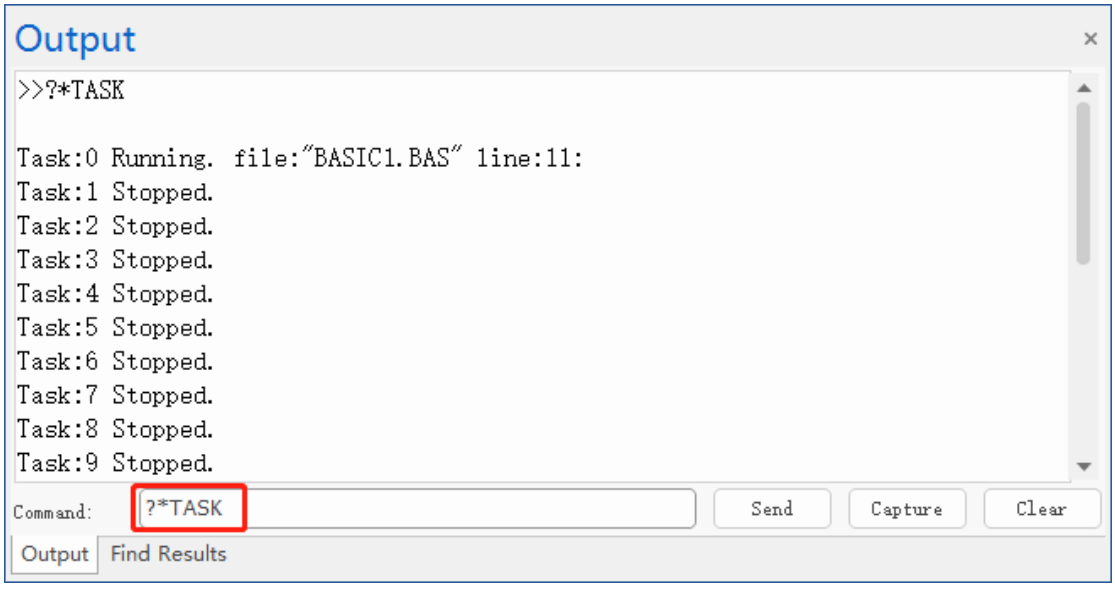
- **?*SET:** print all axes' parameter values and system parameter values



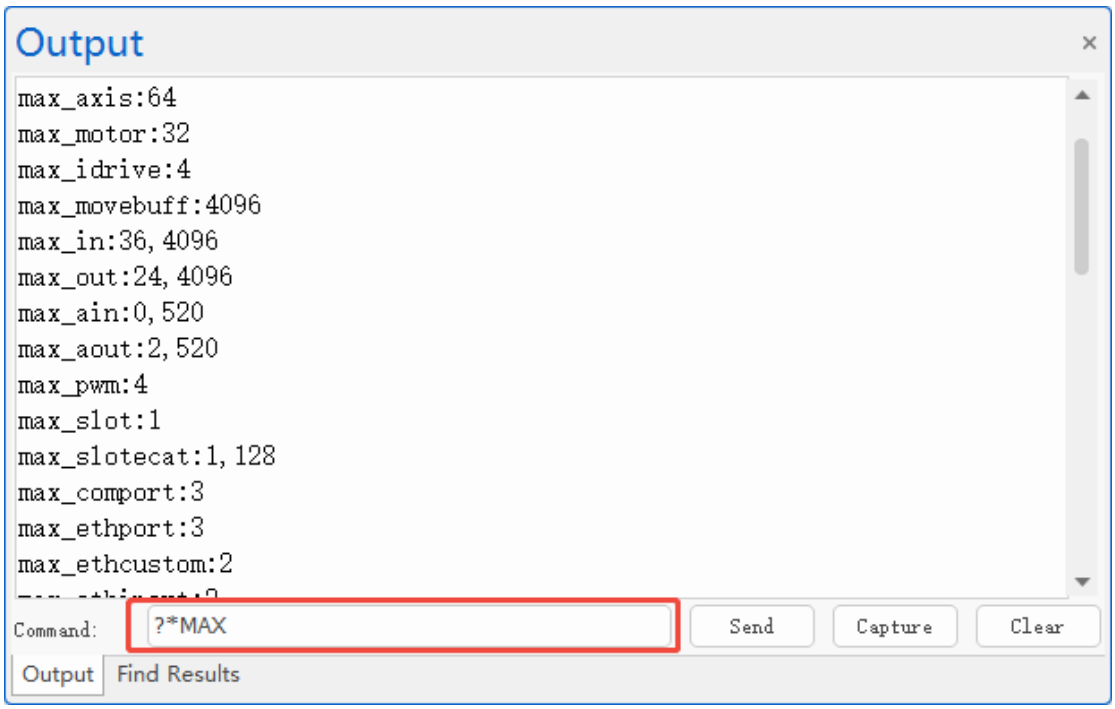
- **?*TASK:** print task information

When the task is normal, it only prints the task state.

When the task is wrong, it will print error task No., and specific error line.



- **?*MAX:** print all specification parameters (here, the example of "simulator"):

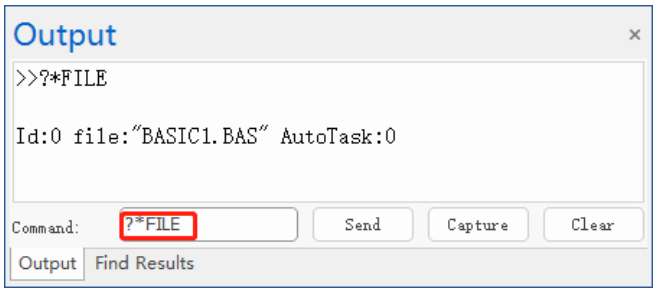


Parameter Information	Description
max_axis:64	Max axes of all axes
max_motor:32	Max controllable motor axes
max_idrive:4	The number of supported drives
max_movebuff:4096	Max motion buffer of each axis or axis group
max_in:36,4096	36: controller itself supported IN numbers, here, the simulator supports 36 inputs. 4096: controller max supported IN numbers, here the simulator IN can be up to 4096 through expansion module.
max_out:24,4096	24: controller itself supported OUT numbers, here, the simulator supports 24 inputs. 4096: controller max supported OUT numbers, here the simulator OP can be up to 4096 through expansion module.
max_ain:0,520	0: controller itself supported AIN numbers, here, the simulator doesn't support analog inputs. 520: controller max supported AIN numbers, here the simulator AIN can be up to 520 through expansion module.
max_aout:2,520	2: controller itself supported AOUT numbers, here, the simulator only supports 2 analog outputs. 520: controller max supported AOUT numbers, here the simulator AOUT can be up to 520 by expansion module.

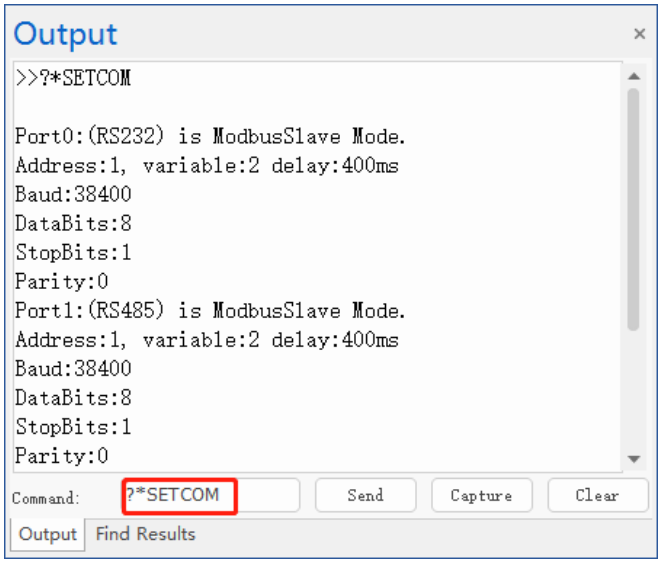
max_pwm:4	How many PWM outputs
max_slot:1	How many bus
max_slotecat:1,128	How many slot No., and max slot numbers.
max_comport:3	How many serial ports
max_ethport:3	Ethernet communication with PC / API function
max_ethcustom:2	Custom ethernet communication
max_ethiport:2	Ethernet communication between Zmotion controllers
max_flashnum:9999	How many FLASH blocks
max_flashsize:20480	Space of each FLASH
max_nand:262144KB	Space of NandFlash in total
max_nandremain:262144KB	Remain space of NandFlash
max_softhwout:4,8,isolate	Hardware comparison output (isolate means independent)
max_hwda:1,8	Hardware comparison analog output: 1: itself supported channels, 8: max expanded channels.
max_pswitch:64	How many software position comparison outputs in total
max_file:111	How many files the system supports at most
max_3file:8	How many 3 files the system supports at most
max_task:22	How many tasks
max_timer:1024	How many timers
max_loopnest:8	Inner loop or selection times
max_callstack:14	How many stack layers (called by subprogram)
max_local of one sub:32	How many local variables of SUB
max_vr:8000	How many VR registers
max_table:320000	How many TABLE registers
max_modbusbit:8000	How many MODBUS_BIT bit registers
max_modbusreg:60000	How many MODBUS_REG word registers
max_var:20480	How many variables at most (global + file variables)
max_array:8000	How many arrays at most (global + file arrays)
max_arrayspace:4000000	Total space of all arrays
max_sub:8000	How many SUB programs at most
max_edgescan:1024	How many rising / falling edges that can be scanned
max_lablelength:25	How long custom characters are (array / variables ..)
max_hmi:2,x:1920 y:1080 (max_hmi:2,size:5120kb)	How many remote Hmi, the max Hmi resolution (how many remote Hmi, Hmi showing size)

max_zvlatch:4	How many channels of vision image latching
max_zvtask:3	How many tasks of vision
SERVO_PERIOD:1000 min:1000, max:1000	System period default time: 1000 minimal time: 1000 maximum time: 1000
Hardid:7532-0 Version_build:20250514	Hardid: hardware version No. Version_build: firmware version No.
function support: Coder Cam MultiMove Circ Merge Frame Robot Zvision	Valid functions

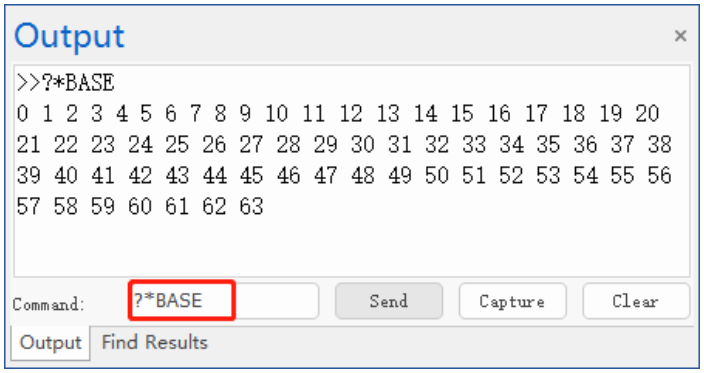
- **?*FILE:** print program file information



- **?*SETCOM:** print current serial port configuration information

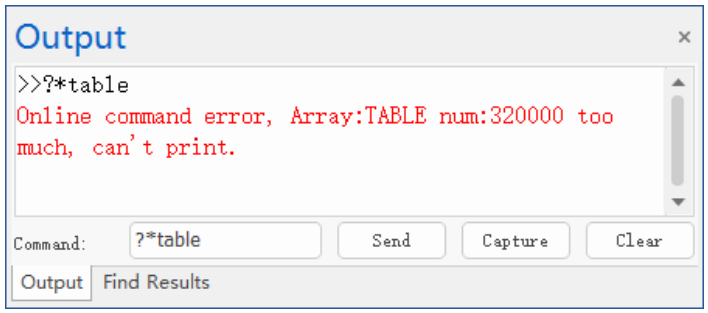


- **?*BASE:** print current task's BASE list (firmware version should be above 140123)



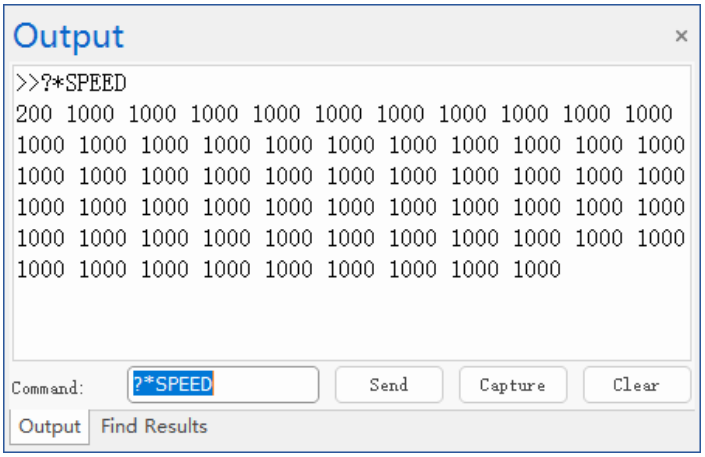
➢ **?*array name:** print array's all elements, please note "array length can not be too long" (you need to define the array and assign it in advance).

```
>>?*data
0:1
1:2
2:3
3:4
```



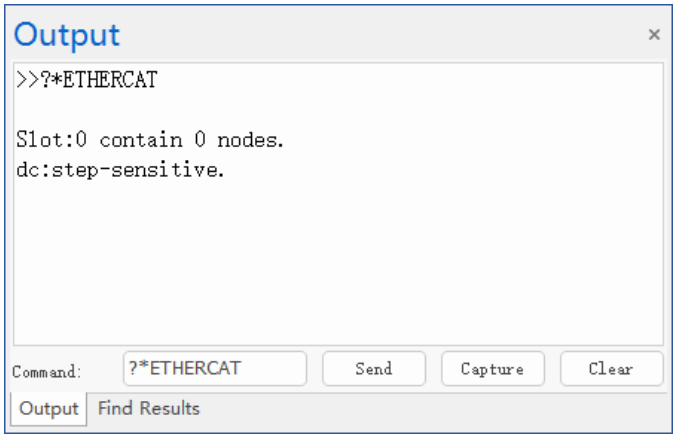
➢ **?*array name (n) / ? array name (n):** print one single element of the array

➢ **?*parameter name:** print all axes' single parameter value



➢ ?*ETHERCAT: print ethercat bus connection configuration state

- slot No. is 0, no device is connected



- slot No. is 0, 1 device is connected

```
>>?*ETHERCAT

Slot:0 contain 1 nodes.
dc:ecat-sensitive.Lostcount:0-0.
Node:0 status:1 manid:41bh productid:1ab0h axises:8 Alstate:8
Node_profile:0.
BindAxis:0 Drive_profile:0 Controlword:fh drive_status:df37h Drive_mode:8h
target:0h encode:0h.
BindAxis:1 Drive_profile:0 Controlword:fh drive_status:df37h Drive_mode:8h
target:0h encode:0h.
```

More, please refer to this form:

Parameter Information	Description
Slot 0 contain 2 nodes	There are 2 devices connected on the slot 0.
Lostcount 0-0	How many packages lost
Node	NODE No. of device

Status	Device connection state, please refer to NODE_STATUS
Manid	Manufacturer ID
Productid	Device ID
Axises	How many axes of device in total
AL Status	Device OP state
Node_profile	Device Profile configuration
Bindaxis	Controller axis No. is to be mapped
Drive_profile	Device PDO sending and receiving configuration
Controlword	Control word
Drive_status	Device now state, please refer to NODE_STATUS
Drive_mode	Device control mode
Target	Motor position
Encoder	Encoder position

- ?*RTEX: print RTEX bus connection configuration state

```
>>?*RTEX

Slot:1 contain 1 nodes.
Lostcount:0-0.
Node:0 status:3 man:Panasonic devicetype:31h axises:1 Alstate:1.
BindAxis:-1 Drive_profile:0 Controlword:0h drive_status:0h target:0h encode:0h.
```

More, please refer to this form:

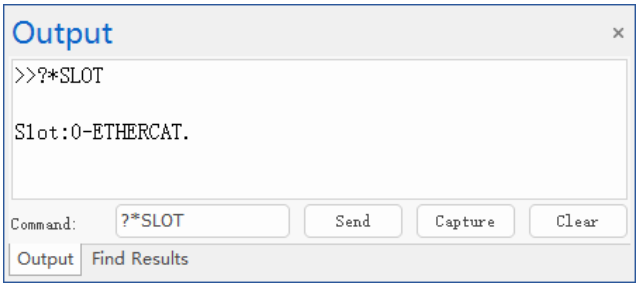
Parameter Information	**Description**
Slot 1 contain 1 nodes	There are 1 device that is connected on the slot 1.
Lostcount 0-0	How many packages lost
Node	NODE No. of device
Status	Device connection state, please refer to NODE_STATUS
Manid	Manufacturer ID
Productid	Device ID
Axises	How many axes of device in total
AL Status	Device OP state
Bindaxis	Controller axis No. is to be mapped
Drive_profile	Device PDO sending and receiving configuration
Controlword	Control word
Drive_status	Device now state, please refer to NODE_STATUS

Target	Motor position
Encoder	Encoder position

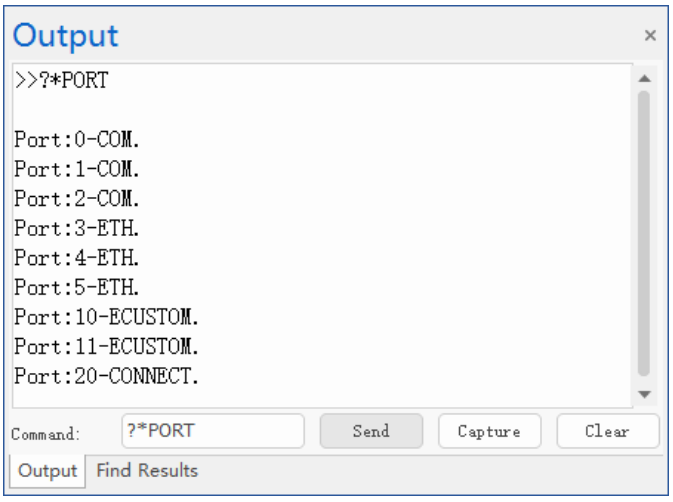
- **?*FRAME:** print robotic arm parameters, firmware version should be 161022 or above.

```
>>?*FRAME

BASE(0,1,2,3,4,5)
CONNFRAME(6,0,6,7,8,9,10,11)
```

- **?*SLOT:** print controller slot port information



- **?*PORT:** print all PORT communication ports



COM: serial port channel

ETH: ethernet port channel

LOCAL: local interface

ECUSTOM: custom ethernet channel

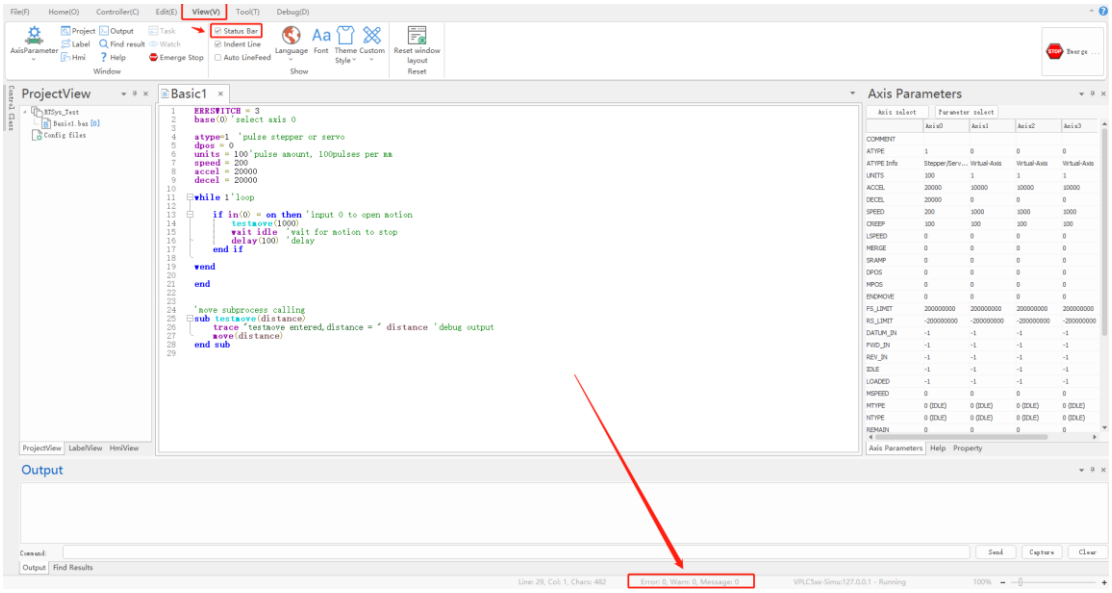
CONNECT: controller interconnection channel

1.7. Alarm Checking

Sometimes, there is no doubt that errors will occur when the RTSys program is running. In RTSys, there are 3 ways to check errors: status bar alarm, AXISSTATUS, alarm information output.

➢ **Status Bar**

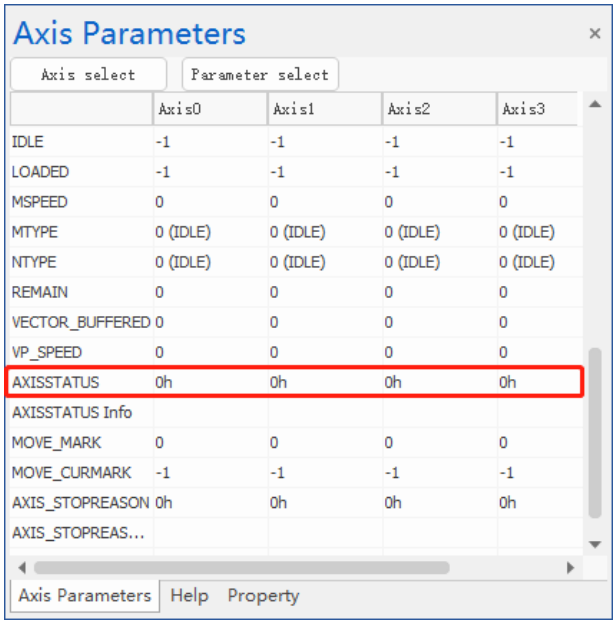
When the program has errors or the axis runs abnormally, “alarm” signal will be triggered. In RTSys bottom status bar, red means alarm. If it can’t be seen, please check “status bar” in the View.



➢ **AXISSTATUS**

When running, if axis is abnormal, AXISSTATUS can check it. You can add this command in the program, or enter this command in the “command” window (?AXISSTATUS(axis No.)), or check the axis parameter window “AXISSTATUS” directly. Please note the print value is decimal system.

In addition, AXISSTATUS values can be checked in “axis parameter” window, and the values shown are in hexadecimal system.



Bit	Description	Value	
1	Alarm: Follow-Up Error Exceeds.	2	2h
2	Communication with Remote Axis Error	4	4h
3	Remote Driver Error	8	8h
4	Positive Hard Limit	16	10h
5	Negative Hard Limit	32	20h
6	Origin Searching	64	40h
7	Hold Signal IN at HOLD Speed	128	80h
8	Error: Follow-Up Error Exceeds.	256	100h
9	Positive Soft Limit Exceeds	512	200h
10	Negative Soft Limit Exceeds	1024	400h
11	CANCEL in Process	2048	800h
12	Pulse Frequency > MAX_SPEED. Please Low the Speed / Reset MAX_SPEED.	4096	1000h
14	“Robot” Command Coordinates Error	16384	4000h
18	Power Abnormal	262144	40000h
19	Buffer of Precision OUT Exceeds	524288	80000h
21	Fail to Trigger Special Commands in Motion.	2097152	200000h
22	Alarm Signal Input	4194304	400000h
23	Axis Paused	8388608	800000h

➢ **Alarm Outputs**

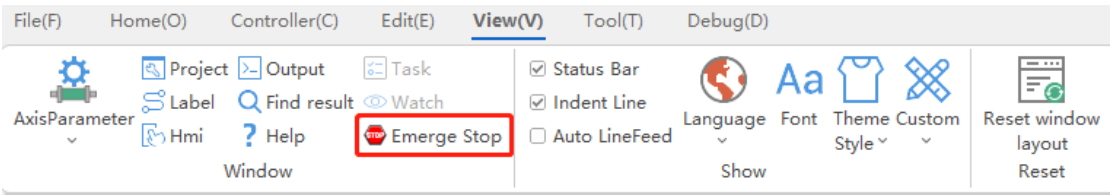
In “command & output” window, it will output alarm information and error code automatically.

```
Down to Controller Ram Success, 2024-04-19 14:10:35, Elapsed time: 140ms.
ZBasic stop running because of 1 init errors.
file:"BASIC1.BAS" line:23 init error:2032:Invalid char is met.
file:"BASIC1.BAS" line:23 init warn:2033:Unknown name:MANLINE
file:"BASIC1.BAS" line:23 init warn:2032:Invalid char is met.
```

1.8. Emergency Stop

The “emergency stop” button can stop all motions immediately.

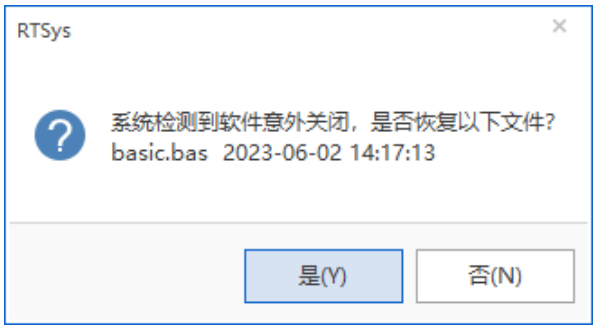
It can be used to avoid some things that are out of control while debugging. When it is pressed, controller will be in standby state (showing in status bar).



1.9. Automatically Reserve

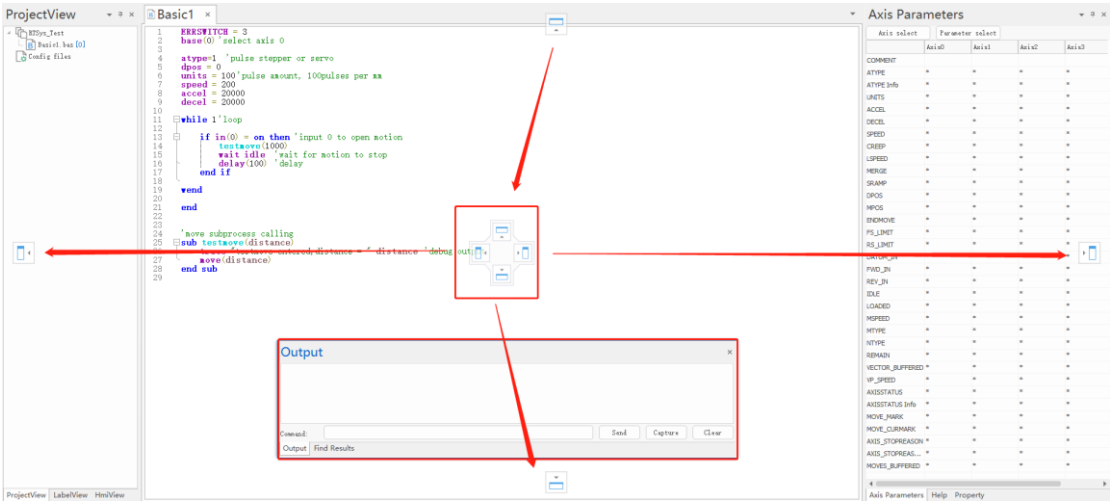
RTSys adds “auto-reserve” function. When RTSys is OFF abnormally, program file can be resumed (reserve once each 10 minutes – default). And when you open RTSys again, below window will appear, you can determine resume or not.

Note: when it reserved, .tmp file will be generated, don’t delete this file!



1.10. Window Position

In RTSys, there are many windows and views, and actually these windows can change the position freely. They can be fixed, also can be at any position. Choose the window, then press it to drag it through mouse, then it can be seen one indication mark, which means the window can be put in these places.



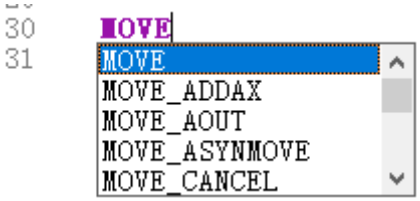
1.11. Programming Guidance

In RTSys, there are 3 programming methods, Basic, PLC, HMI. These 3 can be programmed at the same time. Let's see them one by one.

1.11.1. Edit Basic

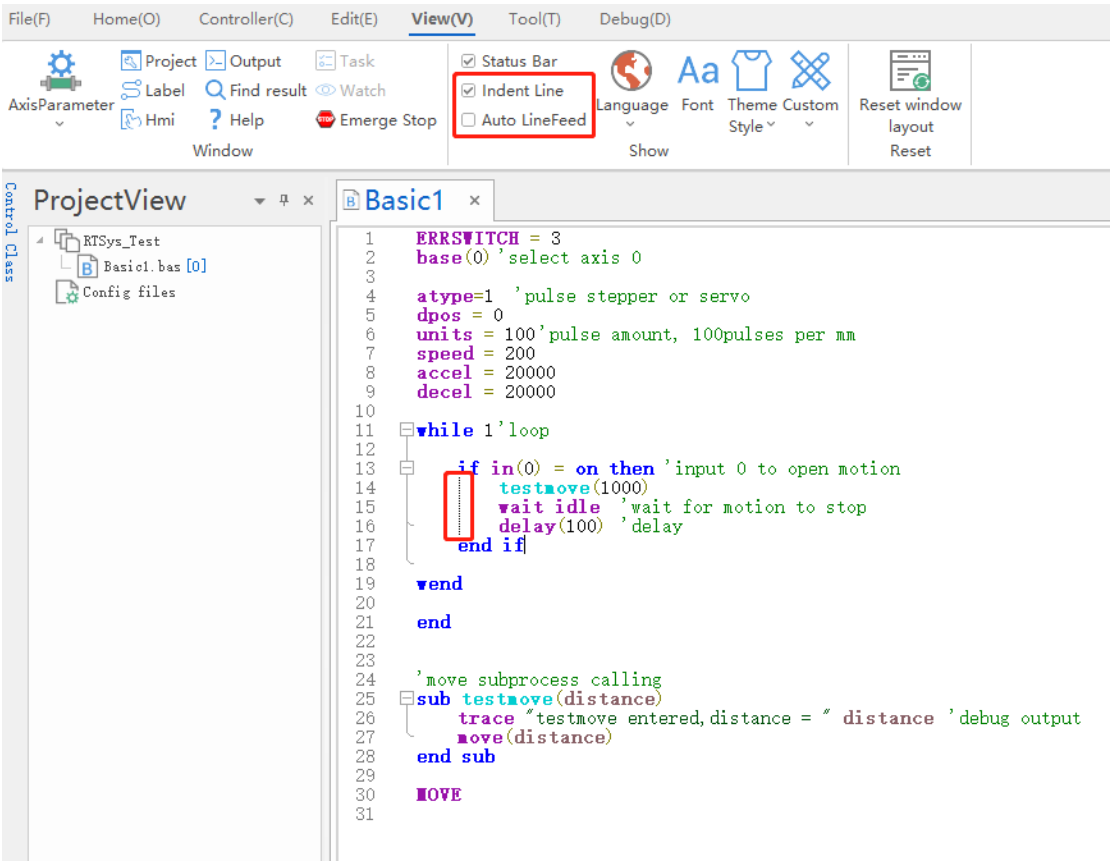
At first, build one new .bas file or open the .bas file. Then edit it. Following shows basic operations:

➢ When you enter command, corresponding commands will show immediately.



➢ Different contents (command, variable, note, etc.) are with different showing colors.

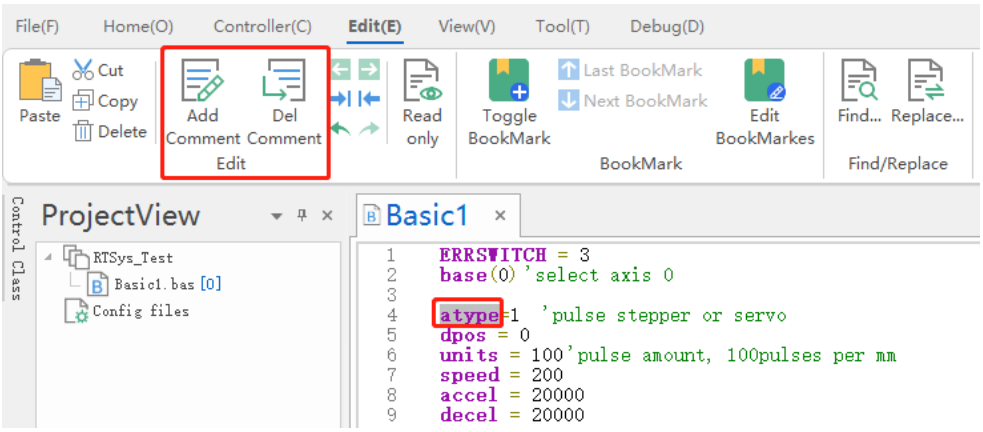
➢ There is "indent line" function, showing in **IF**. And there is "auto linefeed" function, that is, it can change the line automatically when the program window changes.



➢ "Add Comment": change the program into comment. "Delete Comment": change the comment into program". There are two methods:

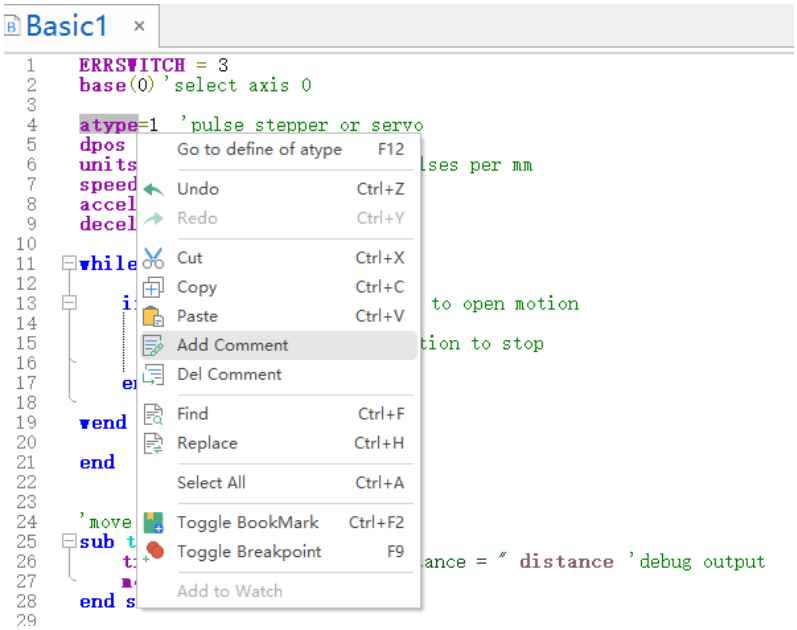
- **Operate in Menu**

Choose the program, in menu, click "Edit" – "Add Comment", then selected program will become comment, and it will show in green. If you want to cancel it, click "Edit" – "Delete Comment".

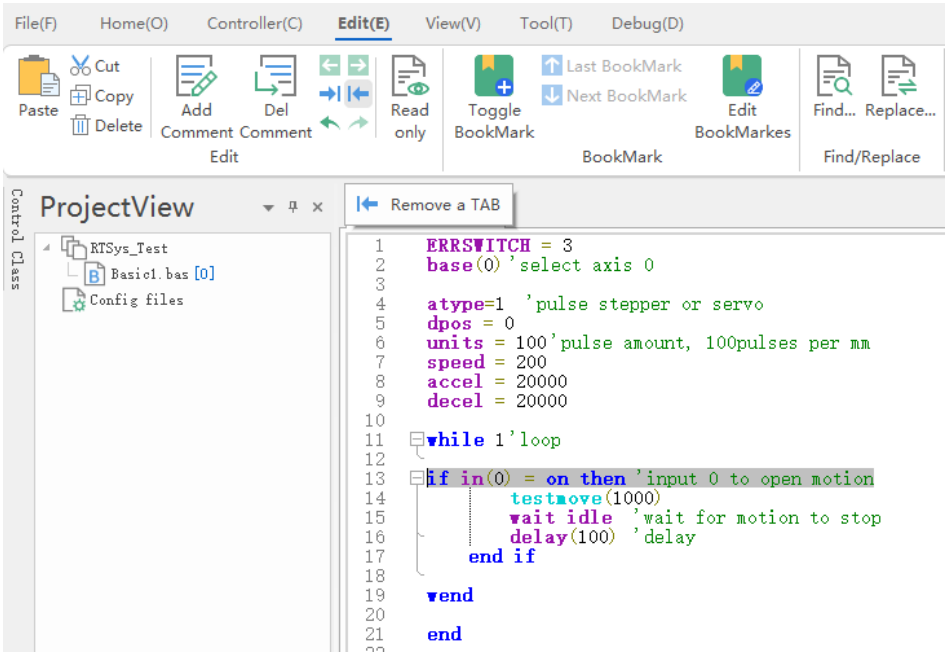


- **Operate via Mouse**

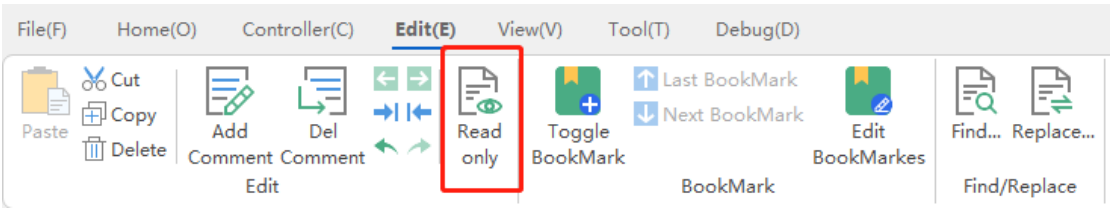
Also, choose the program, then right click this program, one window will appear, click "add comment' or "delete comment" to achieve functions.



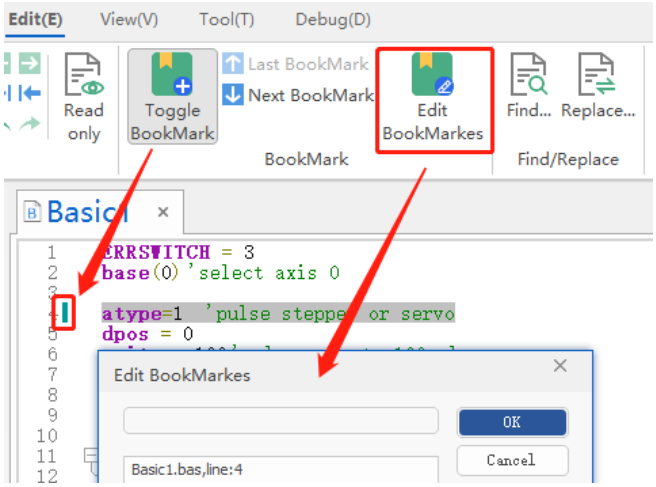
➢ In menu – "Edit", "Insert TAB" and "Remove TAB" are used to control program indentation. For example: operate "remove TAB" for "FOR", one time, it will indent once.



➢ "Read-Only" means the program can't be edited. Click "Read-Only" in Edit, then program files under current project are in read mode. Click again to cancel it.

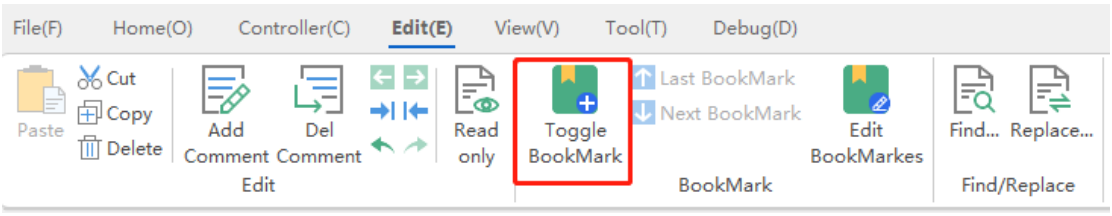


➢ "Toggle BookMark" (set / delete book mark) is used to add the bookmark for Basic / PLC program, then you can find one certain program quickly. And in "edit bookmark", used program line can be checked. For the Basic program that has been set the bookmark, one green column line will be shown at the beginning, for PLC program, one M mark will be shown. There are two methods:



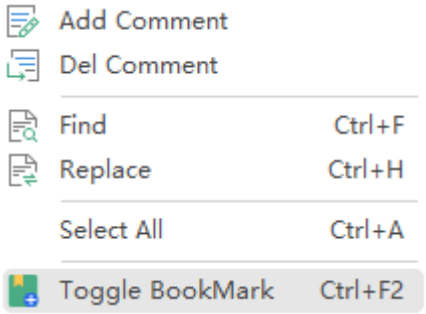
- **Operate in Menu**

Select one certain line program of Basic / PLC, click Edit – Toggle Bookmark. Click it to set, click again to delete it.

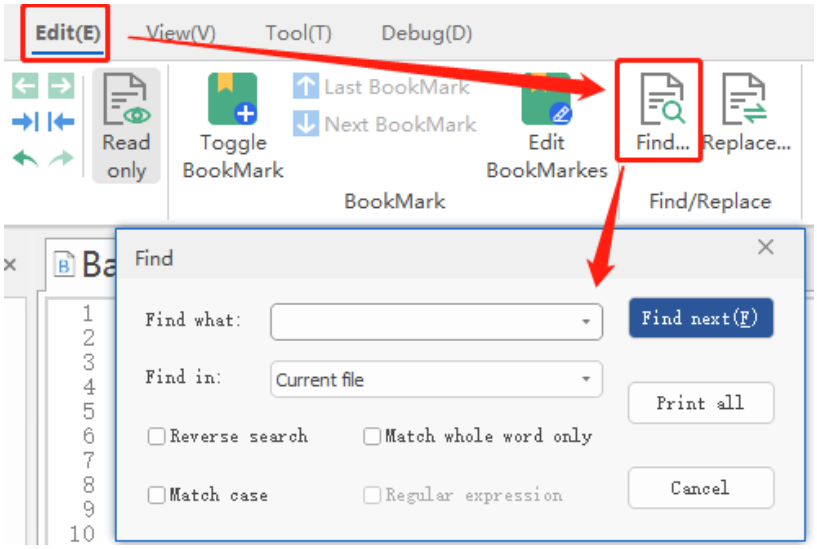


- **Operate via Mouse**

Select one certain line program of Basic / PLC, right click the mouse to set directly.



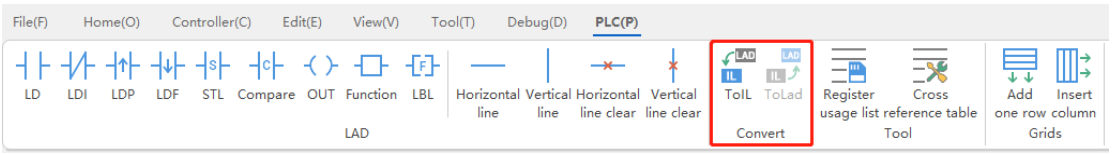
➢ In menu "Edit" – "Find", you can find needed information.



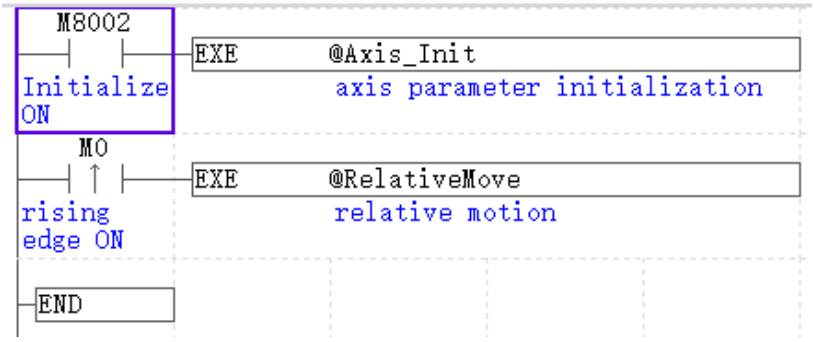
➢ In menu "Edit" – "Replace", you can find and replace needed at the same time.

1.11.2. Edit PLC

RTSys provides two programming methods, LAD (ladder of diagram) and IL (instruction list). And these two can be switched directly, not case sensitive. For LAD, it is intuitive. For IL, you should be familiar with PLC commands. For PLC command, RTPlc helper manual can help you.



--LAD--

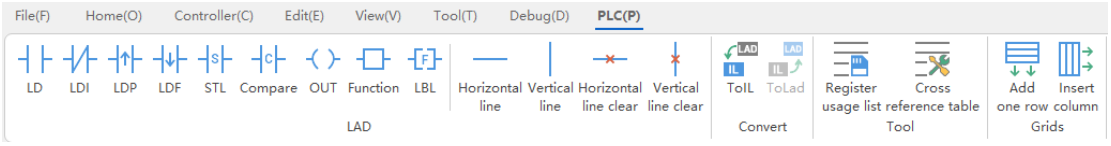


--IL--

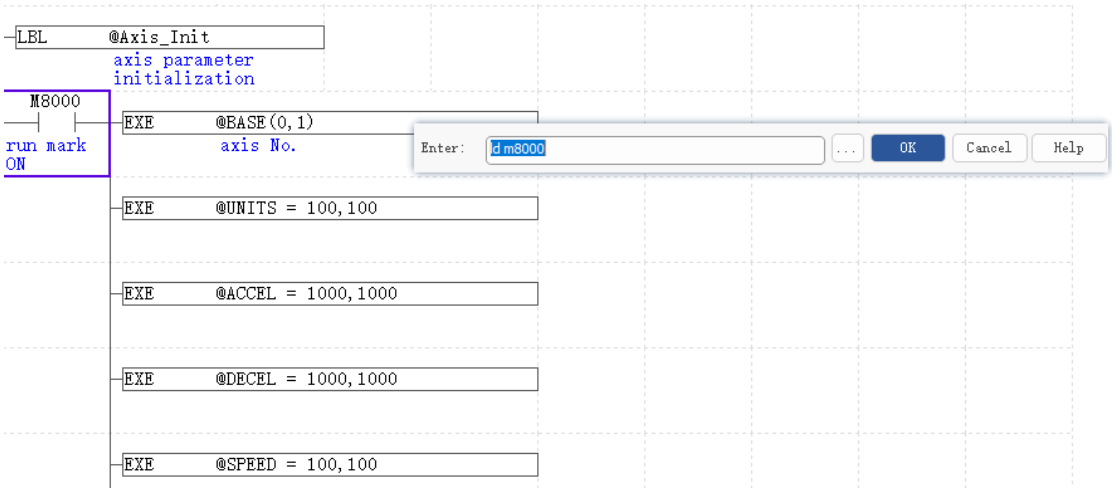
```
1   ld m8002
2   exe @Axis_Init
3   ldp m0
4   EXE @RelativeMove
5   end
6   lbl @Axis_Init
7   ld m8000
8   EXE @BASE(0,1)
9   EXE @UNITS = 100,100
10  EXE @ACCEL = 1000,1000
11  EXE @DECEL = 1000,1000
12  EXE @SPEED = 100,100
13  EXE @DPOS = 0,0
14  EXE @MPOS = 0,0
15  sret
16  LBL @RelativeMove
17  ld m8000
18  EXE @BASE(0,1)
19  exe @Trigger
20  exe @MOVE(300,400)
21  sret
22
```

➢ **How to Edit**

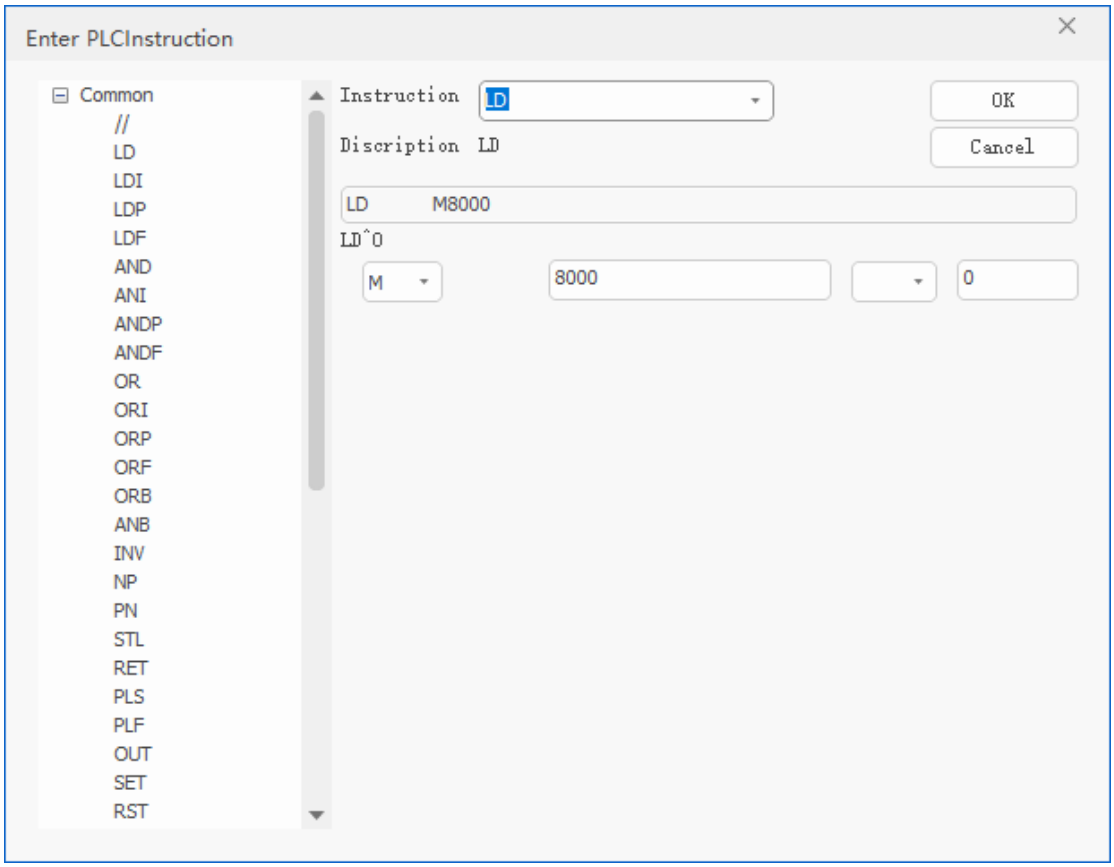
Open PLC file, and in menu, find PLC, there are many buttons, you can insert buttons to achieve corresponding functions.



In corresponding unit, double click or enter the command directly. It can enter the command or modify parameter, please remember to click OK. And the character should be English state.



The "…" button is used to open PLC command inputting frame, it can select command and command operands:



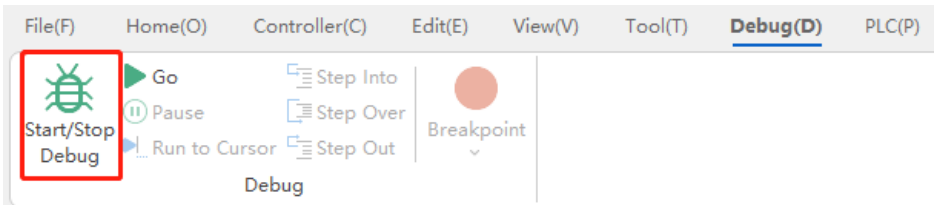
And please note END must be edited at the end of the program, otherwise, it can't run.

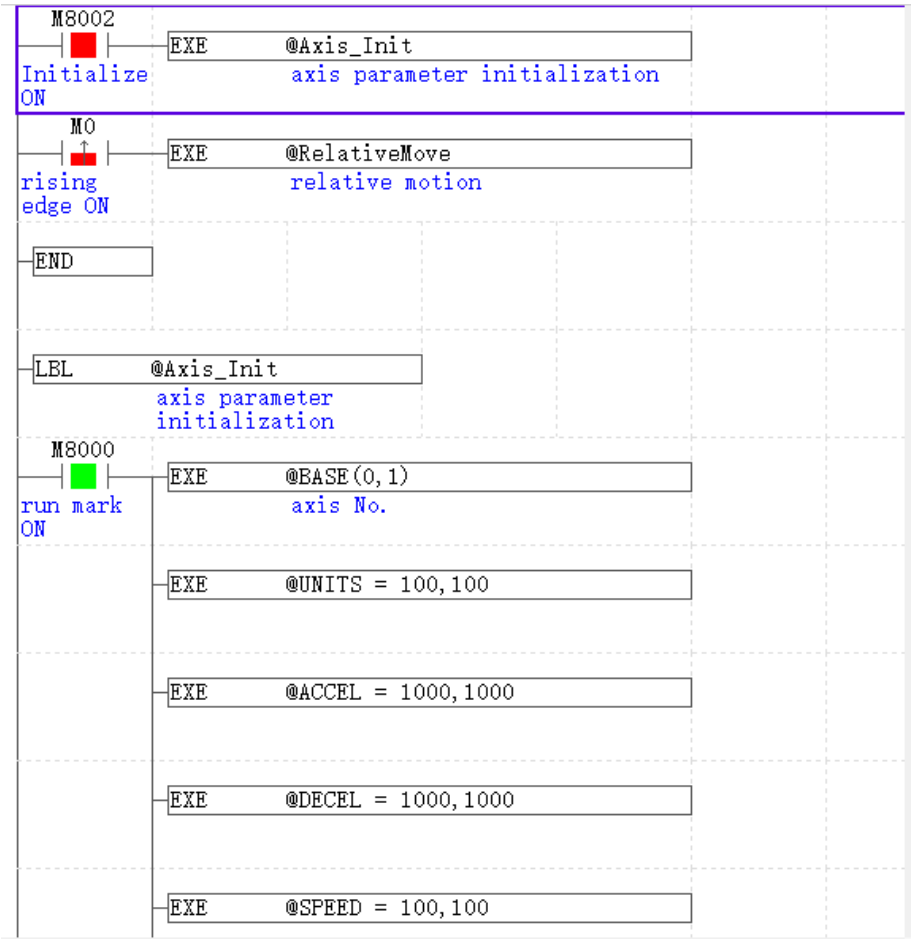
In addition, Basic program and PLC program can call each other. In PLC, use EXE@ command to call Basic command. More details, please refer to PLC manual.

➢ **How to Debug**

When the PLC program is edited well, in menu "Debug" – "Start/Stop Debug", click once, enter debugging mode. While debugging, for LAD, green means conducted, red means disconnection, and it will show current value of register above the register.

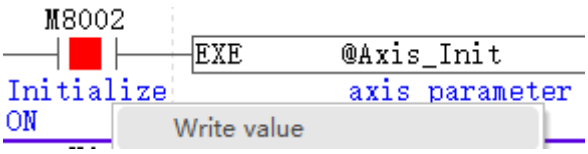
Note: for LAD mode, don't support breakpoint debugging, for IL, it supports.

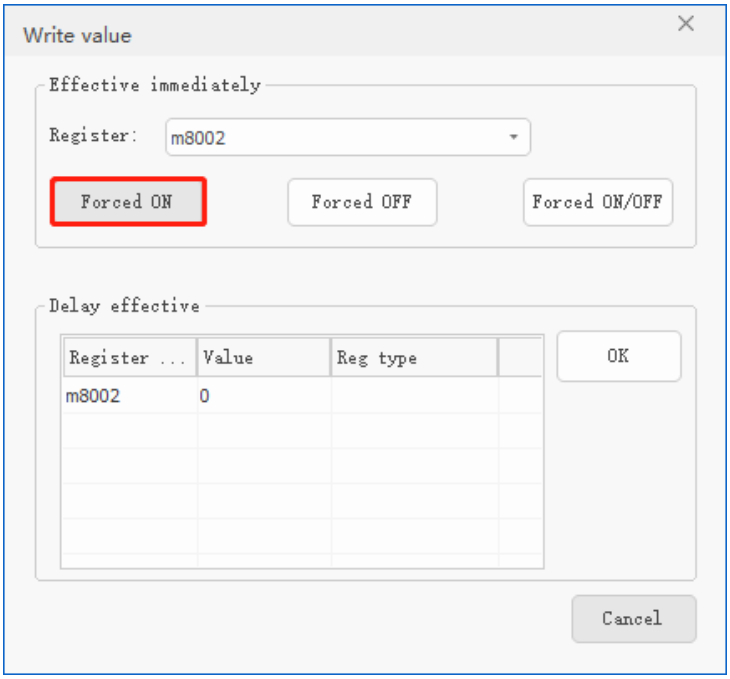




➢ **How to Write Value**

While debugging, select one content, right-click the “write value” to open the window, then you can edit or modify bit variable M / byte variable D values quickly. Below takes one example to write data ON or OFF for M0: it only shows current selected content’s register, click “Forced ON”, then M0 state will become ON from OFF, which means conducted, green.



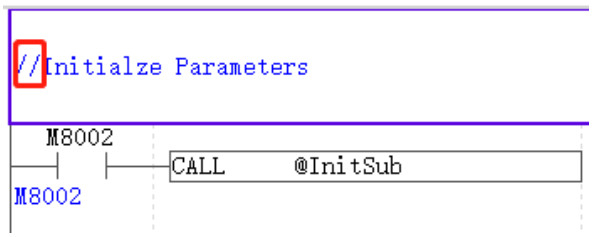


➢ **How to Note**

There are 3 ways to add the comment: use PLC to note the command, right click “Notes Edit”, menu “Controller” – “Notes”.

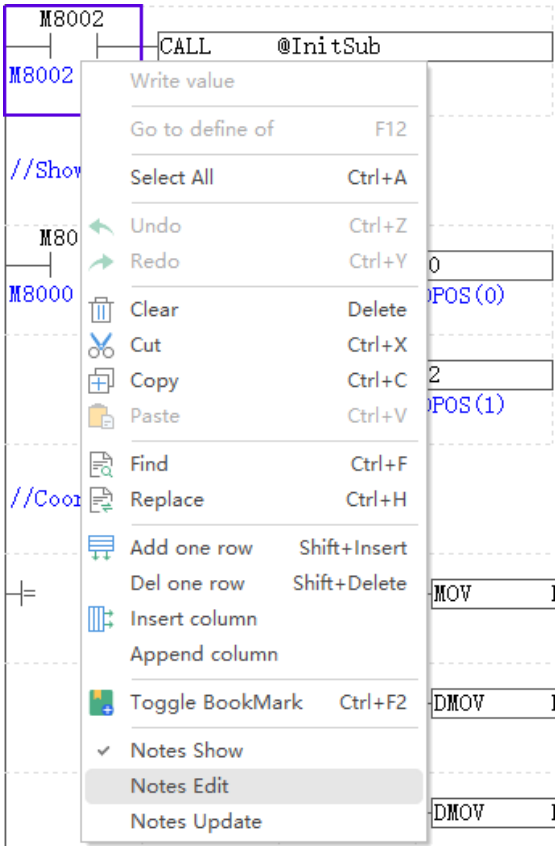
- **Note Command: //**

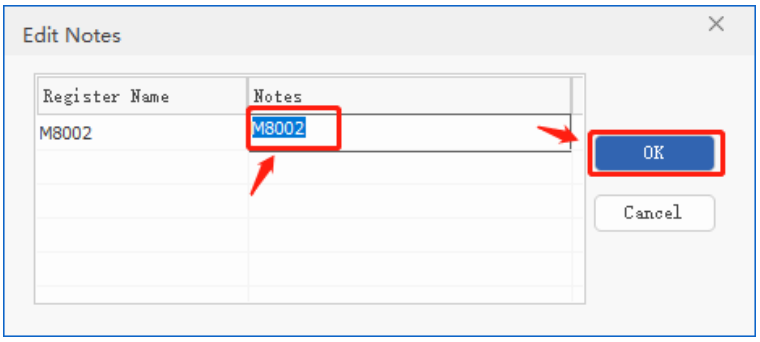
Use this command to note, note should be at another new line, usually add the note before one certain functional module to compensate this function’s corresponding information.



- **Note Edit**

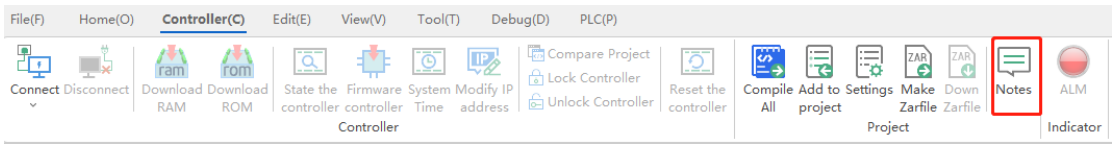
Select the component, then right click “Note Edit”.





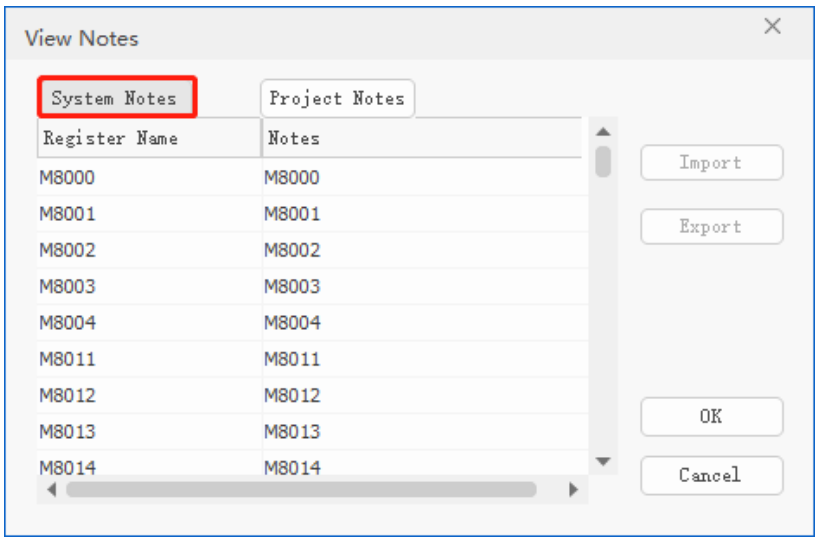
- **"Controller" – "Notes"**

In menu, click "controller", then find "Notes".



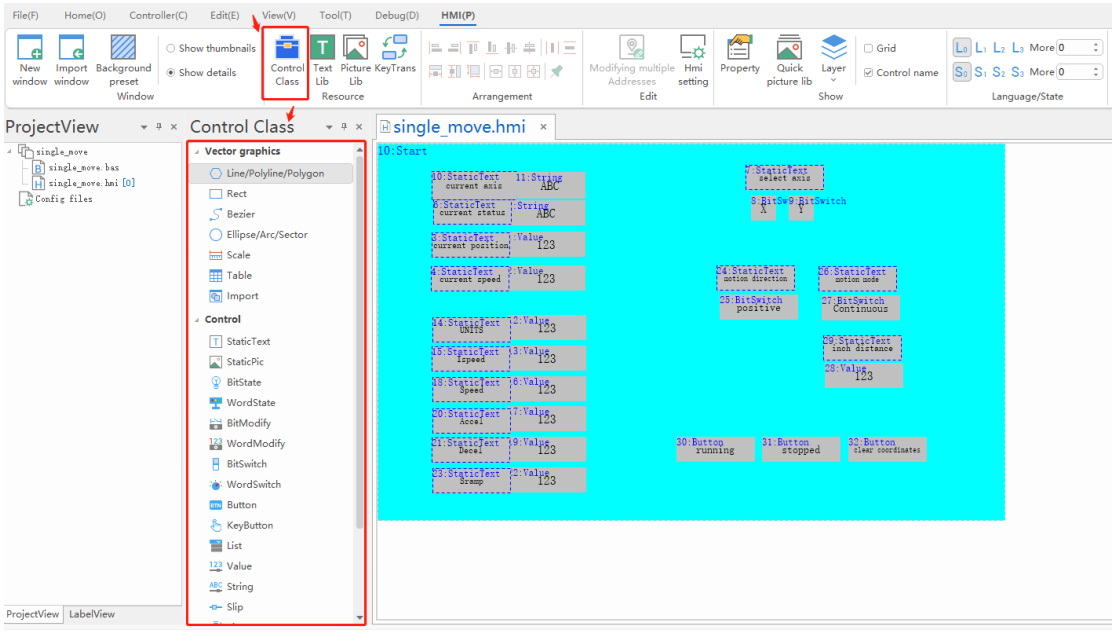
"System Notes": mainly for PLC special relay M and special register D.

"Project Notes": you can enter register name and notes, which is similar to "note edit", and it supports notes importing and exporting.

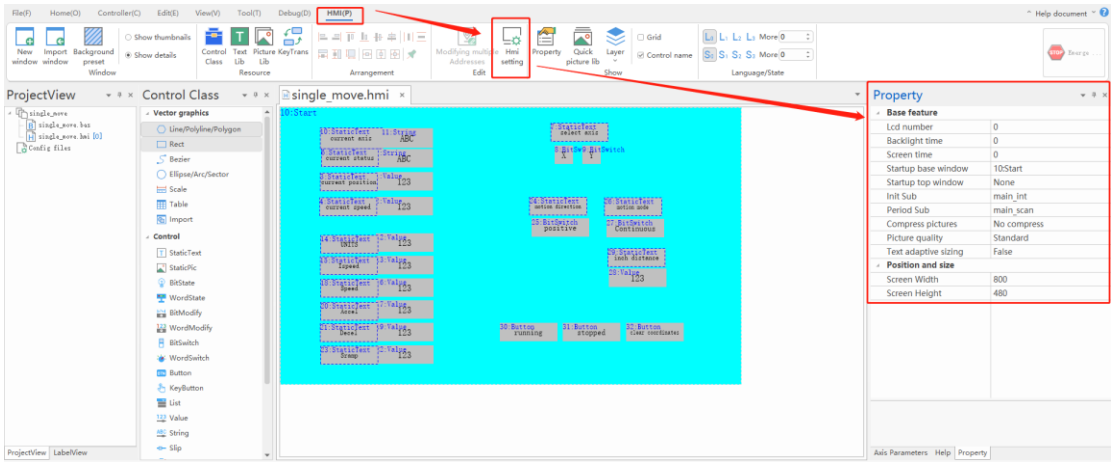


1.11.3. Edit HMI

HMI programming is used to show information in display screen, using TOUCH method to execute corresponding operations. In menu, “view” – “control class”, select needed component. More details, please refer to RTHmi manual.



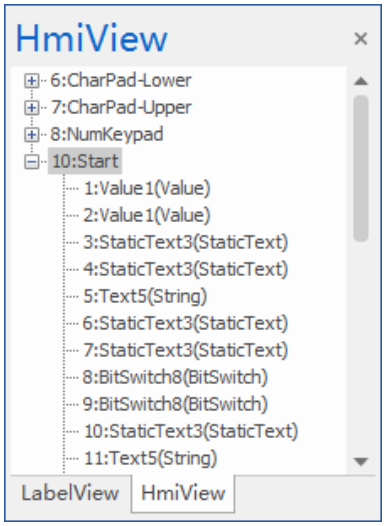
Before running, please set HMI file at first. Click menu HMI – Hmi Setting, and do configuration in Hmi setting window.



It can be seen here shows and sets properties. For "Position and Size", it sets the resolution according to display screen size, common screen is 7inch, the size is 800*480. If the screen is 10.1 inch, the resolution should be 1024*600. For "Init Sub", it is the function that is only called once after powered on, which is defined in the Basic file, and the function definition must be global (GLOBAL). For "Period Sub", it is the function that is scanned continuously, also, defined in the Basic file, and type should be global. However, these two functions are optional, depend on your real needs. Except these two, custom component drawing function or refresh function, or component action "call sub" can be used to call Basic function.

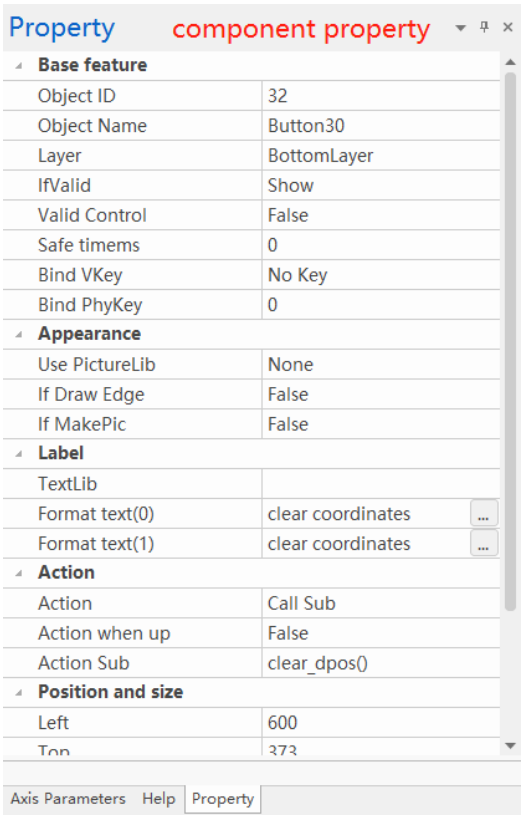
--how to open window / component "Property" window--

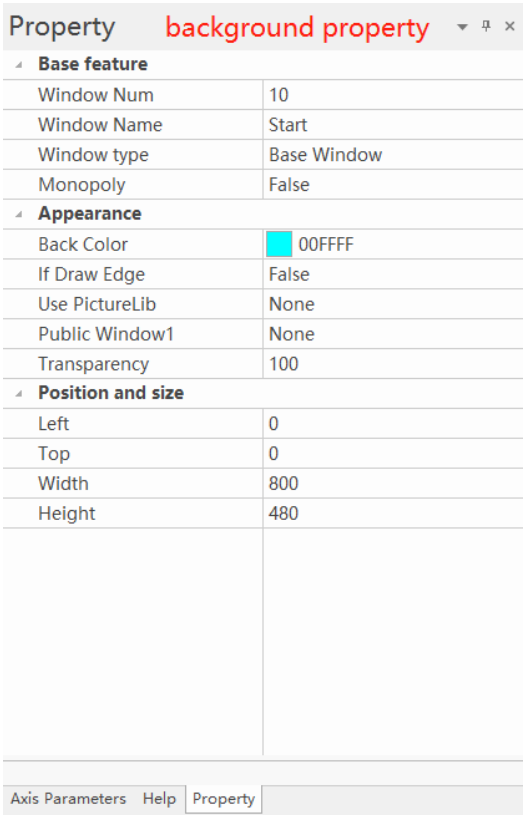
A. in the soft left HmiView, it can check all windows and components of this HMI. In the window name, right click menu "window property", then you can edit window information. And double click component name can open component property window quickly.



B. open HMI file, in the HMI window interface, click edited component, then in the software right, the component property window will pop up, like below left image, then in the HMI window,

click empty place (in the HMI interface), corresponding HMI property window can be opened. And click empty place (out of the HMI interface), HMI system configuration window will be shown.





Above property specific parameters, please refer to RTHmi manual.

--how to build one new HMI file to run--

a. build one HMI file in the project, add autorun No.

b. open “Hmi system setting” window, set screen position and size (resolution), starting window, etc.

c. edit HMI file, build the window, add components, open corresponding “property” window to set window and component functions.

d. build new Basic file, and edit Basic sub function that is called by HMI component.

e. in “Hmi system setting”, add init sub and period sub, then add subfunction name at the component action of Basic program to be called.

f. connect to controller or simulator, download the program. If no controller, it can connect to xplc screen to do simulation.

1.12. Help Documents

- RTBasic Programming Manual: https://www.zmotionglobal.com/pro_info_290.html
- RTHmi Programming Manual: https://www.zmotionglobal.com/pro_info_291.html
- RTPlc Programming Manual: https://www.zmotionglobal.com/pro_info_292.html
- RTVision Programming Manual: https://www.zmotionglobal.com/pro_info_289.html
- Product Model Selection: https://www.zmotionglobal.com/pro_info_287.html
- Others: PC programming manual + routines + quick start: Zmotion Download Page
- Learning Articles: Zmotion Support Page

Chapter II Operations of Controller

RTSys can connect to Zmotion motion controller through EtherNET / RS232 / RS485 / Local interfaces. After communication, in RTSys, you can read controller related information and can set controller, such as, read controller state, modify IP address, update firmware, etc.

--how to operate quickly--

Name	Image Mark	Description
Controller		
Connect		Connect to controller / simulator
Disconnect		Disconnect to controller / simulator
Download RAM		Download project into controller / simulator's RAM, don't save when powered on.
Download ROM		Download project into controller / simulator's ROM, it will be saved when powered on.
State the controller		Check controller state information: controller basic information, ZCan node information, slot node state, communication configuration, etc.
Firmware controller		Update controller firmware version.
System Time		Check controller current time, and support custom controller time or synchronize with PC time.
Modify IP address		Modify controller IP address, also can check current controller IP address.
Compare Project		Compare current PC project file with controller file, whether they are consistent.
Lock Controller		Lock the controller through password, when locked, host computer program can't be downloaded into controller.
Unlock Controller		Unlock the locked controller, enter the correct password to unlock it.
Reset the controller		Restart the controller, then it needs to connect to RTSys manually after powered on.
Project		

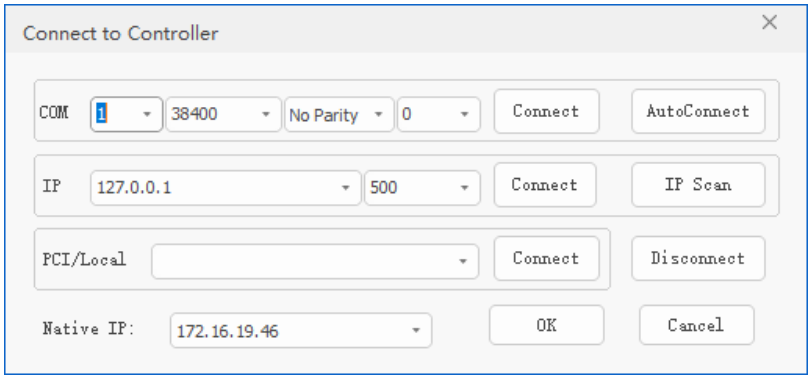
Compile All		Compile all files under the project, but don't download into controller.
Add to project		Add the file into current project, support adding program file, font file, image, etc.
Settings		Reserved
Make Zarfile		Generate specified ZAR encryption file, it can be encrypted by password or controller ID binding, the file suffix should be .zar.
Down Zarfile		Download ZAR encryption file into controller ROM.
Notes		Note the register of project file.
Indicator		ON / OFF ALM led of connected controller.

2.1. Connection

2.1.1. Connect RTSys to Controller

Click "Controller" – "Connect" – "Controller" to connect with controller.

RTSys supports connecting by serial ports / ethernet / PCI / Local.



Serial port parameters: COM (serial port No.), 38400 (Baud Rate), No Parity, 0 (serial port ID, usually is 0).

IP: controller IP address, the connection waiting time.

PCI/Local: PCI card No. / connect to MotionRT.

(1) Connect by Serial Port:

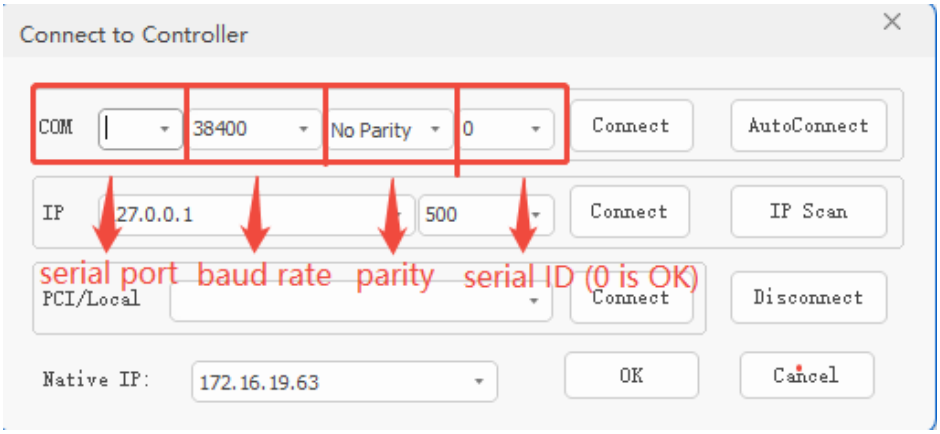
Step 1: use one serial port cable to connect with PC (please note their serial port parameters should be consistent, controller serial port default parameters: *baud rate 38400, data bit: 8, stop bit: 1, no parity*).

Step 2: in RTSys, open the "Connect to Controller" window (above), corresponding valid serial port of the PC are shown in first row "COM", select needed serial port No., and parameters, set baud rate, parity bit, etc.

Step 3: after that, click "Connect" to do connection.

Step 4: check the result, when connected, it will show success information in the "output & command" window that is at the bottom of RTSys interface.

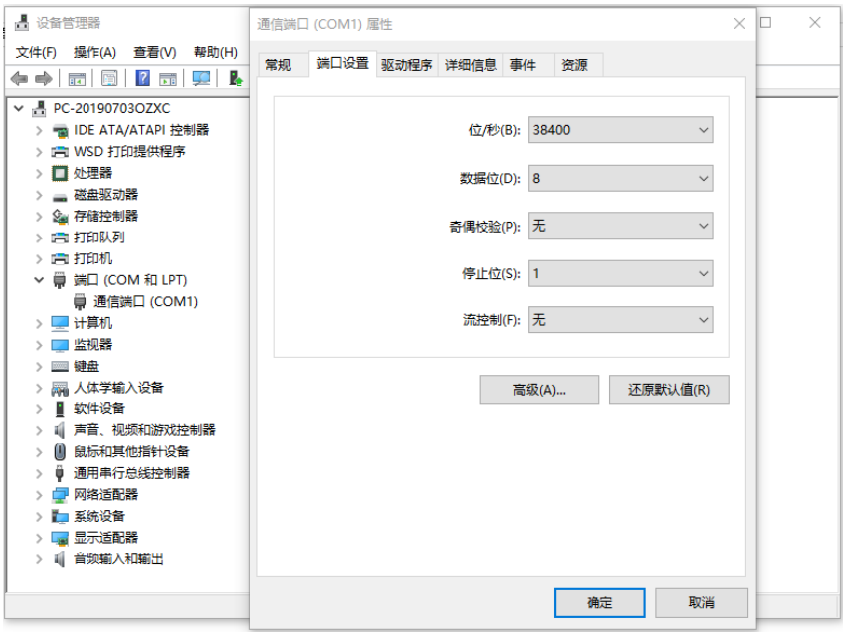
If you use USB to connect, virtual serial port No. will be generated automatically, then select serial port No. to connect.



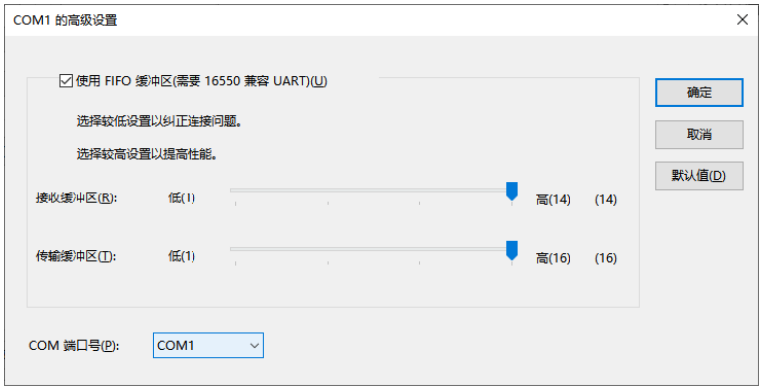
If connection failed, please check according to below steps:

- Check serial port connecting cable, whether it is a cross cable.
- Check serial port parameters (COM No., Baud Rate, etc.)

--open PC "device manager" – "port" – "COM" – "COM Setting", check it, controller serial port default parameters are Baud rate 38400, data bit 8, stop bit 1, no parity--



--in “COM settings” – “advanced”, you can change the COM port No.--



- While connecting to controller through serial port, corresponding controller serial port must be configured as MODBUS slave station protocol mode (default mode), and when power-off and restart it, the mode will be resumed.
- See whether COM port is used by other programs, like, serial port debugging, etc.
- Check whether there is enough serial port hardware on PC side
- Change the serial port cable / test PC.

(2) Connect by Ethernet:

When select IP in IP address list, it will find available controller IP address in current local ethernet range.

The controller default IP address is 192.168.0.11, and in “Connect to Controller” window, local IP address can be shown. Please note “PC IP and controller IP should be in the same network segment,

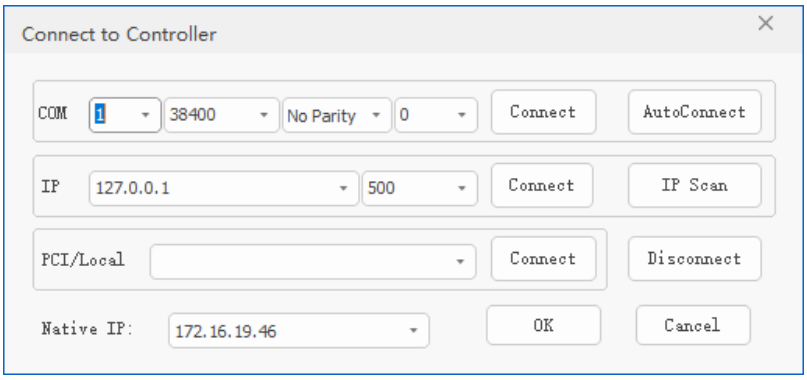
that is, front three segments are the same totally, only the last one is different, so PC IP should be 192.168.0.xx".

For some controllers, there are 2 ethernet ports. For example, VPLC5XX controllers, LAN 1 and LAN 2. (LAN 1 default IP: 192.168.0.11, LAN 2 default IP: 192.168.1.11)

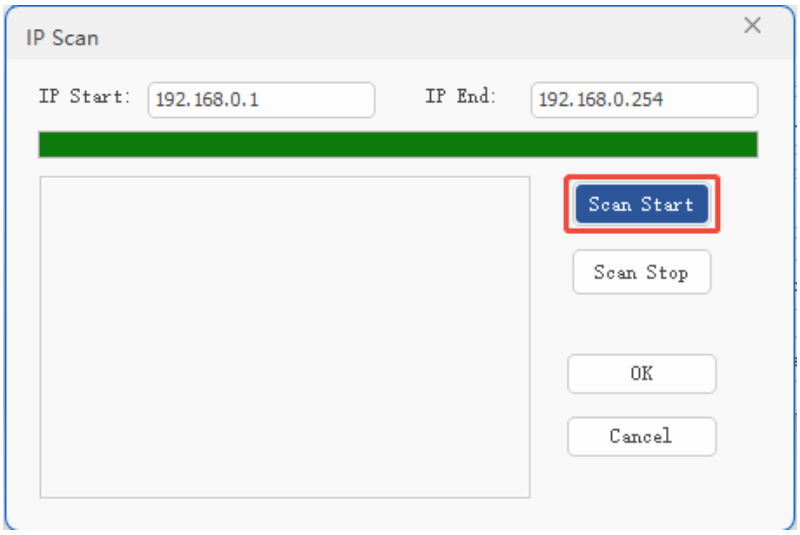
--how to check local IP quickly--

Please note to set wired ethernet IP and no-wired ethernet IP.

In "Connect to Controller" window, easy to see local IP.



When there are several controllers on the same network, and in IP list, no more that can be found, use "IP Scan" to check others.

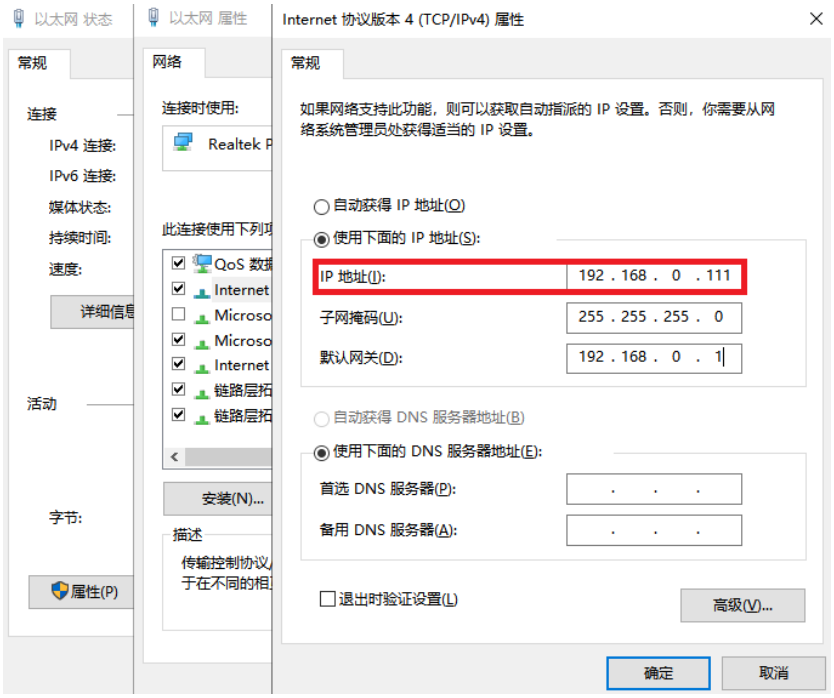


--how to modify IP--

When PC IP and controller IP can't match, we can modify one of them.

- **Modify PC IP**

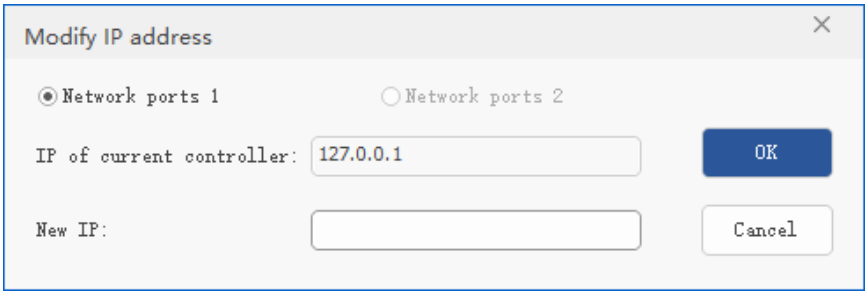
Check whether your PC IP protocol version 4 is "192.168.0.xxx", front 3 segments should be the same as controller's, and the last one can't be the same, because the controller default IP is 192.168.0.11. If the PC IP segment 3 is also different, please modify corresponding subnet mask as 0. After that, do connection.



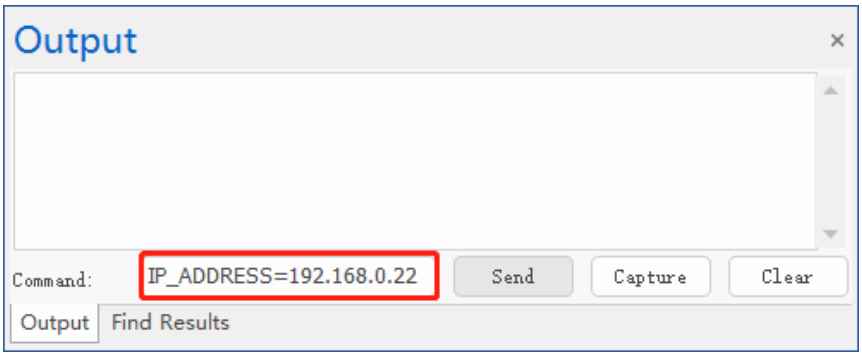
- **Modify Controller IP**

At first, connect to controller through serial port, then get controller IP, then you can modify it according to below methods.

Method 1: click “controller” – “modify IP”, then corresponding “modify IP” window will appear, usually ethernet port 1 IP is shown default, you also can switch it to see 2 IP. Then in this window, you can enter new IP directly, then click OK. After that, do connection again through this new IP. Attention port 1 and port 2 can’t be the same address.



Method 2: click “controller” – “controller state” to check IP address, or send the command to get IP, then use IP_ADDRESS to send new IP to modify directly.



NOTE: after modification, connection between controller and RTSys will break, then please select new IP to connect again.

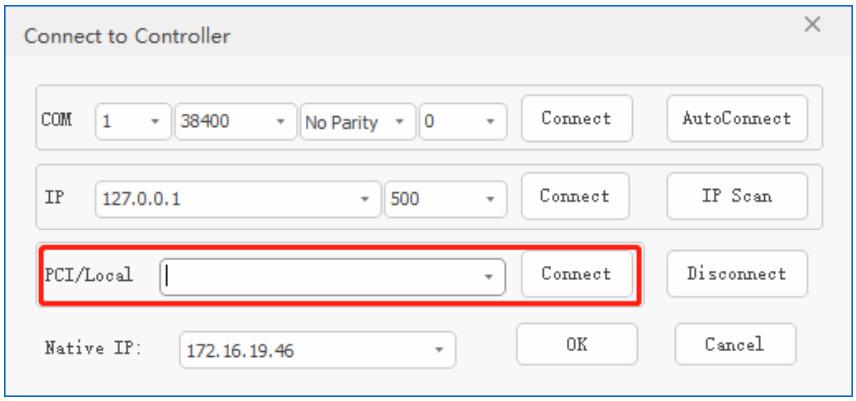
(3) Connect by PCI

This connection is only for Zmotion PCI series motion control cards. It scans PCI motion control card No. inserted in IPC, then find corresponding card No. to do connection. But before that, please install the drive according to card user manual.

(4) Connect by LOCAL

This connection is mainly used for RTSys and MotionRT to do real-time communication. Control cards can be XPCI / XPCIE serial cards. At first, insert the card into PC / IPC card slot, then install MotionRT drive according to user manual, next open MotionRT software to do some configurations, then click "start", at this time, in RTSys – Local, MotionRT can be scanned. Select related MotionRT to connect. And one MotionRT sequence No. corresponds to one motion control card.

Whether the connection is successful or not, check "output" window.

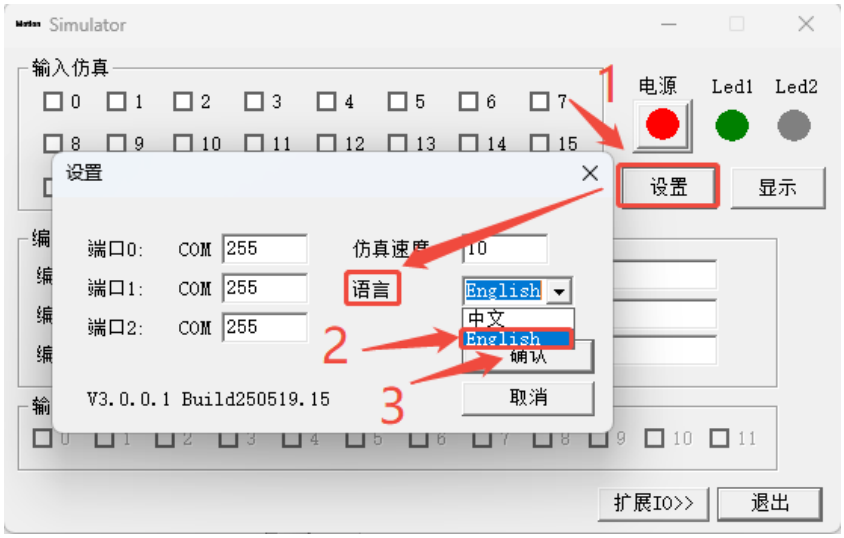


2.1.2. Connect RTSys to Simulator

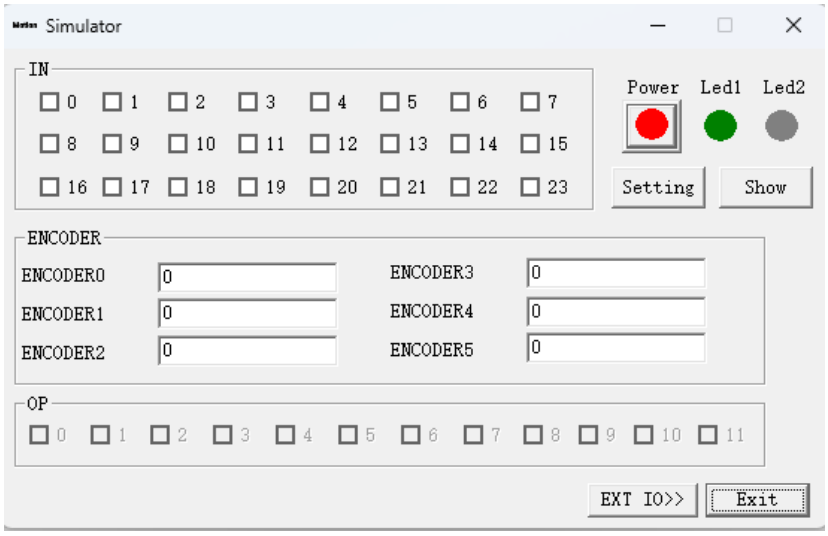
RTSys supports offline simulation, which means simulate and test in advance when there is no controller.

Click "Controller" – "Simulator", then it will connect to simulator automatically and open it directly. And there will be indication in the "output" command. Please note don't close the simulator window, you can click "-", not click "x".

And it can be seen there are many functions, IO simulation, encoder simulation, HMI functions, etc., and **you could switch it into English simulator**: click 设置 – 语言, then pull down the list, choose English, click "确认", then it will pop up one window that tells you to restart RTSys for making it valid, so please restart it manually.



Now, it can be seen it becomes English Simulator:



--IN--

When there is no external input device, in RTSys simulator connection, you can operate the IN simulation, specifically, check corresponding port to open it, for example, check 0 to open IN 0, check 1 to open IN 1.

--OP--

It is used to watch whether the output port outputs the signal.

--ENCODER--

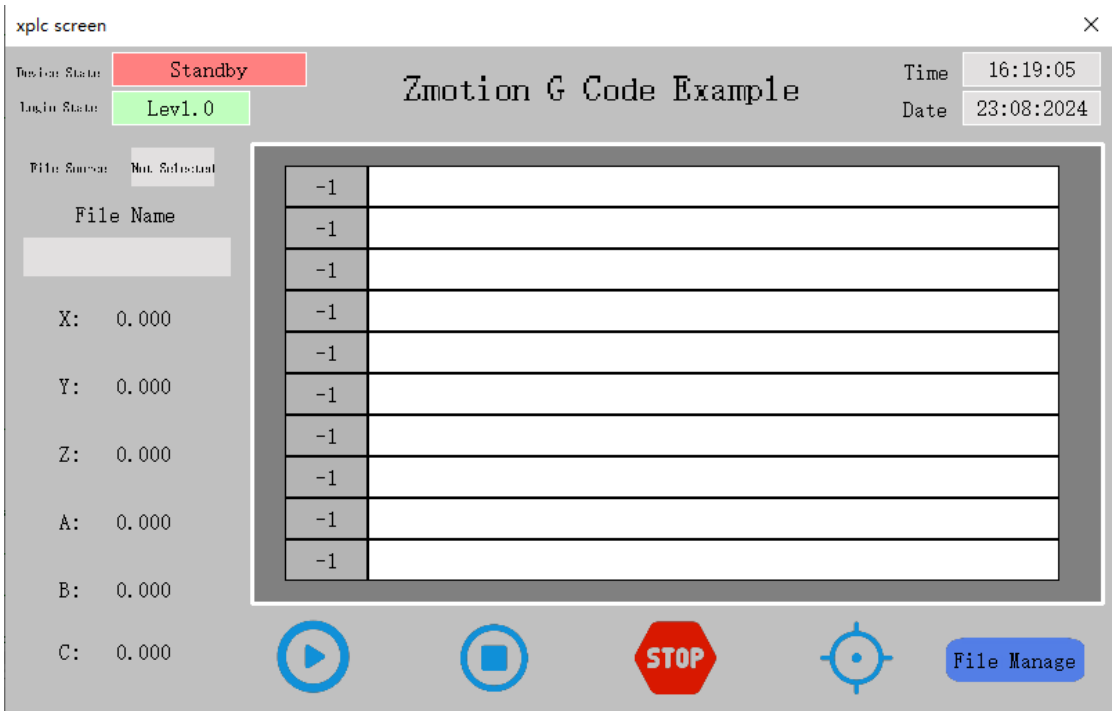
It is used to input corresponding value for ENCODER, while using, please set ATYPE as the type that is with encoder.

--Show--

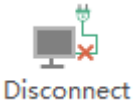
When your project includes HMI, click it to simulate HMI interface.

NOTE: when IO is not enough, click "EXT IO" (expand IO) to simulate more IOs, 64 IO can be extended.

For example: one certain HMI simulation:



2.1.3. Disconnect



When RTSys is connected to controller or simulator, use “Disconnect” button to disconnect directly.

Specifically, there are 3 ways to do disconnection:

- When connect to controller, power-down controller will lead connection between controller and RTSys break. However, recommend disconnect at first, then power-down the controller.
- When connect to simulator, close the simulator window can disconnect, but not recommended.
- When connect to controller or simulator, close RTSys software, then the connection also breaks directly.

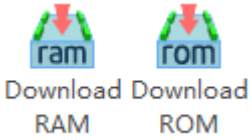
2.2. Download Program

Download means download the edited program in the RTSys to controller or simulator, then do operation. If it only compiles the program, can't download into controller, and can't run normally.

Notes:

- It must build the project at first.
- It must do connection (controller / simulator)
- One of one file must set Auto Run Task No.

2.2.1. Download into RAM / ROM



--Download RAM--

Download project into controller RAM, when downloaded, it will run immediately, but current downloaded project **can't be saved** when powered-off.

--Download ROM--

Download project into controller FLASH, when downloaded, it will run immediately, and current downloaded project **can be saved** when powered-off.

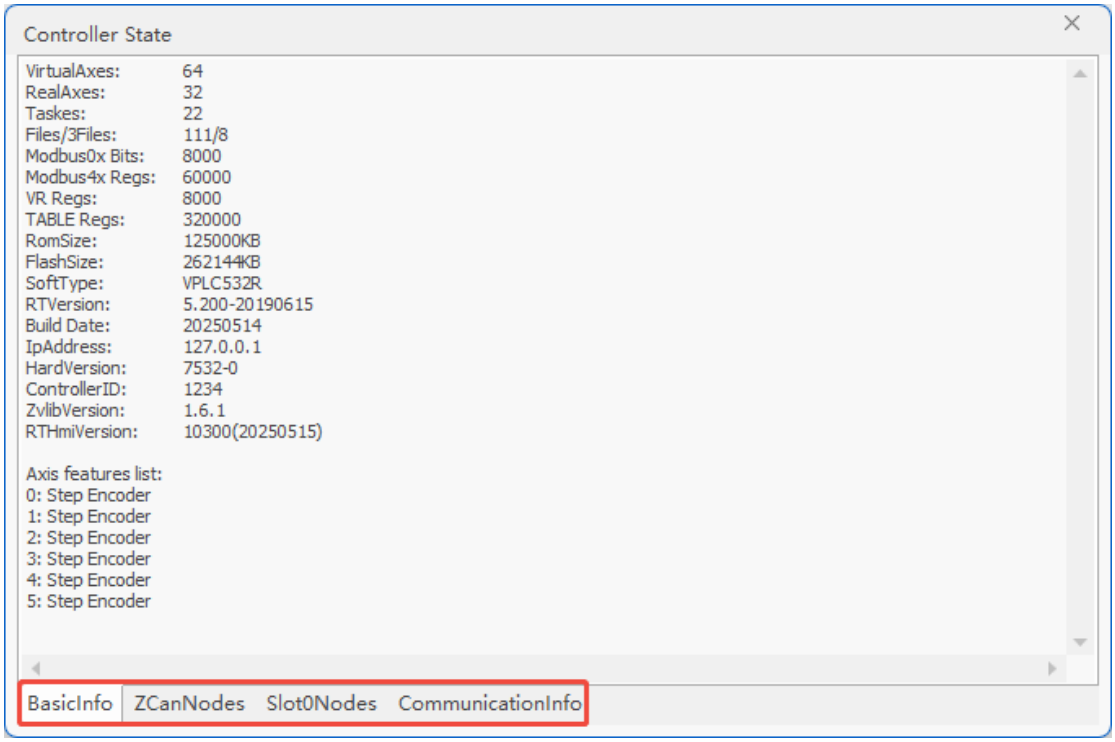
--NOTES--

- When program appears "error", it can't be downloaded successfully, please check whether controller model supports, for example, ZMC0XX don't support "Download RAM".
- When program appears "warn", it can't be downloaded successfully, please check whether the auto run No. is set for the program file.
- If the file is large, recommend you compile it at first, then download it. That is, click "Controller" – "Compile All", then Download, in this way, speed is faster.

2.3. Controller State

After connection, current controller state and information can be checked through ***"Controller"*** – ***"State the controller".***

"controller state" is mainly to view controller information, which includes controller basic information, ZCan node state, slot node state, communication configuration.



It can be seen there are 4 parts in total:

BasicInfo: show current connected controller's basic configuration parameters

ZCanNodes: show information of connected CAN expansion module

Slot0Nodes: show information of EtherCAT and CAN

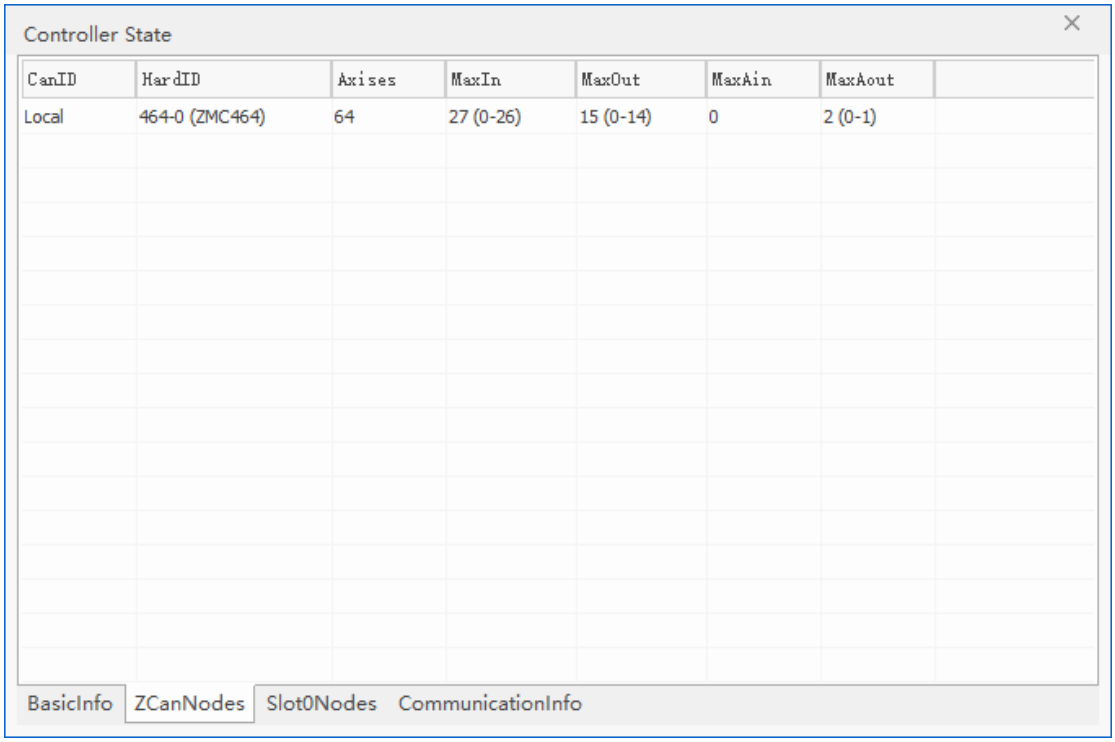
CommunicationInfo: show CAN communication related parameters

2.3.1. Basic Information

Parameter	Description
VirtualAxises	Max virtual axes
RealAxises	Max real motor axes
Taskes	Max tasks

Files/3Files	Max files / max .Z3P file
Modbus0x Bits	Valid MODBUS bit register space
Modbus4x Regs	Valid MODBUS word register space
VR Regs	Valid VR register space
TABLE Regs	Valid TABLE array space
RomSize	ROM size
FlashSize	Flash size
SoftType	Software type
RTVersion	System software version + firmware version
Build Date	Firmware version generated time
IpAddress	Controller IP address
HardVersion	Hardware Version
ControllerID	Controller unique ID
ZvlibVersion	Vision library file version
Axis features list	Axis type list

2.3.2. ZCan Node Information



The CAN ID shows local, and it shows the hardware is ZMC464 controller, followings are it's local

resources, it has 64 axes, 27 digital inputs, 15 digital outputs, no analog input (MaxAin), and 2 analog outputs.

If there are more CAN devices, it will show in behind line.

2.3.3. Slot 0 Node Information

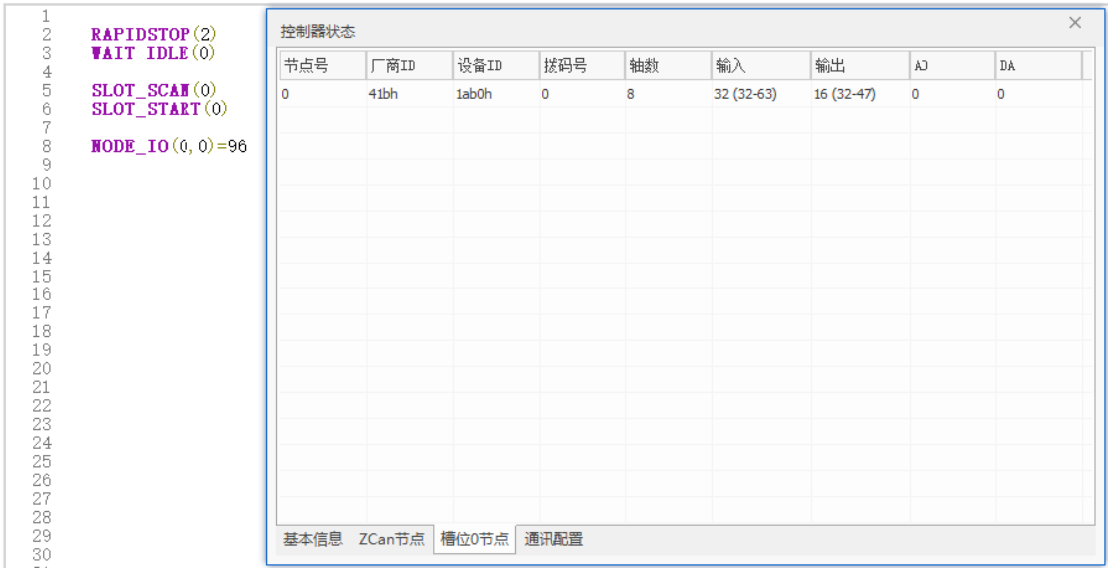
For single-bus, then the slot No. will be 0.

- If there is only EtherCAT, slot 0 is EtherCAT.
- If there is only RTEX, slot 0 is RTEX.

For dual-bus, slot 0 is EtherCAT, slot 1 is RTEX.

- If there are EtherCAT and RTEX bus, slot 0 will be EtherCAT, then slot 1 will be RTEX.

For Example:



It can be seen there are 2 parts:

- Left is one very basic EtherCAT initialization program. Actually Zmotion provides one general EtherCAT initialization .bas file (Bus Init), you only need to set corresponding variables according to real situation.
- Right shows EtherCAT devices' information from row 1 to row N.

Node 0, and the hardware manufacturer ID is 41bh (Zmotion Technology ID), hardware manufacturer device ID is 1ab0h (Zmotion EIO24088), the DIP code is 0, 8 means 8 expanded axes, 32 digital inputs and 16 digital outputs, no analog input and analog output.

2.3.4. Controller Communication Configuration

--CAN information & RS232/RS485/RS422 serial port information--

See below image, it can be known:

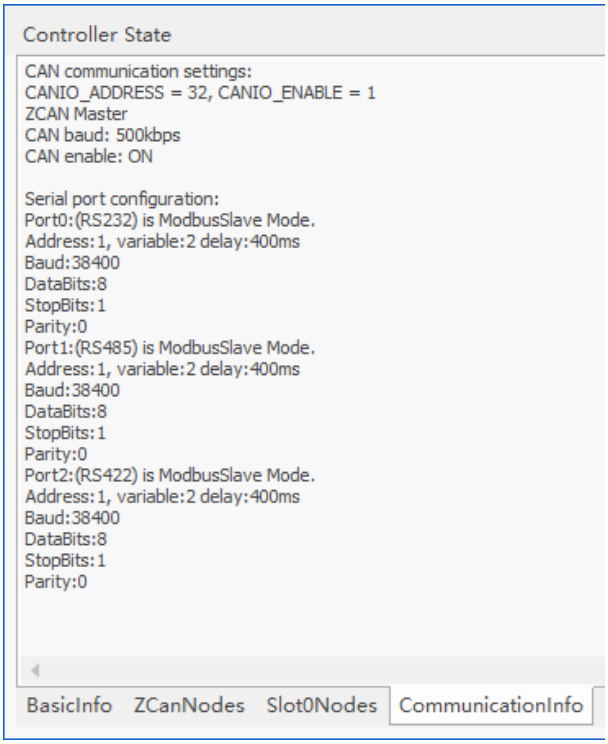
Now CAN communication configuration: CANIO_ADDRESS = 32, CANIO_ENABLE = 1, then it can be known the controller is ZCAN Master main station module.

In addition, CAN bus communication speed ratio is 500kbps, then CAN is in enable state. If you need to modify CAN communication, modify CANIO_ADDRESS and CANIO_ENABLE parameters.

Port 0 is RS232, state is ModbusSlave, address is 1, VR and MODBUS registers are two independent areas.

Port 1 is RS232, state is ModbusSlave, address is 1, VR and MODBUS registers are two independent areas.

More details, please refer to SETCOM parameter.



2.4. Upgrade Firmware

When current firmware version can't meet program operation, or some commands are not supported, please update the firmware.

There are 2 methods: update in RTSys directly, or use zfirmdown tool to download zfm firmware package, then do updating.

Let's update in RTSys:

Step 1: download the new version firmware, and save it in your PC

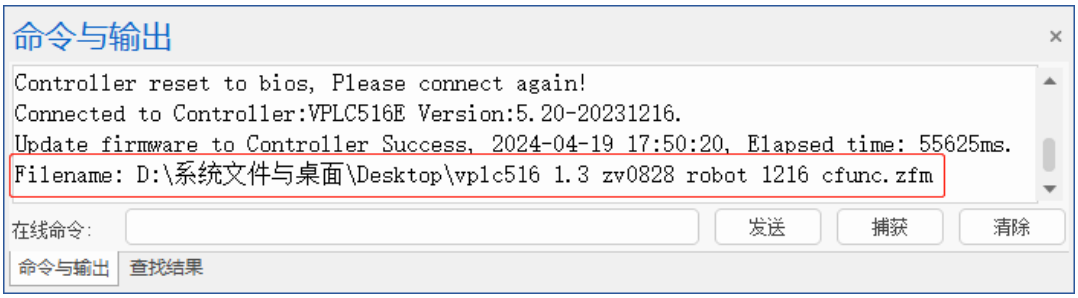
Step 2: open RTSys, connect to controller, and click "controller" – "Firmware controller", then one window appears, current controller firmware version can be checked, then you can check the version.

Step 3: click "Browse", then open the file you saved of step 1.

Step 4: then click "update". After that, it will open one window that indicates the controller needs to be restarted to ZBIOS, then, please click OK to do reconnection (note: updated firmware version should be consistent with controller hardware model, otherwise, it will report errors).

Step 5: after connect again, "update firmware" window will pop up again, now, system enters ZBIOS state, the current model will show as "VPLC516E-ZBIOS", please click "Update" again. Then, it will download, don't close it while downloading.

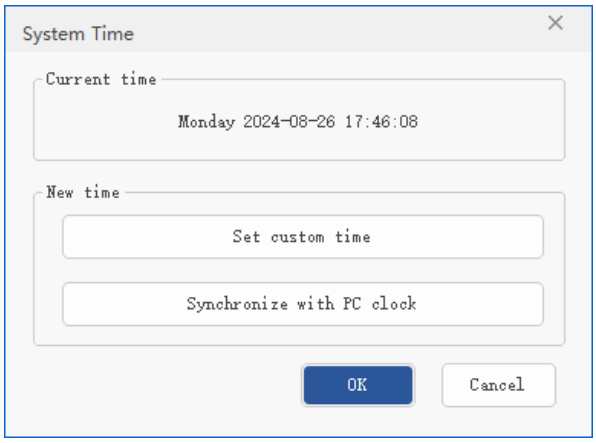
Step 6: when downloaded, "update firmware" interface will disappear, and in "output", success information will be shown.



Step 7: connect to controller again, and check controller state to check firmware.

2.5. System Time

"System clock" is inside the controller, which can be used to check current controller time. What's more, you can set custom time or synchronize with PC time.



--set custom time--

You can modify controller time as you needed, it includes day, year, month, data, hour, minute, second.

--synchronize with PC clock--

Synchronize with your PC time automatically.

2.6. Modify IP Address

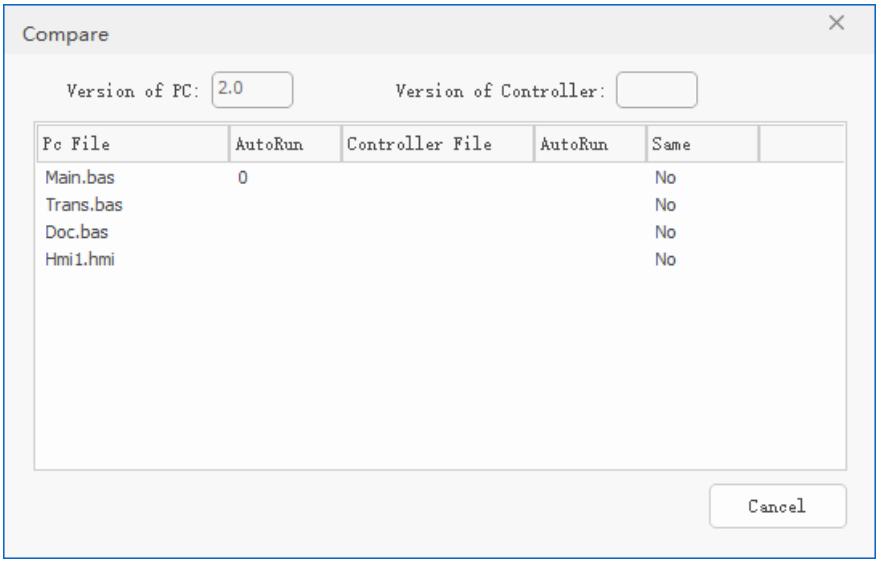
Please refer to above 2.1.1 Connection – how to modify IP address.

2.7. Compare Project

"Compare Project" function is used to compare current project program with controller program, and check whether they are consistent.

There will show "PC files", and corresponding autorun No., then controller files, also corresponding autorun No., the last will tell you whether they are same or not. Yes means same, No means not same.

Note: for encryption, controller doesn't support unloading the program.

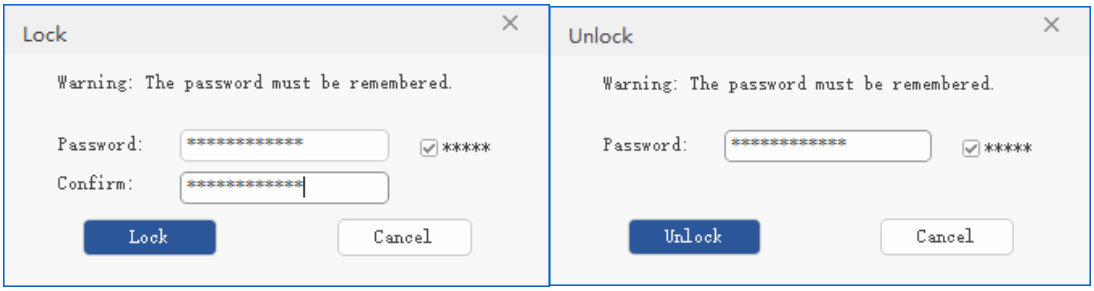


2.8. Lock / Unlock Controller

"Lock Controller", you can set the password to lock the controller, in this way, PC program can't be downloaded into controller, controller program can be protected well. But please note generated ZAR file also can be downloaded. Correspondingly, enter correct password can unlock it.

- When controller is locked, no way to do debugging.
- Password can be digit, letter, special signals, but the length can't exceed 16 characters.
- When the password is forgotten, no way to unlock, please remember well.

Click "controller" – lock controller / unlock controller.



2.9. Reset the Controller

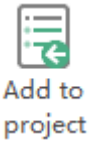
"reset the controller" is used to restart the controller. After connection, click "controller" – "reset the controller", then open below window, click OK. The controller will power on again, please do connection manually again.

2.10. Project Operations

2.10.1. Compile All

Reserved

2.10.2. Add to Project

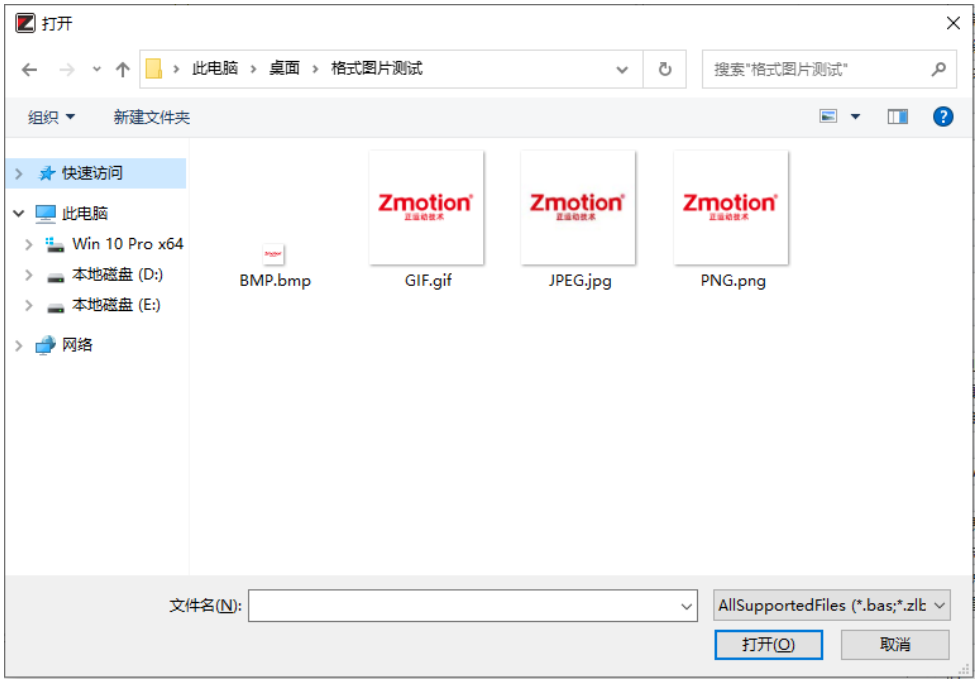


This function is used to add file under current path or other paths into now project, files can be program files (.bas / .plc / .hmi), font files (.zft / .ttf), graphic files (.bmp / .png / .jpg / .gif).

Actually there are 2 methods to achieve it:

➢ **"Add to Project" by Menu**

Click menu "controller" – "add to project", then below window will pop up, then please find needed file path, and select target file, open it. After that, corresponding file can be added, and which can be checked in the project view.



➢ **"Add to Project" in Project View**

In RTSys left ProjectView, right click, then click "Add to Project", followings are the same as above.

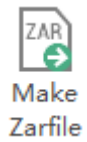
Note: usually, "ProjectView" is opened default, if there is no this window, please click "View" – "Project" to open it.

2.10.3. Settings

In this function, it can check current project path and version, also can set the compile platform for this project. Generally, it can be used in C language compiling, and it supports customizing gcc compile items, but different controllers are with different compiling platforms. For details, please contact us.

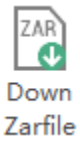
Note: the breakpoint only can pause one single task. After adding, it can select whether to stop the single task or the whole project.

2.10.4. Generate ZAR File



It can compile current project and generate it as specified ZAR encryption file, in this way, it can achieve independent downloading and protect the program. After that, no codes can be seen, but it still can download the ZAR file into controller, then run it.

2.10.5. Download ZAR File



This can download generated ZAR file into controller, it needs entering correct password or bound controller ID.

More, please refer to Chapter XI How to Download ZAR File.

2.10.6. Notes

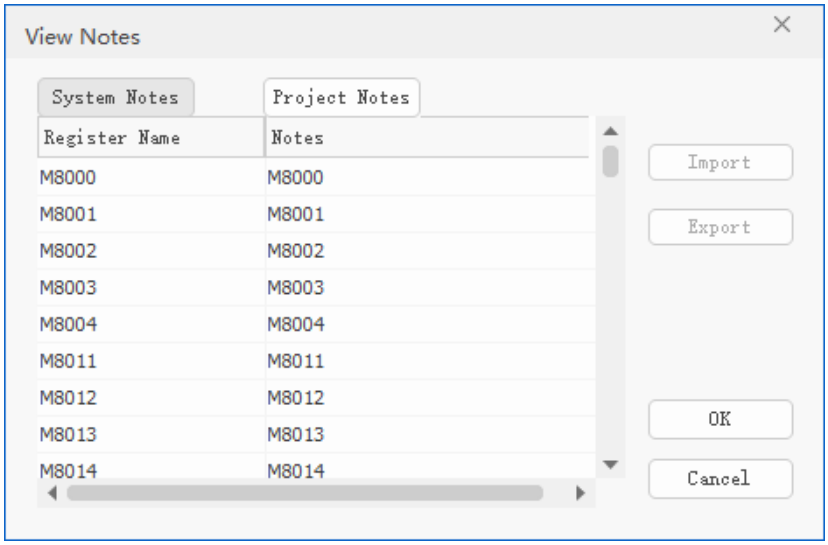
This function is mainly to do noting for register, including usage of system register, project register.

--System Note--: check the usage of register that has been noted by system.

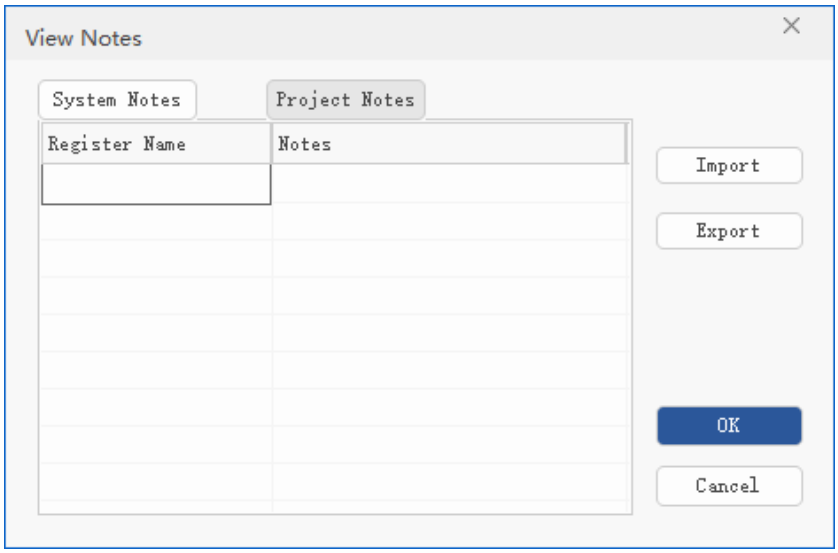
--Project Note--: add the register and corresponding note information by yourself.

Operation Steps:

(1) Click menu “controller” – “notes” to open below window, then it can be seen system has noted how to use registers.



(2) Here, you can switch into “project notes”, at this time, you can custom the register and notes, please remember to click OK to save it.



(3) If you need to save customized note as others, please click Export, and set file name, select file path, click SAVE. Then, .ini file will be generated.

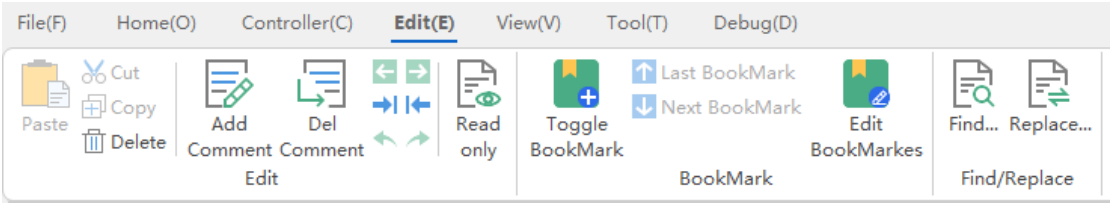
(4) If you import notes into current project, click Import, and find .ini file to open it.

2.11. Indicator

In RTSys, you can open or close ALM led connected on the controller directly. This function is mainly to find needed controller when there are several controllers connected.

Chapter III Operations of Editing

RTSys supports program editing and program debugging.



--how to operate quickly--

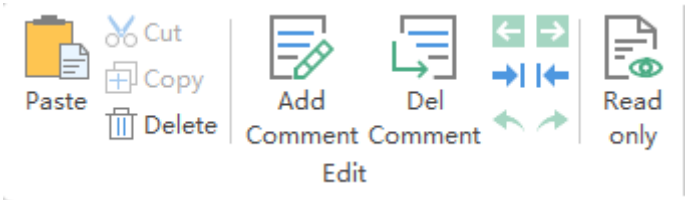
Name	Image Mark	Description
Edit		
Paste		Paste clipboard's content into project file.
Cut		Cut project file's selected program content / elements to clipboard temporarily.
Copy		Cut project file's selected program content to clipboard temporarily.
Delete		Deleted selected content of the file.
Add Comment		The whole line that is selected can be noted.
Del Comment		Delete notes of selected line.
Insert one Tab		Add one tab for the line where the cursor is.
Delete one Tab		Delete one tab for the line where the cursor is.
Go to Last Position		Jump to last position
Go to Next Position		Jump to next position
Undo		Undo last operation
Do Undo again		Restore undo operation.
Read-Only		ON / OFF read-only mode, please note it is valid in basic file and plc file.
BookMark		

Toggle Bookmark		Set / delete the bookmark for selected line in file.
Last BookMark		Jump to last bookmark of the same project
Next BookMark		Jump to next bookmark of the same project
Edit BookMarks		Check file and line No. of the bookmark that was set, and it can edit the bookmark.
Find / Replace		
Find		Find needed content according to entered keyword (the range can be customized)
Replace		Replace content according to entered keyword (the range can be customized)

3.1. Commonly Used Editing

There are several editing functions that are usually used for you, including copy, paste, cut, delete, add / remove a Tab, etc.

Following functions only can be used in menu “Edit” – “Edit”



(1) Cut / Copy / Paste / Delete:

Used to edit Basic/Plc/Hmi program or component. They can be achieved by menu “Edit” or right-click.

(2) Add / Remove a Tab:

It is mainly for Basic program to add or delete one Tab. If you need Tab several time, please operate several times, and the length is fixed, no way to change it. also, you can press “Tab” button of the keyboard directly.

(3) Go to Last Position / Next Position:

Jump former or next position for Basic/Plc/Hmi. And support jumping in the bookmark, in the result.

(4) Undo / Restore Undo:

If you want to back to former operation, please click undo, if you don't want to undo again, please click corresponding button.

(5) Read-Only:

It is mainly for Basic / Plc file, click it to open or close "read-only" mode. First click, open it, click again, close it. When it is opened, no way to edit the program. Therefore, please close it when you need to do editing.

3.2. Add / Delete Comment

Add program content as comment content or delete program comment content. After adding comments to a line/multiple lines of a program, the commented content will no longer run, and after that, the content will become green. When deleted, it will run again.

Operation Method:

1. You can open it in the menu bar "Edit" → "Add Comment"/"Delete Comment".

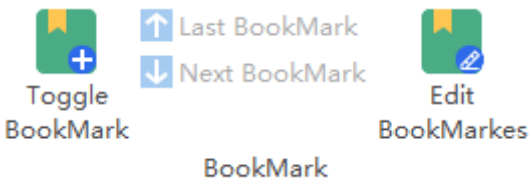
2. Select the target program line content, right-click "Add Comment"/"Delete Comment".

3. Add an English single quotation mark (') before the content to be commented on to comment on the content.

```
base(gAxisX)
ATYPE=1
units=1000
ACCEL=1000
DECEL=1000
SRAMP=30
SPEED=100
DATUM_IN=-1
FWD_IN=-1
REV_IN=-1
ALM_IN=-1
FS_LIMIT=2000
RS_LIMIT=-2000
```

```
base(gAxisX)
'ATYPE=1
units=1000
ACCEL=1000
DECEL=1000
SRAMP=30
SPEED=100
DATUM_IN=-1
FWD_IN=-1
REV_IN=-1
ALM_IN=-1
FS_LIMIT=2000
RS_LIMIT=-2000
```

3.3. Bookmark



It can be seen it includes above functions.

(1) Toggle BookMark

Used to add the bookmark for Basic / PLC program, then you can find one certain line program quickly. For the Basic program line that has set the bookmark, it will show one green line. For PLC program line that has set the bookmark, one M mark will be shown. And now only support adding one single line once.

(2) Last BookMark / Next BookMark

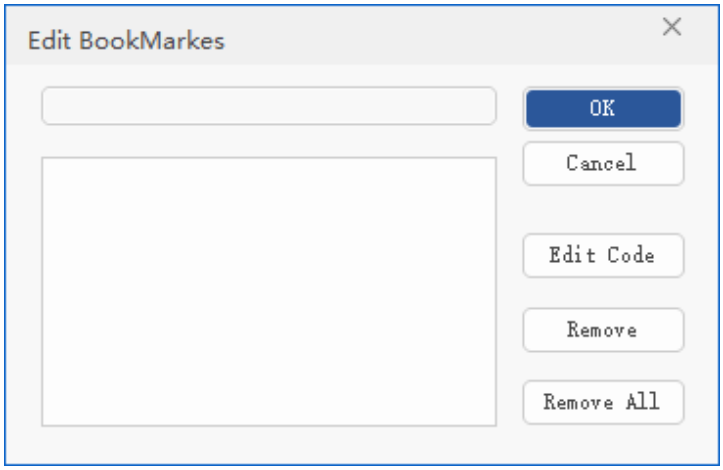
It is used to jump between programs that have bookmarks.

You can achieve it through menu “edit” – “toggle bookmark”, or you can select the program line at first, then right click popped window to set.

(3) Edit BookMark

Used to check which one program sets the bookmark, and it will show exact position.

Operation Method:



“jump the bookmark”: click one certain line’s bookmark to show this bookmark’s saving path, then click “edit code” to jump to target path. Double click one certain line bookmark, it will go to target bookmark position directly in current project.

"remove the bookmark": select one certain line's bookmark, then click "remove" – "ok. If you need to cancel all, please click "remove all" – "ok".

3.4. Find / Replace

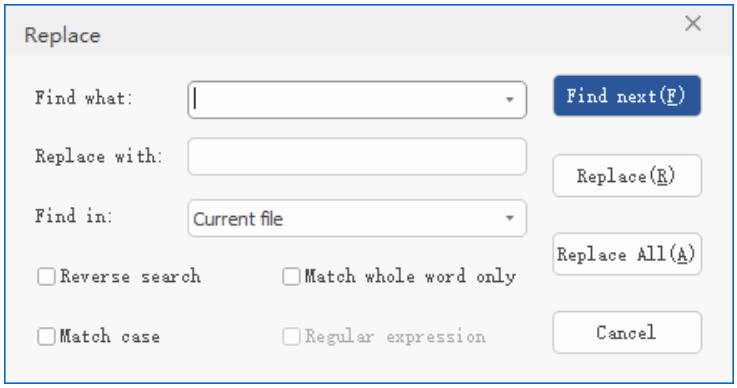
- **Find**

Search for target content in the current project/file, so that you can quickly locate the target content. It can select the range, find reversely, etc., at the same time, it can support finding / replacing registers in PLC, like, X/Y/M/T/C/S (for example, replacing X0 with M10 or replacing the command LD M0 with LDI M0).

It can be seen there is one "Print all", it is used to print found content in "Find Result" window. And specific file name, line No., and line content all can be shown. For "find result", it can be opened in menu "view" – "Find result".

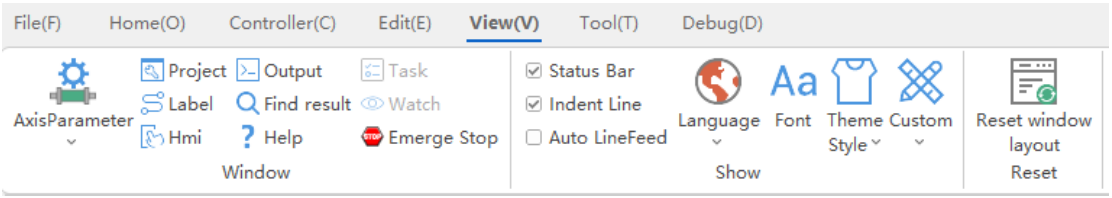
- **Replace**

Replace the searched target content with new content. It is convenient to quickly replace the same content in the file with new content. RTSys supports selecting the replacement range, replacing a single target or replacing all.



- Replace: replace one single content. After entering “find what” and “replace with”, click “Replace” once, then it will check needed information in the order of “up to down”, then the target content will be shadowed. If no need to replace this one, click “find next”, if you need to replace it, click “replace” again.
- Replace All: this will replace all needed information directly.
- Find in: here, you can select the range, including current file, selected content, all files, current project.
- Reverse search: when it is checked, content will be found starting from down to up.
- Match whole word only: it will find information that is 100% matched with needed content. For example, when you need UNITS, only UNITS will be found, UN, UNIT are wrong.
- Match case: whether to search for the content based on case. If this function is checked, the search will only be based on the uppercase/lowercase characters entered in the search content. Example: when the search content is entered as "units", if this function is checked, only the lowercase "units" will be matched, and the uppercase will not be searched.
- Regular expression: reserved.

Chapter IV “View” Window



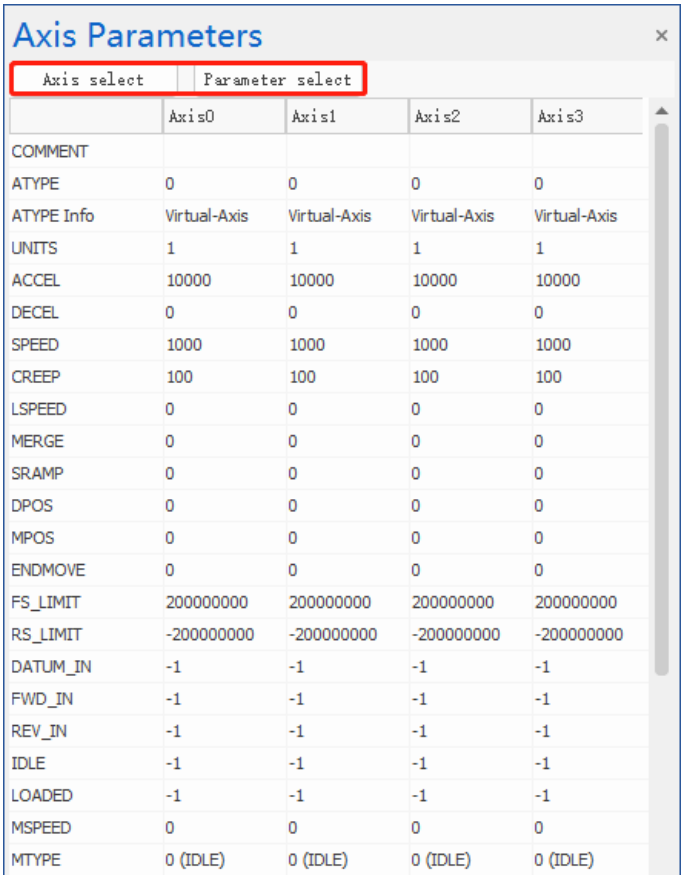
--how to operate quickly--

Name	Image Mark	Description
Window		
Axis Parameter		ON / OFF “axis parameter” window, it can check commonly-used parameters in motion control.
Project		ON / OFF “project view” window, it can check how many files, file types, and auto run task No. in the current file. And support axis configuration, EtherCAT node configuration, etc.
Label		ON / OFF “label view” window, then it can check all SUB functions defined in basic file.
Hmi		ON / OFF “Hmi view” window, then it can check window information and component information included in Hmi file.
Output		ON / IFF “Find result” window, used to show results.
Find Result		ON / OFF “output” window, it can check content, print running result, online input command, etc.
Help		ON / OFF “help” window, used to show help documents.
Task		ON / OFF “Task” window, it will show when debugging. It can check each task’s details.
Watch		ON / OFF “Watch” window, it will show when debugging. It can view variables, registers.
Emergency Stop		Stop all tasks immediately.
Show		
Language		Change RTSys showing language, there are Chinese

		and English, after choosing, please restart it.
Encode		Convert the encoding method.
Font		Set program file's font format, size.
Theme Style		Set RTSys software showing type (there are 4 styles).
Custom		Set window custom formats (there are 4 windows).
Reset		
Reset window layout		Reset software window layout, resume as default, please restart it.

4.1. Axis Parameter Window

Axis Parameter window is used to watch commonly used parameters in motion control, and real-time change can be watched. This window is shown in the right side of the RTSys software. And it can be opened or closed by menu "View" – "Axis Parameter". What's more, some parameters can be modified directly by double-click. But for only-read parameters, no way to change them here.



	Axis0	Axis1	Axis2	Axis3
COMMENT				
ATYPE	0	0	0	0
ATYPE Info	Virtual-Axis	Virtual-Axis	Virtual-Axis	Virtual-Axis
UNITS	1	1	1	1
ACCEL	10000	10000	10000	10000
DECEL	0	0	0	0
SPEED	1000	1000	1000	1000
CREEP	100	100	100	100
LSPEED	0	0	0	0
MERGE	0	0	0	0
SRAMP	0	0	0	0
DPOS	0	0	0	0
MPOS	0	0	0	0
ENDMOVE	0	0	0	0
FS_LIMIT	200000000	200000000	200000000	200000000
RS_LIMIT	-200000000	-200000000	-200000000	-200000000
DATUM_IN	-1	-1	-1	-1
FWD_IN	-1	-1	-1	-1
REV_IN	-1	-1	-1	-1
IDLE	-1	-1	-1	-1
LOADED	-1	-1	-1	-1
MSPEED	0	0	0	0
MTYPE	0 (IDLE)	0 (IDLE)	0 (IDLE)	0 (IDLE)

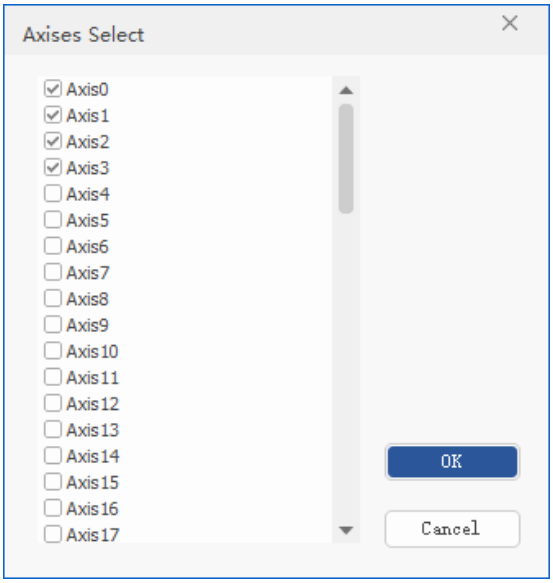
- **Axis Parameters:**

Parameter	Description
ATYPE	Axis type, 0-virtual axis, 1- pulse output, 3- orthogonal encoder input, 65-EtherCAT CSP mode
UNITS	Pulse amount, which indicates how many pulses to be sent in per unit (support 0.00001 precision)
ACCEL/DECEL	Axis acceleration/deceleration, the unit is units/s/s. When in multi-axis motion, acceleration of axis group interpolation is master axis merged vector acceleration/ master axis merged vector deceleration. When DECEL is not set, ACCEL = DECEL by default.
SPEED	Axis speed, the unit is units/s. When in multi-axis motion, it is main axis' merged vector speed in the axis group interpolation motion.
CREEP	The creep speed when axis homing, it is used for origin searching, the unit is units/s.
LSPEED	Axis starting speed, also it is the stopping speed, default is 0, and the unit is units/s. when in multi-axis motion, it is the merged vector starting speed of axis-group interpolation. If you need higher efficiency, also can set it as non-0, but don't too large.
MERGE	When it is ON, in the front and back buffer motions will be connected to together, that is, the motion is continuous, mainly used for continuous interpolation.
SRAMP	S curve setting when in acceleration and deceleration motion, the unit is ms. When in multi-axis motion, it is the time of axes' merged vector curve.
DPOS	Axis demand / target position, the unit is units.
MPOS	Axis measured position, the unit is units.
ENDMOVE	The end target absolute position of current motion, the unit is units.
FS_LIMIT	Axis forward soft position limit, the unit is units. If axis motion exceeds the limit, it will stop and report an error of "FSOFT".
RS_LIMIT	Axis reverse soft position limit, the unit is units. If axis motion exceeds the limit, it will stop and report an error of "RSOFT".
DATUM_IN	Input No. that corresponds to origin switch position, -1 means invalid.
FWD_IN/REV_IN	Input No. that corresponds to + / - hard position limit, -1 means

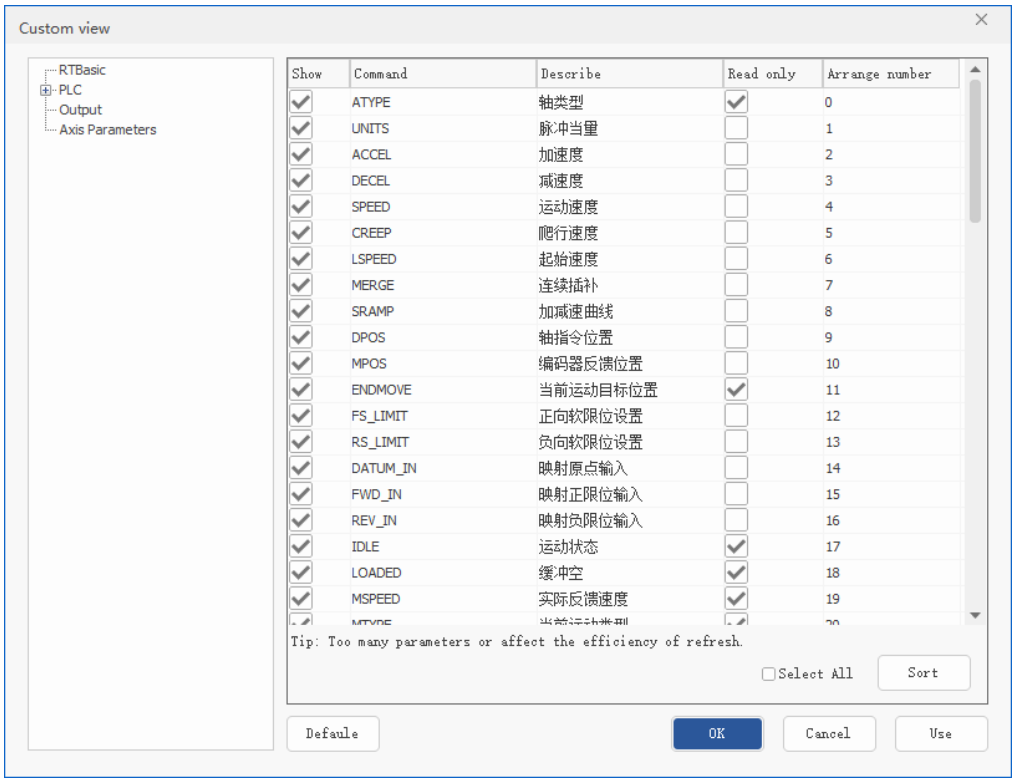
	invalid. When controller position limit signal takes effect, axis will stop immediately, and stop deceleration is FAST_DEC. Normally, FAST_DEC is set to be 10 multiples of DECEL.
IDLE	Check current axis motion state, 0 – in motion, -1 – motion ends, read-only parameter.
LOADED	When there are no motion instructions to be buffered in motion buffer, it will return TRUE, otherwise will return FALSE, read-only parameter.
MSPEED	Axis actual speed form measured feedback, unit is units/s. MSPEED is differential value from MPOS, read-only parameter.
MTYPE/NTYPE	MTYPE: current motion command's type. NTYPE: the first one motion command's type in buffer. When in interpolation linkage, for slave axis, it always returns to merged vector main axis's motion command type, read-only parameter.
REMAIN	Return uncompleted distance of axis current motion MTYPE, unit is units, read-only parameter.
VECTOR_BUFFEREED	Return to uncompleted distance of axis current motion and buffer motion, unit is units. It is compound vector distance for multi-axis interpolation, unit is units.
VP_SPEED	Return to current axis motion's planned speed, unit is units/s. When in multi-axis motion, interpolation motion's merged speed will be returned, not the sub-speed of the main axis. If it is not master axis, it returns to related merged vector speed of axis and relative component velocity, the same effect as MSPEED, read-only parameter.
AXISSTATUS	Check all bits' statuses, such as, forward/reverse hard position limit, forward/reverse soft position limit, axis running status, etc., read-only parameter.
MOVE_MARK	MARK label No. of motion instruction, this is written into motion buffer together with motion instructions. Each motion command is called, MOVE_MARK will add one automatically. If need to force to assign MOVE_MARK, it needs to be set once before motion. It can pause in different MARK boundaries through MOVE_PAUSE.
MOVE_CURMARK	Return MOVE_MARK label No. of current axis which is running.

For more details, please refer to "RTBasic Programming Manual".

(1) Axis Select: you can select axis as needed.



(2) Parameter Select: you can choose needed axis parameters, but not recommend too many parameters, the refresh efficiency will be affected.



"Show": show the parameter or not?

"Command": axis parameter

"Describe": please refer to above form

"Read Only": when it is checked, which means the command is only read, if not, it can be modified.

"Arrange number": the parameter No. from up to down.

"Sort": it can sort parameters by yourself, click it, then it will become "sorting", at this time, you can click parameters one by one as needed order, corresponding No. will become new No., after that, please click "sorting" again, and click "OK" / "Use". If you want to sort one certain parameter as 10, but make a mistake, it is 9, sorry, please end it, then sort again.

"Default": click it to make this window resume as default state.

4.2. Project / Label / Hmi Window

4.2.1. Project View

Generally, "Project View" is shown in the left side of the RTSys. It mainly shows how many files in the project, file type, file task No., axis parameters configuration, EtherCAT node, ZCAN node, etc., which can be opened by menu "view" – "project". But axis configuration and EtherCAT configuration functions should be opened separately (menu "controller" – "settings" – check "enable axis config & EtherCAT config").

*For EtherCAT node configuration, please refer to Appendix C.

(1) Add Project File / Configure File

a. Add Project File (add to project):

For program files, there are Basic file, PLC file, HMI file, C language file, etc., and these files' auto run task No. can be set. Please choose one program file, then right click "add to project", and select the format.

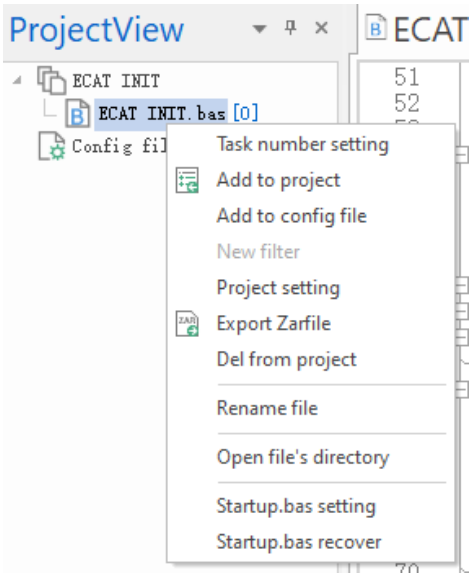
b. Configure File (add to config file):

Generally, configure drive's .zml file. Right click "add to config file" – "add to project" – then select existed .zml format file, and for configuration file, no need to set run task No.

c. Startup.bas file

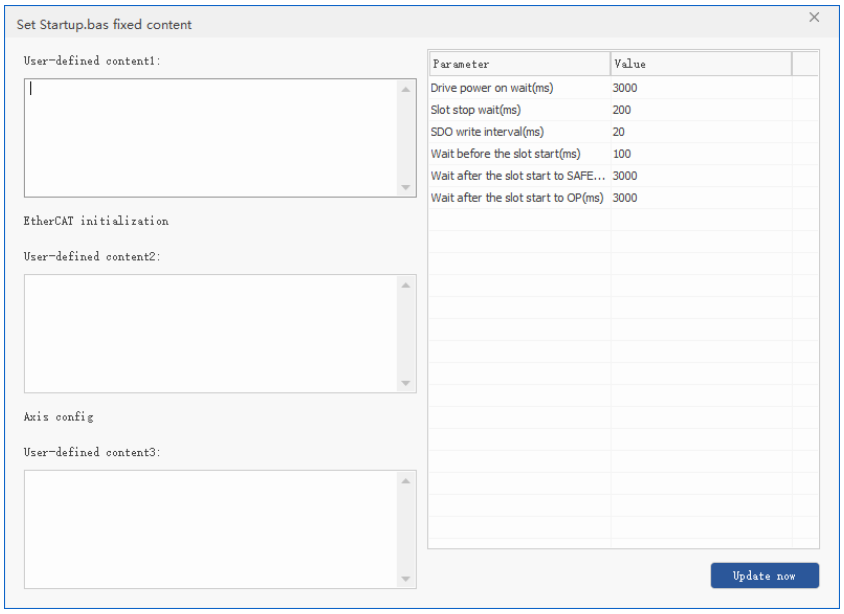
It is the controller's EtherCAT bus initialization configuration file, which can be generated and opened automatically in "Axis Config" / "Controller" – "Settings".

d. Note: names of added project file and configuration file can't exceed 26 characters.



Double-click file to open the file and you can edit then, in project view, right click, corresponding file setting window will appear, information is below:

- Task number setting: set auto run task No. for selected file.
- Add to project: add this file into the project, for details, please refer to "project operation".
- Add to config file: add .xml file / .zml file into project. After adding .xml file, it can be converted to .zml file automatically.
- New filter: build new file group to classify all kinds of files.
- Project setting: set the project compile platform.
- Export zarfile: generate ZAR encryption file, for details, please refer to "ZAR Downloading".
- Del from project: delete the selected file from current project, but the under this project path will not be deleted.
- Rename file: rename the selected file, and the file of the project path will be modified synchronously (please close the file before rename).
- Startup.bas setting: add fixed configuration content by yourself to Startup.bas file or modify some parameters' data in Startup.bas., then corresponding window will appear, please see below.



In left part, you can add basic program code, then it will automatically insert to corresponding program line while generating Startup.bas file. **In right part,** you can modify parameters in Startup.bas file.

Notes:

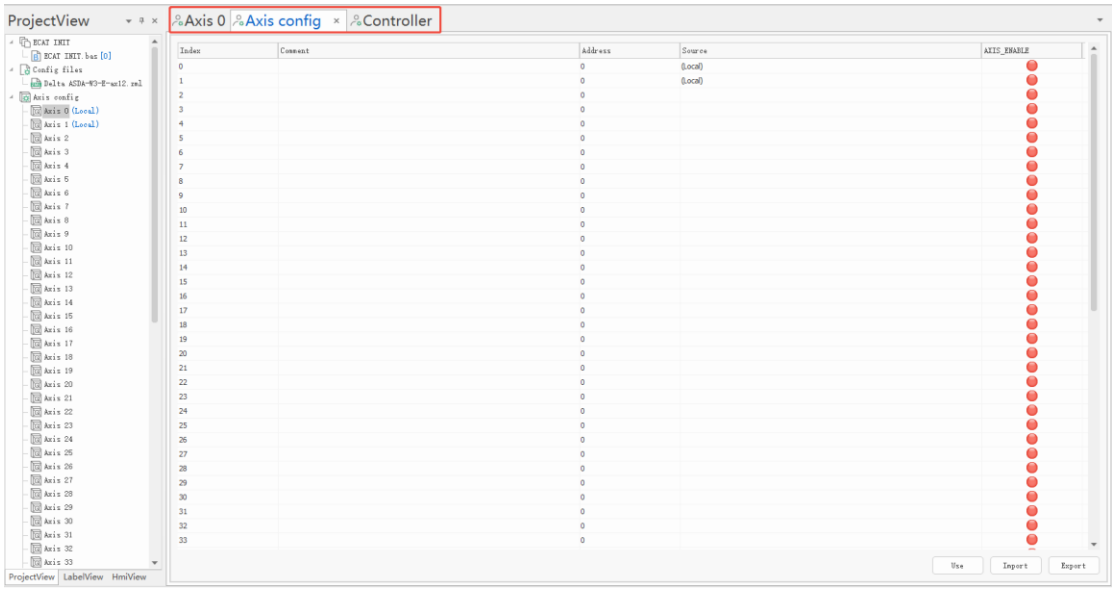
- After applying generated Startup.bas file, you can set and modify in this window, click "update now, then parameters are updated synchronously.
- Startup.bas file can be generated in "Use" button of the interfaces of "axis config", "controller", "Drive n", "ZCan Node – Node N".
- Startup.bas recover: record the Startup.bas file that was modified and saved, and it can resume the Startup.bas file of one certain time.

(2) Axis Configuration

It is controller's axis list, which shows how many axes in total of now connected controller (actual axis + virtual axis). Here, you can configure functions and parameters of each axis directly. And configured axes' type all can be viewed in "project view" – "axis config", local is local pulse axis, EtherCAT is bus axis.

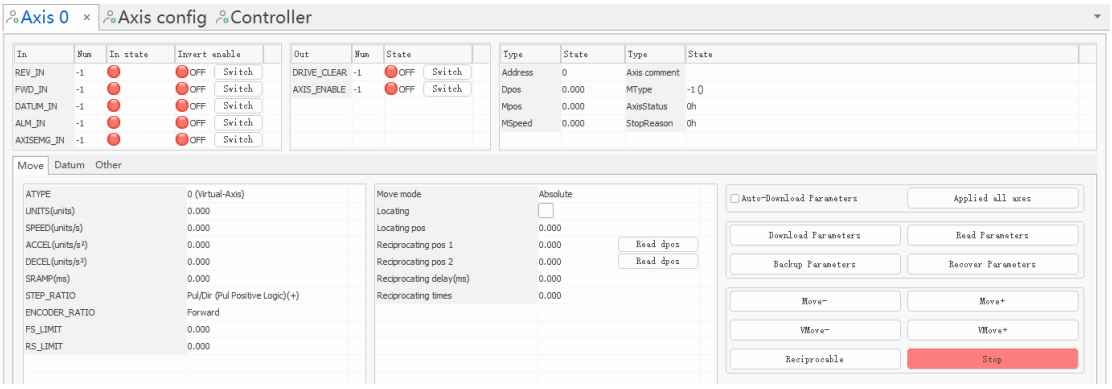
a. Axis Configuration Main Interface

It can show each axis' basic information, including axis note, axis source (each axis' type configuration), axis enable state, etc., and it supports generating this axis configuration content as Startup.bas configuration file by one click, and supports importing, exporting other axis configuration files.



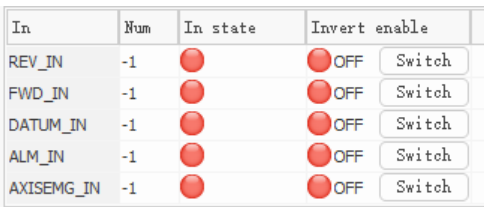
b. Single-Axis Configuration Interface

Here, you can do axis configuration for each single axis, and you can directly set single-axis' hard position limit / origin switches, axis enable, IN / OUT invert, etc., also configure axis basic parameters, homing, simple manual motion.



--how to operate each module--

■ **IN:**

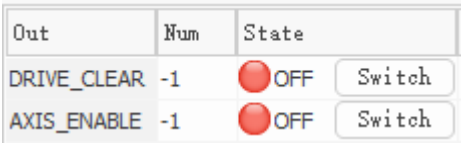


modify each IN No. according to real situation, and it will take effect immediately, set as -1 to cancel configuration. After configuring switch, you can select whether opens INVERT_IN or not. At the same time, it supports "auto-configure IO", please go to "controller" – "in mapping",

after modified IO No., now the No. will be configured automatically according to starting IN No., but please meet below 3 conditions:

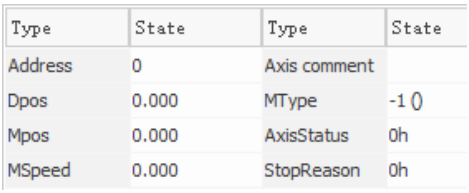
a)	Axis No. should be corresponding.
b)	The quantity can't be 0.
c)	IN No. is -1, or start from original starting No., +1 gradually.

- **OUT:**



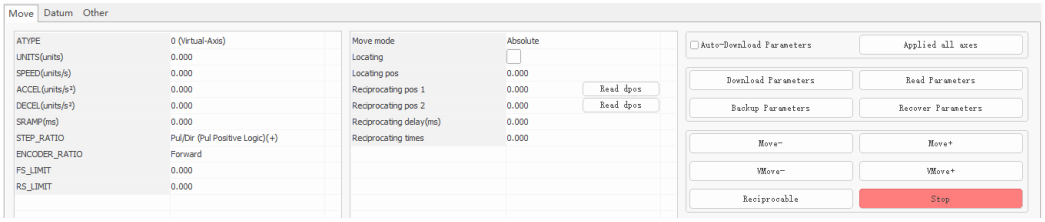
modify each OUT No. according to real situation, and it will take effect immediately, and select whether opens OUT or not (-1 = cancel configuration).

- **Read Axis State:**



real-time get axis state and axis motion position data, among them, axis note and axis stop reason can be written, others are read-only parameters.

- **Move:**

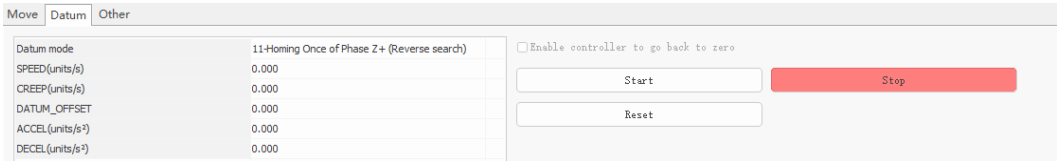


set single-axis' basic parameters and axis motion methods, and the "stop" only stop single-axis.

Button Name	Functions & Description
Auto-Download Parameters	Automatically download parameters of "axis config" into controller, will synchronize in "axis parameter" window.
Applied All Axes	Apply the current axis' configured parameters in all axes (please check "auto-download parameters" to make it valid at first).

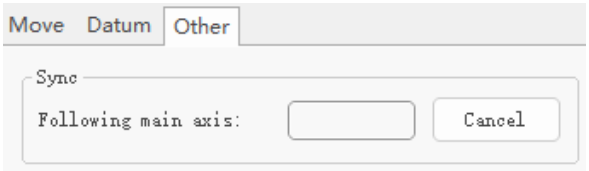
Download Parameters	Download parameters of “axis config” into controller, will synchronize in “axis parameter” window.
Read Parameters	Read the parameters configuration of “axis config”, and apply it into “axis config” parameters.
Backup Parameters	Backup the current axis’ configured parameters, and generate one “ParaBackup.ini” file into the project folder.
Recover Parameters	Make the current axis parameters resume as parameters in backup file.
Move-	Control the current axis do reverse motion.
Move+	Control the current axis do forward motion.
VMove-	Control the current axis do continuous reverse motion.
VMove+	Control the current axis do continuous forward motion.
Reciprocable	Control the current axis do inverse and forward motion according to configured position and times.
Stop	Stop current axis motion.

- **Datum (homing):**



set homing mode and corresponding homing parameters, and enable homing motion (there are many homing methods according to different axis types).

- **Others:**



read superposition main axis No. while doing synchronous motion, also can cancel the synchronous motion.

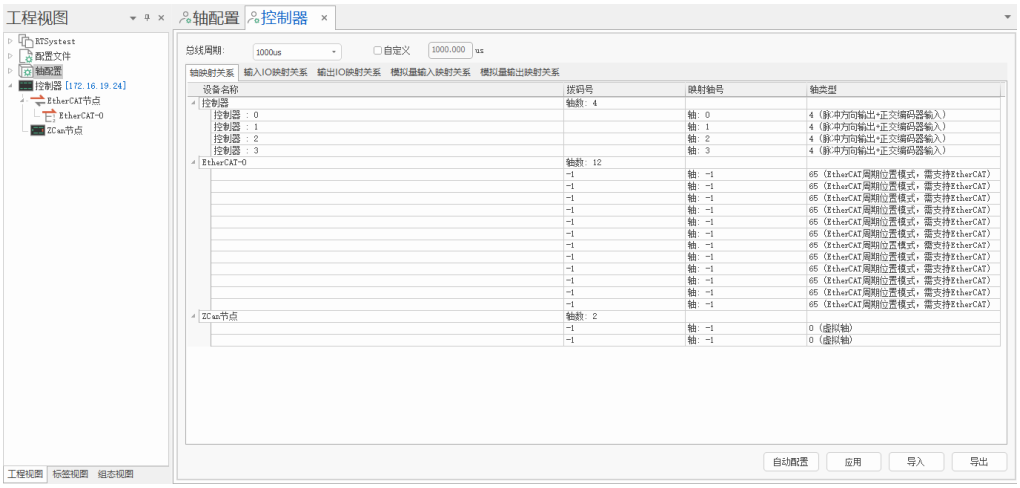
(3) Controller

You can configure controller's corresponding parameters, including axis mapping, digital IO mapping, analog AD/DA mapping, controller bus period modification, etc., and support setting axis / IO / analog mapping No. and type automatically by one button, at the same time, you can export configured data to save or import other configured data.

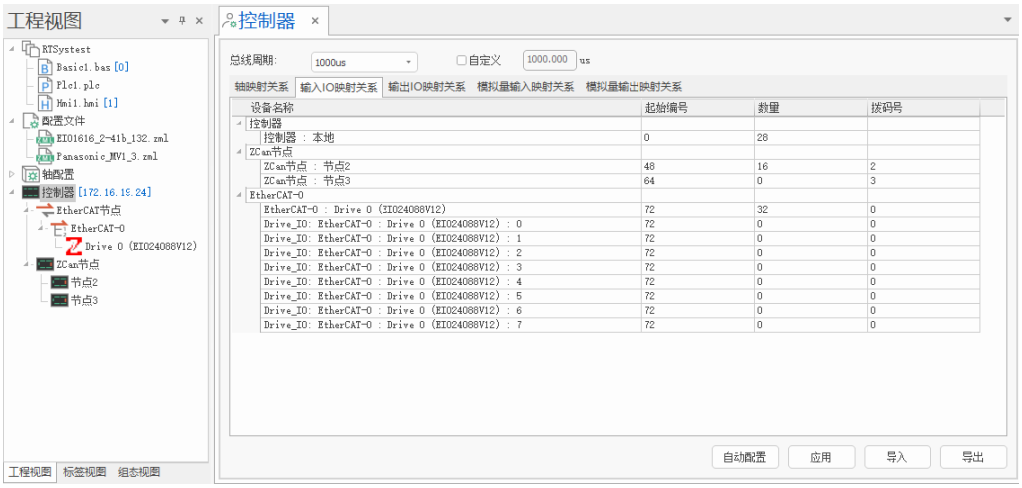
Right Click "controller":

Item	Description
Add to xml / zml list	when new xml/zml file is added into EtherCAT folder under RTSys software directory, it needs to update here, then software can scan the file.
Update xml / zml list	Add .xml/.zml file into EtherCAT folder under RTSys software directory.

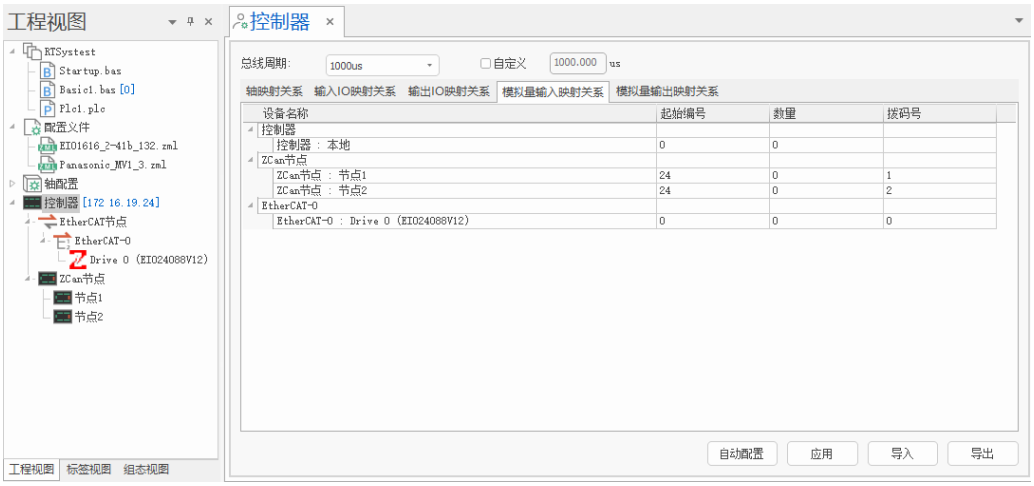
- **Axis Mapping**: it can manually assign controller's axes, map axis No. and configure axis types. Specifically, according to controller total axes, assign how many pulse axes, EtherCAT axes, CAN expanded axes. After that, please click "auto config", then axis mapping can be completed and axis type can be set (default: pulse axis = 4, EtherCAT axis = 65, which can be manually modified).



- **IN Mapping / OUT Mapping**: read device node of connected controller, CAN & EtherCAT module, at the same time, show IN mapping starting No. and how many INs of each device. "Auto configuration" and "use configured parameters in "Startup.bas" file" are supported.

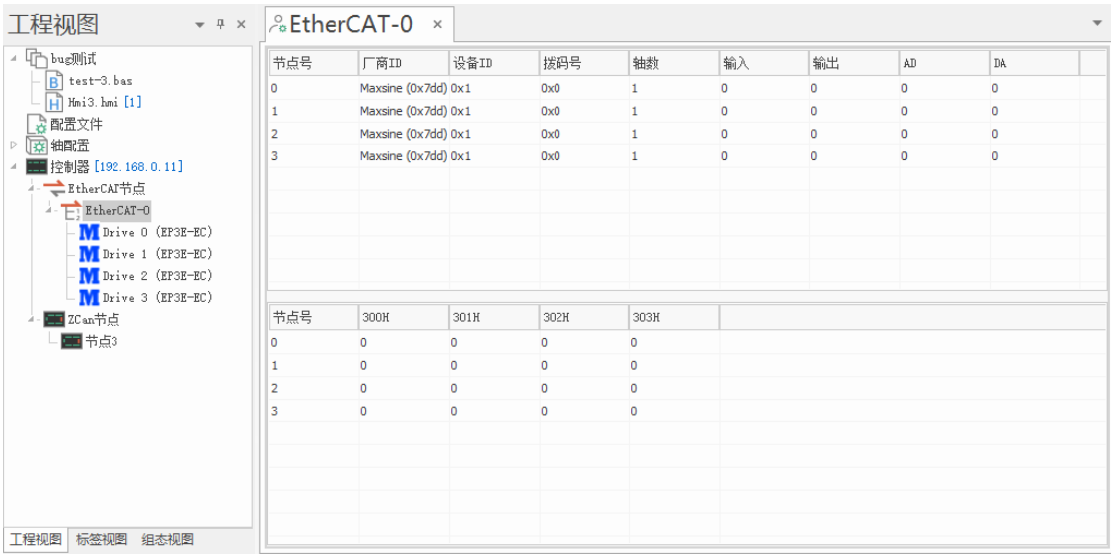


- **AIN Mapping / AOUT Mapping**: read device node of connected controller, CAN module, at the same time, show IN mapping starting No. and how many INs of each device. “Auto configuration” and “use configured parameters in “Startup.bas” file” are supported.



(1) EtherCAT Node / EtherCAT – 0

It shows EtherCAT node devices that are connected to controller’s each slot No., and reads scanned EtherCAT devices’ parameter automatically.

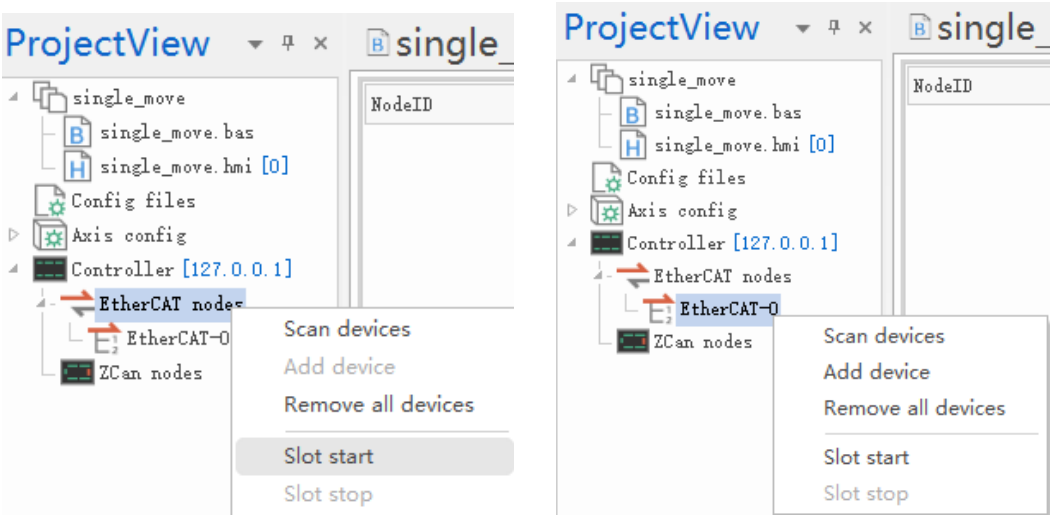


If you right click the EtherCAT Node / EtherCAT – 0, below buttons will be shown:

- ✓ **Scan device:** scan EtherCAT node devices that are connected to controller, please manually click it, then node devices will be shown.
- ✓ **Add device:** manually add corresponding EtherCAT devices, and it can manually add devices to configure in advance even when there is no real device connected.
- ✓ **Delete all devices:** deleted all devices on the EtherCAT node.
- ✓ **Open the bus:** open all EtherCAT devices' bus on the slot No.
- ✓ **Close the bus:** stop all EtherCAT devices' bus on the slot No.

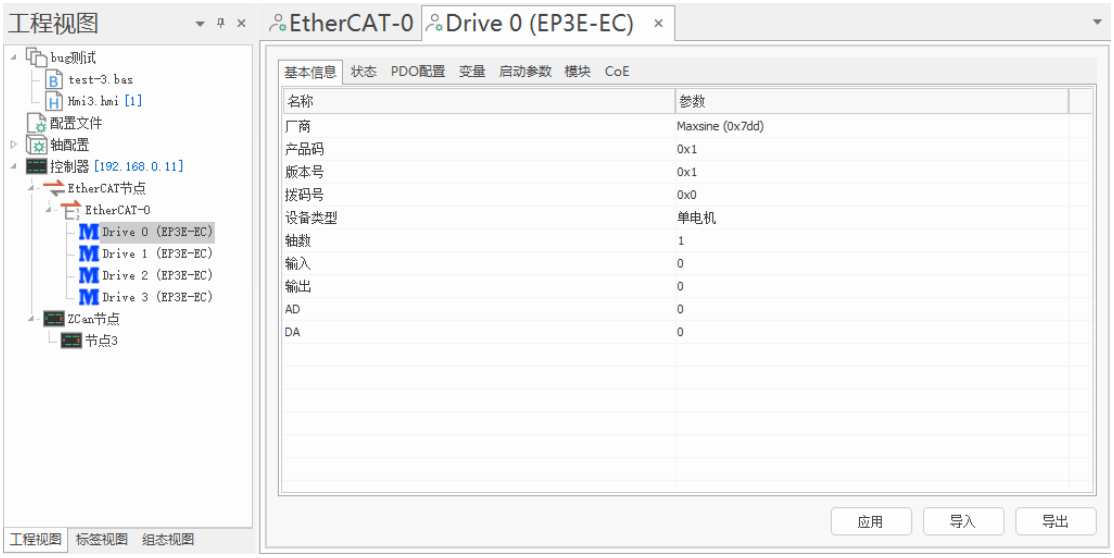
If you right click one single EtherCAT device, below buttons will be shown:

- ✓ Insert device: insert other EtherCAT devices manually (it will be inserted above current clicked device)
- ✓ Delete device: delete selected device
- ✓ Rename: rename selected EtherCAT device
- ✓ Copy config to device: copy selected device's configuration to other node devices.



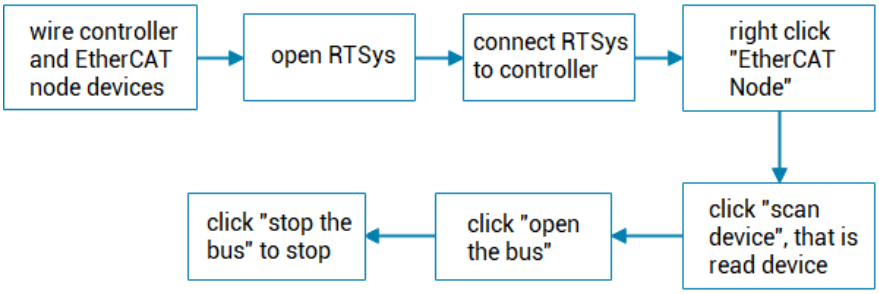
a. single node device (Drive n)

It shows selected EtherCAT device's basic information, connection state, variables, etc., and supports customizing PDO configuration, opening parameters, etc.



"ON parameter" can modify / add corresponding data dictionary parameters in CoE protocol (in RTSys project view – double click "Drive n" – "ON parameter" – "add/modify").

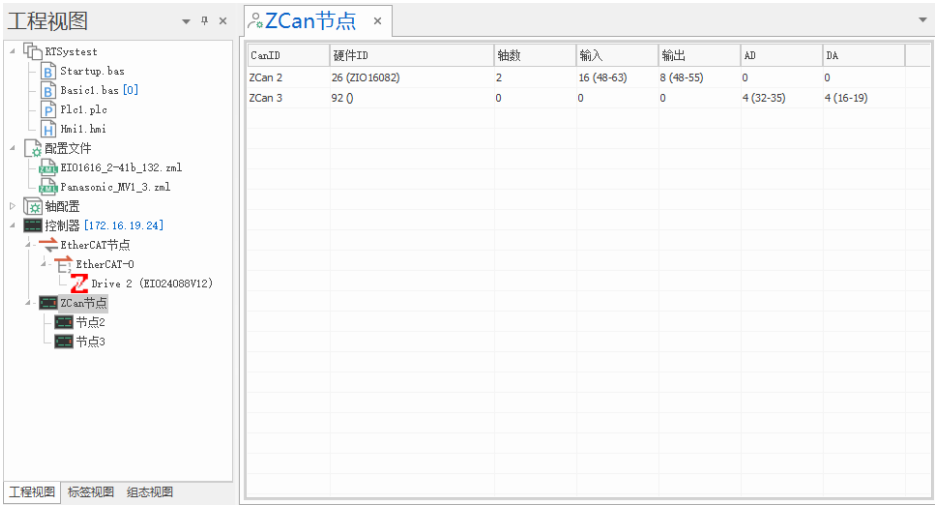
b. EtherCAT Node Usage Steps:



Note: if the software can scan the drive, but not show specific drive name, generally it lacks of this device's xml file.

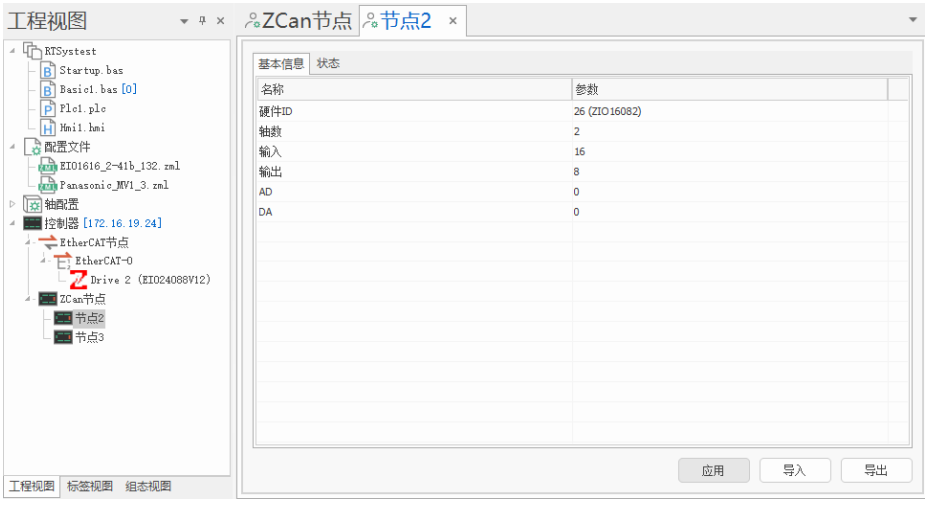
(2) ZCan Node

Here, read scanned ZCan node devices, double click Zcan node, then all ZCan devices' basic information will be shown.



a. Node n

According to ZCan DIP combination value, it is node No., double click to see ZCan node device's basic information, IO, AD, DA, PWM, etc.

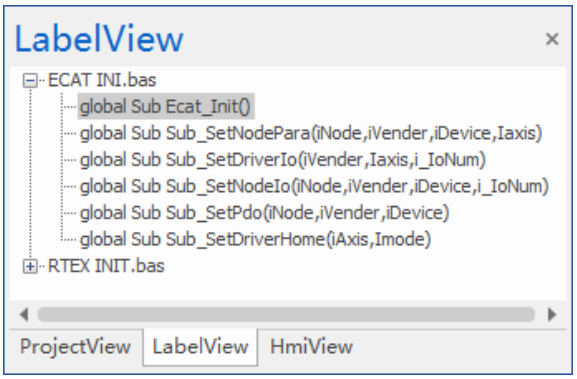


b. ZCan Node Usage Process

After wiring controller and EtherCAT node device correctly, then connect RTSys to controller, next click “scan device”, now device and information can be read.

4.2.2. Label View

Generally, it is also shown in the left side of RTSys, which is embedded in “Project View”. It mainly shows SUB functions list of all Basic files. And double click one certain SUB, it will jump to corresponding definition file and line No. in the program (view – label).

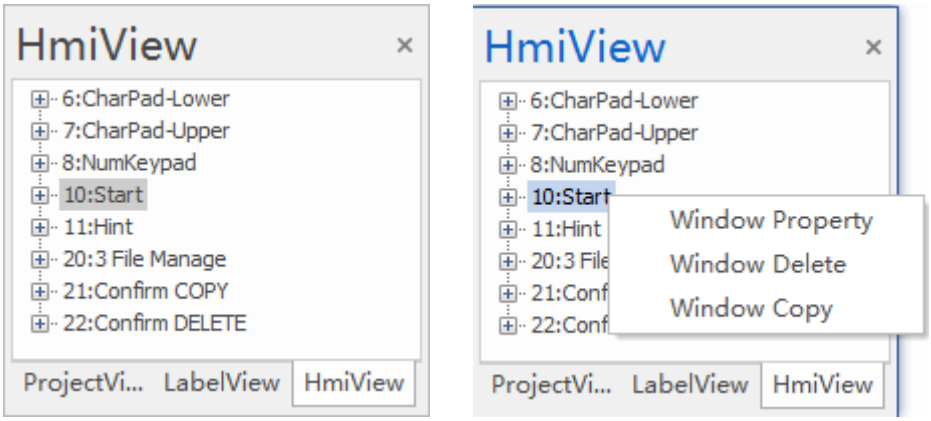


4.2.3. HMI View

Generally, it is also shown in the left side of RTSys, which is embedded in “Project View”. It mainly shows all windows and components of each window of HMI file (view – HMI)

In this HMI view, window 6, 7, 8 are inner keyboard window by default. For window 10, it is initial window, you can add components in this window. Double click the window, all components under this window can be shown / hided, or click +/- before window No.). Double click the corresponding component, the component’s “property” editing window will pop up. Put the mouse at window name, then right click, below right menu window appears.

- Window Property: open “property” editing window of selected window
- Window Delete: delete selected window and all included components
- Window Copy: copy selected window and new build window, then puy content into new window.



4.3. Command & Output Window

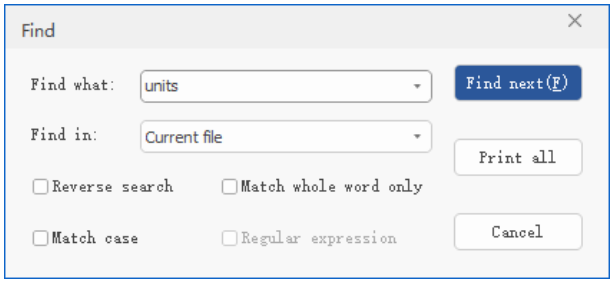
This window can check and output controller's all kinds of parameters, axis motion, program running result, program error information, etc., also can print content of "print & output function" in the program.

More details, please refer to "Command & Output",

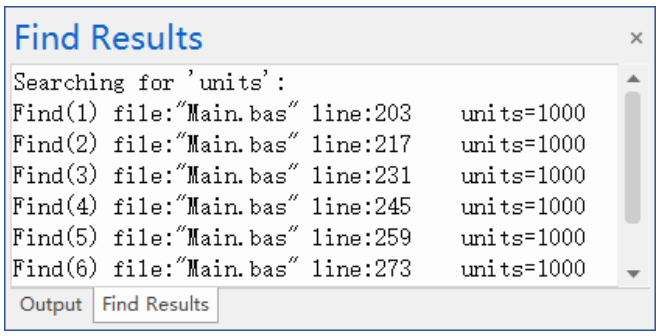
4.4. Find Result Window

This window can show information that is found and printed out, which is matched with "Find" function.

"Find" method: click menu "edit" – "find" to open below window.



Click "Print all". Then, searched results will be shown as below, it can be seen corresponding file name, line No., content all are here.



4.5. Help Window

Click menu “view” – “help”, help documents will be shown. In addition, in Basic / PLC program, double click one certain selected program, corresponding command introduction also will be shown in this window, if it doesn’t appear, press F1 after selecting the command.

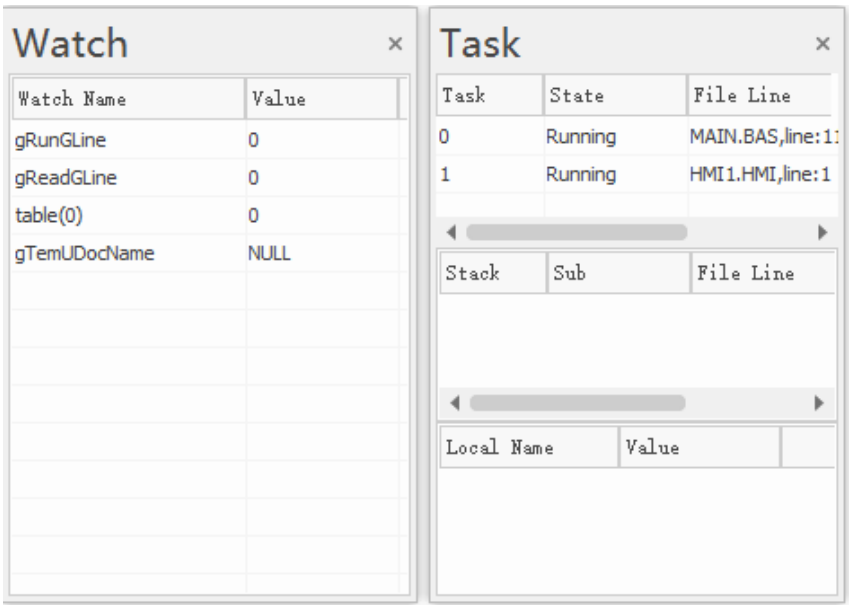
4.6. Task / Watch Window

Task / watch window is used to check task running situation. It belongs to program debug part, while debugging, open these two windows.

Note: task & watch window only can be opened while debugging.

--How to Operate--

After connection (controller / simulator), click menu “debug” – “Start/Stop Debug”, when opened, you can open task / watch windows by “view” – “Task” / “Watch” (usually windows will be opened automatically after starting debugging)



A. Watch Window

It is used to monitor the value changes of valid expressions such as global variables/file variables/structures/registers in the current file, and automatically obtain parameter values and display them when the program is running. But what content to be watched, you need to enter by yourself, double-clicking a blank space can enter/delete. Also, you can select a variable in the program editing area under debugging state, then right-click "Add to watch", and modify or add monitoring items by double-clicking the monitoring content name. (don't support watching "LOCAL" local variables, please check in TASK window).

Global variables support double-clicking to modify value manually. If the watched value shows "Online command fail of error……", which means this content is not global variable and it is not in current watch file.

B. Task

It is used to view the detailed status of each task in the current project. According to the task No., you can view the running/stopped status of the task. If it is running, it will display "Running", and if it is stopped, it will display "Stopped". At the same time, it will display the file name and the corresponding line No. for running file. The example in the above figure shows multiple tasks. The controller supports multi-task operation. The maximum number of supported tasks can be viewed in the Taskes parameter in the "Controller Status" window.

C. Stack

When the program calls the SUB procedure, the original state and local variables are automatically stored, which is called a stack.

D. Local Name

Monitor the LOCAL variable definitions in the current project file and the parameters passed in by SUB calls.

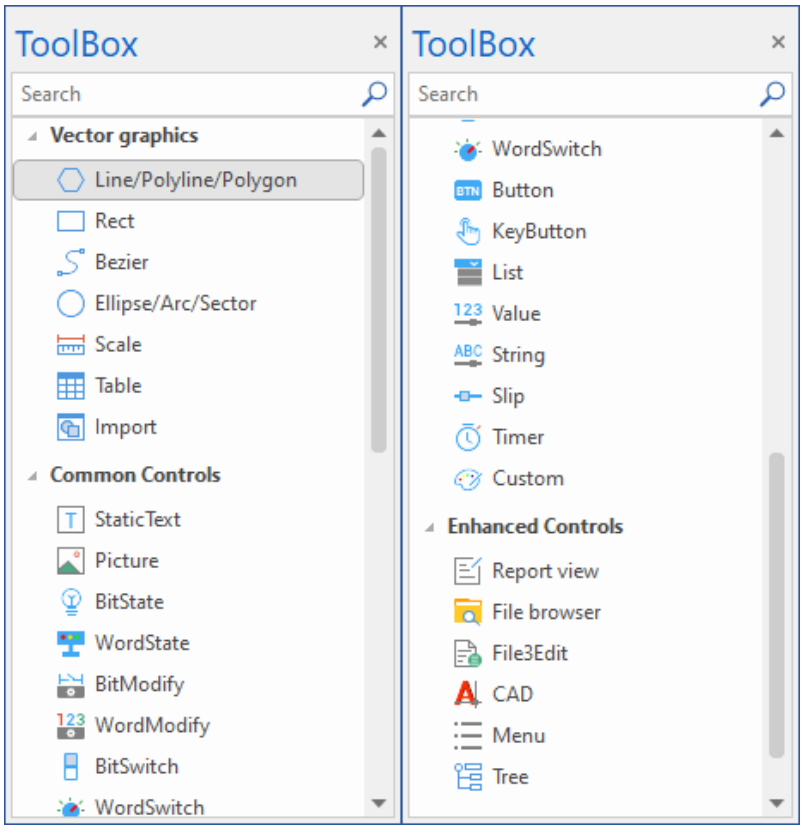
NOTES:

- The local variables of different stacks of the same task are different, even if the names are the same.
- The stack level is limited. The stack level for subprogram calls is generally 8 levels. Please pay attention to the use of recursion.

4.7. Tool Box

Tool box is used to save all kinds of HMI components.

When you developing HMI, you can add them in this window. It is opened by menu View – Tool Box. Generally, it is shown in the left.

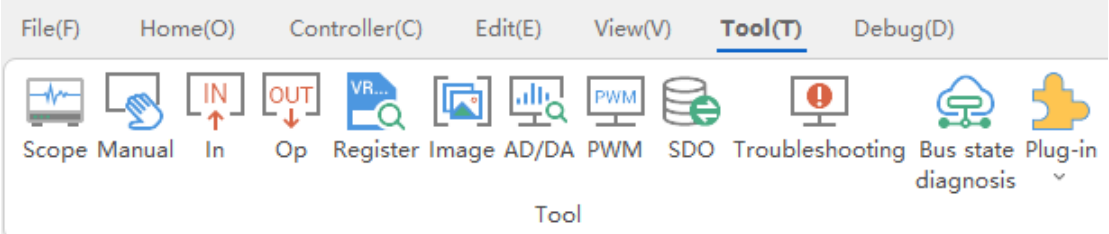


Name	Image	Description
Vector Graphics		
Line/Polyline/Polygon		Draw the corresponding line segment or polygon according to the number of points drawn.
Rect		Draw the rectangle.
Bezier		Draw a third-order Bezier curve with four points
Ellipse/Arc/Sector		Drag to draw a full circle/ellipse/arc/sector
Scale		Draw interval scale with equal-space
Table		Draw a 3x3 table, you can customize the table style
Import		Import vector image
Common Controls		
Static Text		Add statuc text in the window, you can customize related properties.
Picture		Insert the image from the system / background picture lib.
BitState		Show the corresponding state according to bound bit register address's value.
WordState		Show the corresponding state according to bound word register address's value.
BitModify		Set bit register address's value according to component action state.
WordModify		Set word register address's value according to component action state.
BitSwitch		Set bit register address's value according to component action state, and show corresponding state (BitState + BitModify).
WordSwitch		Set word register address's value according to component action state, and show corresponding state (WordState + WordModify).
Button		Achieve state switching/window switching/soft keyboard switching according to component actions, etc. There are only two display states and registers cannot be bound

KeyButton		Used to bind with virtual keys/actual keys through actual key actions
List		Multiple list items can be displayed, and the corresponding options can be switched by the value of the bound register
Value		Edit and display the value and modify the value of the corresponding binding register
String		Edit or display a string and change the value of a word register
Slip		Change the word register value by dragging the slider
Timer		Timed refresh for repeated actions
Custom		Dynamic drawing is achieved by calling basic functions in the component area
Enhanced Controls		
Report view		Display multi-group data in a form way, which can show and manage form data.
File browser		Show current content, and show file information in a form way.
File3Edit		Supports editing components for developing third-party programs in HMI
CAD		Show vector image
Menu		Set menu item, that is, SUB actions can be triggered and called by clicking correspondinh menu item.
Tree		Show all form items in a dendrogram way, it can be show / hide sub-tree through single clicking the corner icon / double clicking tree node contents.

For details, please refer to RTHmi Programming Manual.

Chapter V Tool Window



--how to operate quickly--

Name	Image Mark	Description
Scope		Watch / debug the program that is running, and it can convert data to graphic, which can show real-time changes.
Manual		Set axis parameters to operate the motor manually and directly.
In		Real-time watch IN state.
Op		Real-time watch OUT state
Register		Real-time watch each register values.
Image		Used to show and check the image in the latch channel, or change latch channel's image.
AD/DA		Watch AD/DA values
PWM		Set / read PWM's duty and frequency value
SDO		Write and read EtherCAT data dictionary into controller.
Troubleshooting		Watch controller state and show trouble shooting information.
Bus state diagnosis		Diagnose and show diagnosis information of EtherCAT and RTEX bus states.
Plug-in		Add custom small plug-in, there are "xplc screen" HMI simulation plug in by default.

5.1. Oscilloscope

5.1.1. Scope Interface

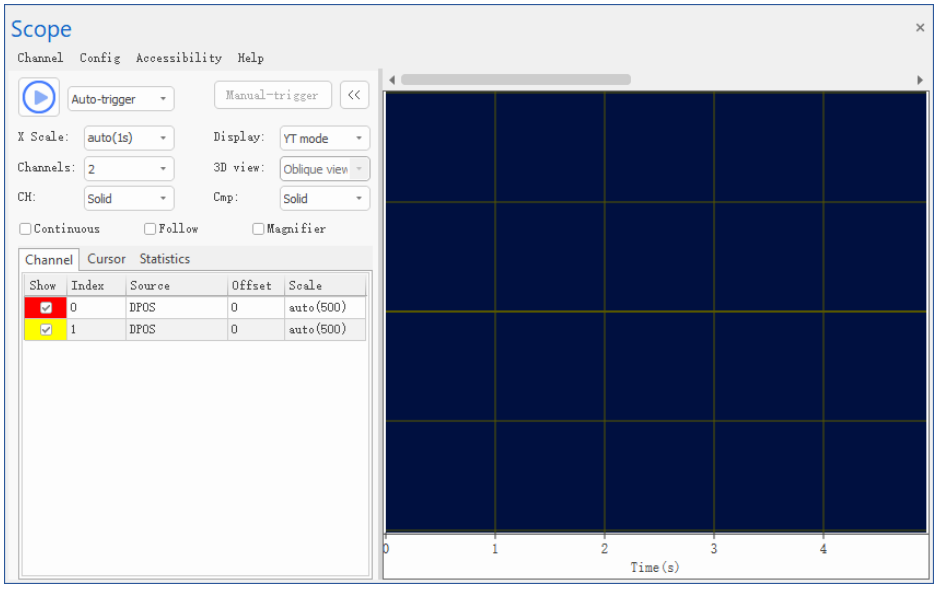
Oscilloscope is extremely important of program debugging and running. It is used to transfer signals that can't be seen by naked eyes into graphics, so it is convenient to analyze change processes of all kinds of signals. Oscilloscope shows controller internal data in graph, it can display different signals, like, axis parameter, axis status, etc., click "Tool" – "Scope" to open the scope window.

--How to Operate--

After editing the program in RTSys, and connecting to controller / simulator, then open the scope, now you can set needed data source and corresponding No., select data sampling trigger method (auto-trigger / manual-trigger / condition-condition), next, click "▶" open button, and download the program into RAM/ROM. At this time:

- if you use auto-trigger, it will sample after clicking ON.
- if you use manual-trigger, after clicking ON, you need to click "Manual-trigger" to sample, then download to RAM/ROM, or download directly after clicking ON, then waiting for Basic to trigger (note, when waiting Basic trigger, "TRIGGER" command should be added in program).
- if you use condition-trigger, please set the condition in advance, check following "trigger by condition" for details.

A. Scope Main Interface & Channel No. Interface



Buttons	Functions
Channel	Selected channel and superposition channel, comparison channel isn't shown.
Config	Open oscilloscope configuration window, set parameters.
Accessibility	Assist in observing waveforms, including searching waveforms, comparing waveforms, and importing and exporting waveforms.
Help	Display the mouse operation guide interface to prompt the mouse shortcut operations in each mode.
	The switch of oscilloscope. When it is in ON state, be shown as  , but it will not trigger the oscilloscope.
Trigger Mode	A. **Auto-trigger:** it will be triggered immediately after clicking the  ON button. B. **Manual-trigger:** it is matched with "manual trigger" button. After clicking ON button, please click "Manual-trigger" to sample, then download to RAM/ROM, or you can choose download it directly after clicking  , then waiting for Basic to trigger it (for this, "TRIGGER" command should be added in program). C. **Condition-trigger:** it is triggered when set condition is met. Therefore, configure the condition, corresponding sub function in advance. After downloading program, it will capture needed data once the condition is matched. Please note it can't modify the condition after clicking "  " button.
Manual-trigger	Trigger oscilloscope manually to sample.
<<	Press to hide the channel name and peak value, and display only the channel No.
X Scale	The scale of the horizontal axis. Select from the drop-down menu to manually enter the value and unit. The default input unit is ms, which is automatically converted to s after input. Place the mouse in the value box and scroll the mouse to zoom in and out the horizontal scale. It is effective in YT mode. Under XYZ mode and XYZD mode, it represents the sensitivity of the left clicking.
Display	➢ **YT Mode** The curve shows different data sources' value with different time, each channel shows a waveform. ➢ **XY Mode**

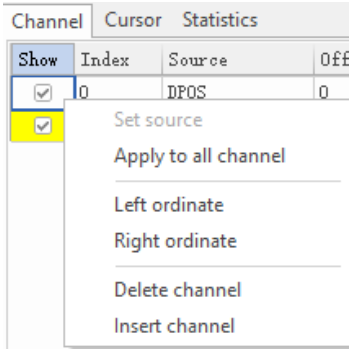
	The XY plane displays the interpolated synthetic trajectory of the two axes, and two consecutive channels of the same type are grouped together to display a waveform. Note: it is invalid when the number of channels is < 2. ➢ **XYZ Mode** XYZ 3D displays the synthetic trajectory. Select the channel as the X, Y, and Z axis in turn. Three channels of the same type are grouped together to display a waveform (channel types include normal channel, overlay channel, contrast regular channel, and contrast overlay channel). Each type can display at most one waveform. Note: When using this mode, the OpenGL version of the display card must be 1.5 or above. And it is invalid when the number of channels is < 3. ➢ **XYZD Mode** XYZD four-channel shows the trajectory in a visualization way, where XYZ is the 3D merged trajectory, and D is the data source displayed in the form of dots. The calculation method is: dot diameter size = current D value ÷ D reference value × D reference size. Parameter modification is located in the "Observer Config" window. Select channels as X, Y, Z axis and D value channels in turn. Four channels of the same type are grouped to display a waveform (channel types include: normal channel, overlay channel, contrast regular channel and contrast overlay channel), and each type can display at most one waveform. Current D value: the size of the data source value at the current position. Note: When using this mode, the OpenGL version of the display card must be 1.5 or above. And it is invalid when the number of channels is < 4.
Channels	Set how many regular channels in total to be sampled. It cannot be modified when ON. When the set number of channels is > the number of channels supported by the controller, a prompt message will pop up: Exceeding the maximum number of channels supported by the controller.
3D View	You can choose oblique angle, front angle, left angle and top angle. The default is oblique angle. This is only valid in XYZ mode and XYZD mode.
Normal channel	The curve is shown under normal channel: point, solid, dashed. For line

	size, and color, please go to “Config” – “Observe Config”.
Contrast channel	The curve is shown under contrast channel: point, solid, dashed. For line size, and color, please go to “Config” – “Observe Config”.
Continuous	When continuous acquisition is not enabled, sampling stops after reaching the maximum acquisition cycle. When continuous acquisition is enabled, the oscilloscope will continue sampling, and will continue sampling after reaching the maximum acquisition cycle, that is, it will not stop sampling until the stop button is pressed. The acquired data will automatically overwrite the previous data. What’s more, all waveform sampling data acquired continuously can be exported (the continuous acquisition function is automatically canceled when using the serial port).
Follow	After turning on the follow, the horizontal axis automatically moves to the real-time sampling position and follows the waveform display.
Magnifier	When this is checked, and the magnified view will be automatically displayed at the lower right of the mouse when the mouse moves to the display area. The magnified view follows the mouse movement and refreshes it in real-time. The magnifying glass parameters can be modified in the "Observer Config" window. YT mode is valid.
Channel	➢ Show Select whether to display the current channel curve. The oscilloscope has four types of channels, including normal channels 1 to 8, overlay channels 1 to 4, contrast channels 1 to 8, and contrast overlay channels 1-4. ➢ Index Select the data source No. to be collected, such as axis No., IN, AIN, TABLE No., VR No., MODBUS No., etc. The No. setting range is from 0 to the maximum number of axes of the controller, and the number can be entered manually. ➢ Source Select the data type to be collected. Left click to manually enter the data type, or click the drop-down menu to select the type parameter. If there is no needed parameter type, you also can add them in the "Data Source Design" window. ➢ Offset

	To set the waveform vertical axis offset, select the offset from the drop-down menu or enter it manually. ➢ Scale The scale of one grid on the vertical axis. When auto is selected, it indicates automatic scale, which is available when the oscilloscope is stopped. The scale value changes automatically according to currently acquired waveform, so that the waveform can be fully displayed on the current oscilloscope interface.
↑	It indicates loss may occur here, which is related to the maximum acquisition cycle. After the oscilloscope starts continuous acquisition, it will re-trigger the acquisition at 80% of the maximum acquisition cycle. At this time, the TABLE data begins to be rewritten, and point loss may occur during this process.
Note: if you need to set the oscilloscope parameters, such as axis No., data source, and oscilloscope "Parameter Config" window, please stop the oscilloscope first and then set them.	

B. Channel Right-Click Functions

When you right click any channel, corresponding items will be opened. It includes “Set source”, “Apply to all channel”, “Left ordinate”, “Right ordinate”, “Delete channel”, “Insert channel”.

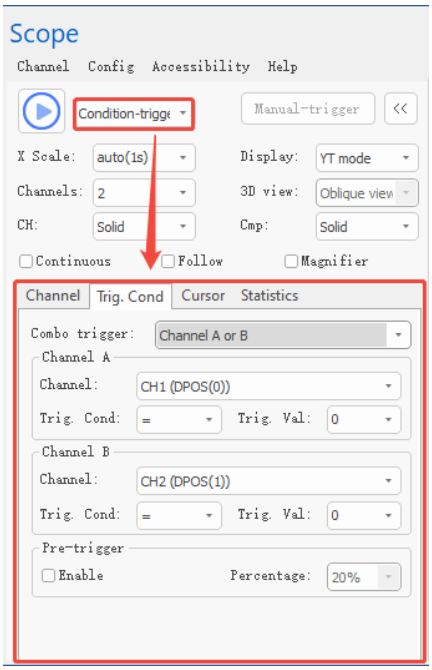


Item	Description
Set source	Set which parameter to be captured (include controller parameters). It is only valid when single-clicking “source”. ➢ Oversampling: it is not valid for normal data source. When it is checked, it captures over the oscilloscope period. By default, capture once in one period. After oversampling is enabled, in one period, set times can be reached (*the length / times is set in*

	"Config" – "Parameter Config"), for example, when it is set as 5, capture 5 times in one cycle. *when oversampling is ON / OFF, it uses and shows the previous end state.*
Apply to all channel	Apply selected items in current showing channels (show, index, source, offset, scale). When the mouse cursor is put in the source, at this time, it only applies the source in all channels.
Left ordinate	Whether to show current channel's vertical coordinate. It is shown in obverse's left side by default.
Right ordinate	Show current channel's vertical coordinate in obverse's right side
Delete channel	Delete current selected channel, when the channel has data, all channels' waveforms will be cleared. And the sub-channel of overlay channel also will change, for example, overlay channel is CH1+CH2, after deleting CH1, original CH2 will become as CH1, original CH3 will become CH2, that is, now new overlay channel is CH2 (before deleting) and CH3.
Insert channel	Insert one channel before the current channel, after inserting, current channel moves down, the total channels + 1. Corresponding overlay sub-channels also change.

C. Condition-Trigger

After opening the oscilloscope, required data will be captured when set conditions are met.



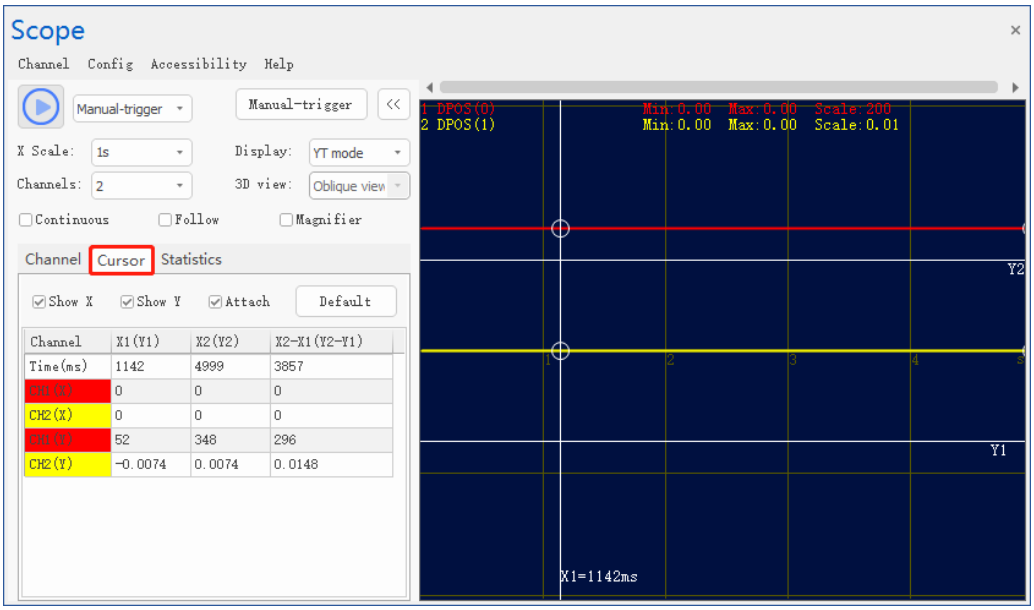
Under condition-trigger, 2 channels can be set, channel A and channel B. Then, 4 kinds of trigger

groups and corresponding conditions can be achieved, please see below form:

Item		Description
Trigger Groups	Channel A	It is triggered when channel A condition is met.
	Channel B	It is triggered when channel B condition is met.
	A / B	It is triggered when channel A / B condition is met.
	A & B	It is triggered when channel A & B conditions are met.
A / B Channel	Trigger Channel	The channel relates to data source.
Conditions	=	It is triggered when it equals to trigger value.
	≥	It is triggered when it is not less than trigger value.
	≤	It is triggered when it is not greater than trigger value.
	Rising Edge	The trigger value only can be numerical value, it is triggered when exceeds the set value (from low value to high value).
	Falling Edge	The trigger value only can be numerical value, it is triggered when exceeds the set value (from high value to low value).
Trigger Value		Enter the condition value. When you select A / B, A & B, channel A trigger value can be channel B, channel B trigger value can be channel A. When you select channel A, or channel B, it can't operate like former situation, it will be triggered immediately.
Pre-trigger	Percentage	Use a certain space (percentage) of buffer to save pre-trigger data (optional range: 0%~50%), for example: percentage = 25%, which means 25% of the collected data is pre-trigger data, and 75% of the collected data is post-trigger data, that is, the collected data graph will contain 25% of the pre-trigger time period data. If the original data graph is < 25%, all collected data will be displayed.

D. Scope "Cursor" Interface

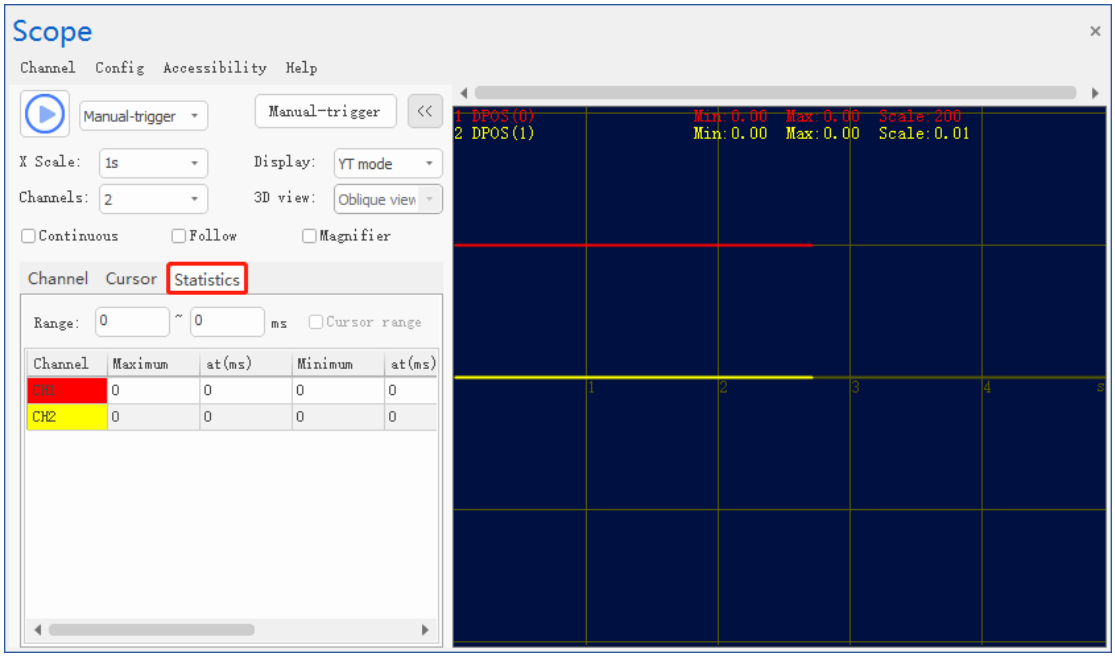
The cursor tool is mainly used to calibrate and measure oscilloscope graph data or coordinate distance. Click the "Cursor" button to display the cursor interface as shown below.



Buttons	Functions
Show X	Select whether to enable cursor X. There are 2 X cursors, X1 and X2. The cursor is at the default position when it is displayed for the first time, and the position at the last startup is displayed when it is enabled again. You can drag (press the left mouse button and move) to change the position of the cursor, and the cursor position cannot exceed the range of the observer. YT mode and XY mode are effective.
Show Y	Select whether to enable cursor Y. There are 2 Y cursors, Y1 and Y2. YT mode and XY mode are effective.
Attach	When checked, several small circles are displayed on the cursor, and the small circles are attached to the current waveform along with the cursor position, making it easier to find extreme values. X1/X2 of YT mode is effective.
Default	Click the button to restore the cursor to the default position. The default position of X1/Y1 is 1/3 of the current waveform display area, the default position of X2/Y2 is 2/3 of the current waveform display area.
Channel	Channel selected by yourself. You can modify the cursor positions on X1(Y1) and X2(Y2) in the “Time(ms)” row.
X1(Y1)	Position of cursor X1(Y1) at this channel.
X2(Y2)	Position of cursor X2(Y2) at this channel.
X2-Y1 (Y2-X1)	The difference between the cursor positions for this channel.

E. Scope “Statistics” Interface

The “statistics” function can automatically count the maximum value, minimum value, magnitude, average value, standard deviation, etc. of the Y-axis data within the specified time range. Click the "Statistics" button to display the statistics interface as shown below. The statistical parameters can be set in the "Config" – “Parameter Config”. It is only valid under YT mode.

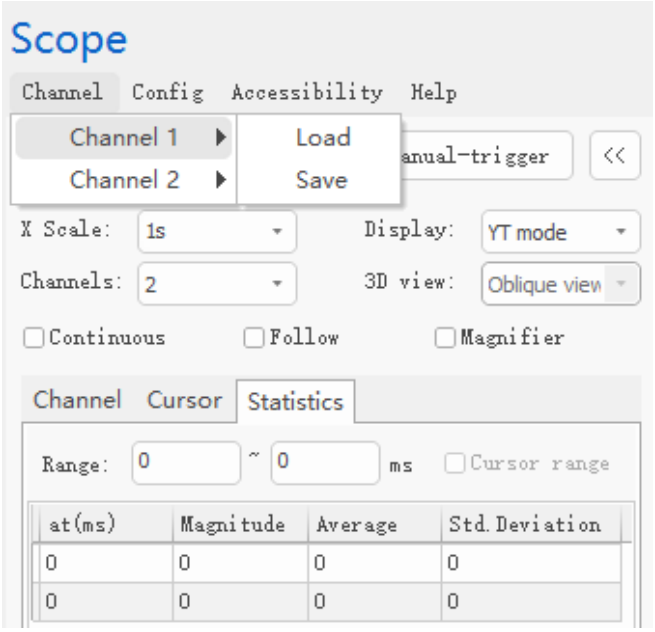


Buttons	Functions
Range	You can enter the range of the statistical horizontal scale, and the statistical content will be automatically updated after entering.
Cursor range	Select whether to use the cursor range. If checked, you cannot enter the range manually. The range of cursors X1 and X2 will be automatically counted. Before using "Cursor Range", you must check "Show X".
Channel	Select which channels are shown.
Maximum	Max value at Y direction in specified range.
at (ms)	The position of max value.
Minimum	Min value at Y direction in specified range.
at (ms)	The position of min value.
Magnitude	Differential value between max value and min value.
Average	Average value at Y direction in specified range.
Std. Deviation	Standard deviation value at Y direction in specified range.

5.1.2. Scope Menu Functions

(1) Scope Channel

Click menu above “channel” button, the interface is shown as below.

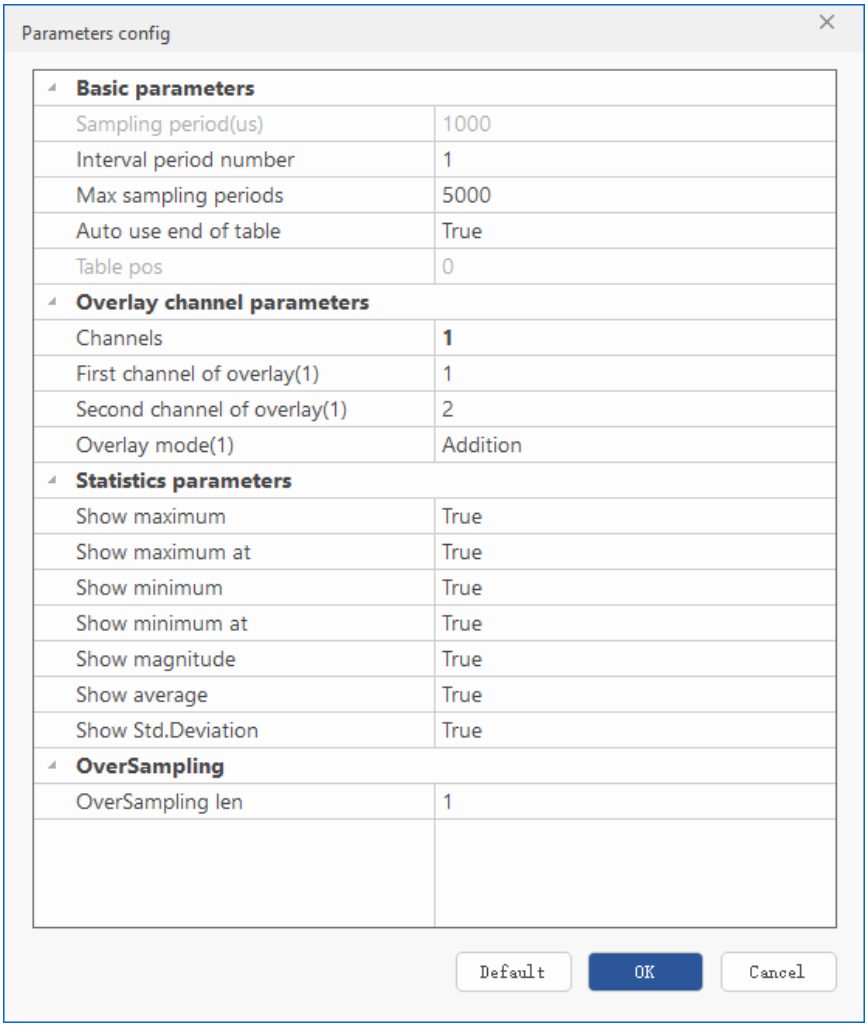


Buttons	Functions
Channel	Show channel selected, but compare channel is not shown.
Load	When loading, it will determine whether the number of points of the loaded waveform is consistent with the current waveform. After successful loading, the waveform of the corresponding channel will be overwritten. The number of points of each channel is allowed to be inconsistent. When the number of points of the loaded channel > the number of points currently displayed, the waveform display will be truncated. When the number of points of the loaded channel < the number of points currently displayed, the waveform display will be automatically completed. The automatic completion is to horizontally complete the points at the position of the last point. (For example, the waveform of channel 1 ends at 50s, and the waveform of channel 2 ends at 60s). The loading file format is .txt.
Save	Export the waveform of the specified channel and the time interval between each point. The format is .txt.

(2) Scope Config

A. Parameter Configuration Window

Click menu above “Config” button, then click “parameter configuration”.

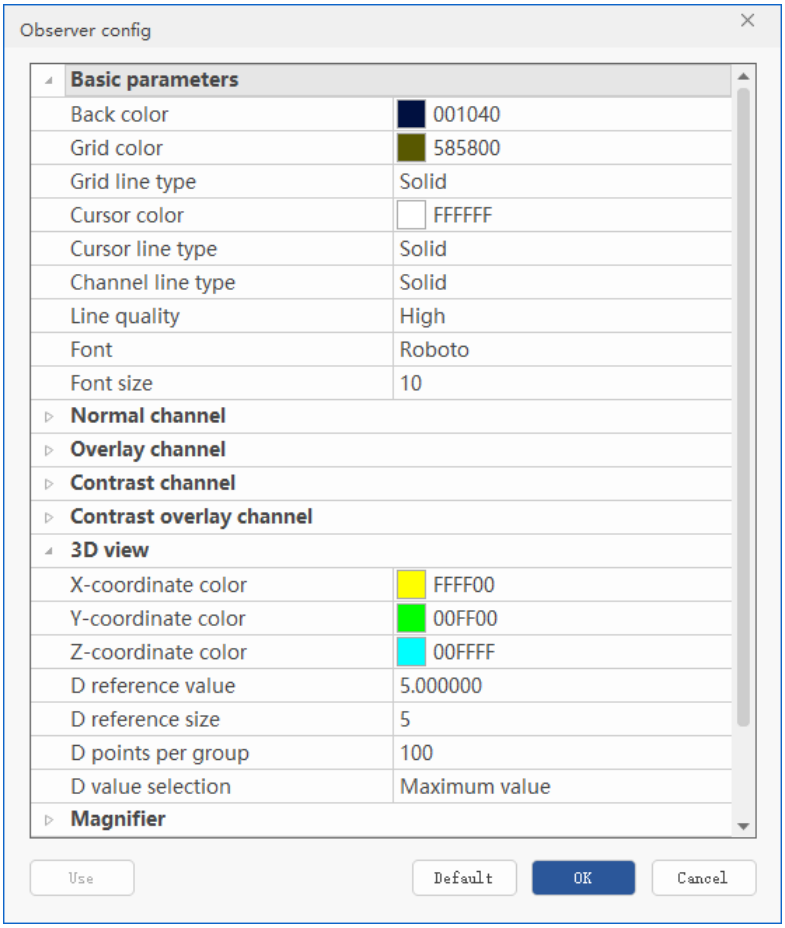


Parameter	Description
Basic parameters	
Sampling period (us)	Time interval between twice sampling by SCOPE, it can’t be modified.
Interval period number	The sampling time interval, the unit is system cycles, which is related to the controller firmware version. The default value is 1ms. You can view it by SERVO_PERIOD. (For example, if the interval cycle number is set to 1, it means sampling once in 1 cycle. If the interval cycle number is set to 5, it means sampling once in 5 cycles, the cycle time depends on the controller firmware version.) Generally, the smaller the interval cycle, the more accurate the sampling data, and

	the larger the data volume per unit time.
Max sampling periods	The total number of sampled data. The larger the value, the larger the sampling range. (That is, the size of the table required for the data collected by one channel)
Auto use end of table	The position where saves the data, default is True.
Table pos	Set the location where the captured data is stored. Generally, the default is to automatically use the space at the end of the TABLE data. When "Auto use end of TABLE array" is set to False, you can customize the setting, but be careful not to overlap with the TABLE data area used by the program. There are three ways to check the size of the controller TABLE space: ➢ use the TSIZE instruction to read. ➢ view in the "Controller Status" window. ➢ print and view the online command? *max.
Export parameters	Select when you need to export oscilloscope channel parameter information. After checking, oscilloscope parameters are exported when exporting waveforms, including: basic parameters, overlay parameters, and channel configuration parameters (No., data source, offset, vertical scale). The default is True.
Overlay channel parameters	
Channels	Select how many channels that are overlayed, select from the drop-down menu (up to 8). Note: overlay sub-channel must be the same type, then the overlay is effective, that is, if the sub-channel 1 is normal data source, the sub-channel 2 is the drive data source, which is not allowed.
1 / 2 CH of overlay (1)	You can choose the channel No. for overlaying.
Overlay mode (1)	Set the overlay mode between 2 channels. There are 2 modes, Addition / Subtraction. After overlaying, the new waveform will be generated, for example, CH: 1-2 means channel 1 and channel 2 overlay in the mode of subtraction.
Statistics parameters	
Statistics parameters	Set the parameter information displayed on the oscilloscope statistics page. The default value is True.
Oversampling	
Oversampling length	Set how many sampling times for each period.

B. Observer Configuration Window

Click menu above “Config” button, then click “obverse config”, then corresponding window will appear, after configured, click “use” to preview how it is after modified, then click “OK”.

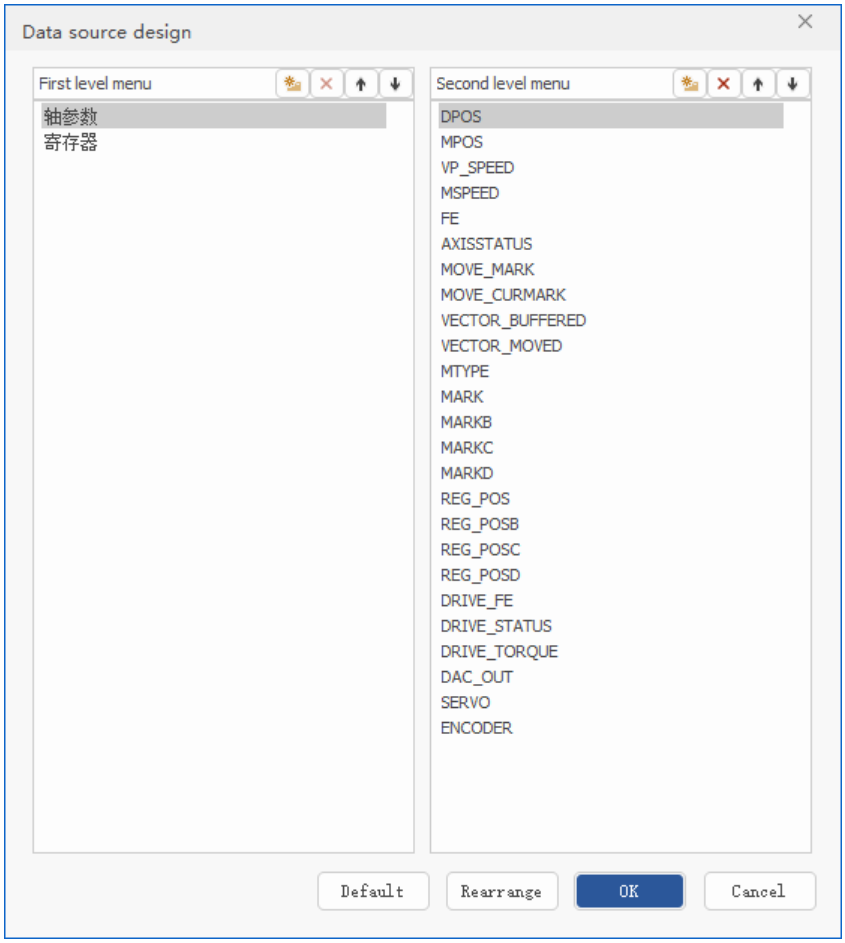


Parameter	Description
Back / Grid / Grid line / Cursor color	Set corresponding needed color.
Grid line type	Set the grid line type, there are solid or dashed lines.
Cursor line type	Set cursor line type, there are solid or dashed lines.
Channel line type	Set channel line type, there are point, solid, dashed lines. **For “point”**, scope will show data that are sampled by SCOPE in fixed period, “point size” parameter can be set. **For “solid / dashed lines”**, sampled points will become one smooth lines, then abnormal data can be easily checked, also, “line width” parameter can be set.
Line quality	Set channel waveform’s line quality, when there are many data, recommend to use standard mode, which can accelerate scope

	performance.
Font / Font size	Set the font and font size of the channel No., channel name and peak value on the waveform display interface.
Normal / Overlay / Contrast / Contrast overlay channel	Set corresponding channel's line width, point size, and channel color.
D reference value / size	Used to calculate the dot diameter size in XYZD mode. The diameter size is related to the ratio of D reference size/D reference value. The larger the ratio, the larger the dot diameter. The calculation formula is: Dot diameter size = current D value ÷ D reference value × D reference size. (The current D value is the value of "D value selection")
D points per group	Display a dot for every N sampling points. (For example, if "D points per group" is set to 100, a dot will be displayed for every 100 sampling points according to the value of "D value selection")
D value selection	The value of the current display dot size in N sampling points can be selected as the maximum value, minimum value and average value. (For example, if "D value selection" is set to the maximum value and "D points per group" is set to 100, the maximum value of every 100 sampling points will be used as the basis for calculating the current display dot diameter)
Magnifier	Set the width, height and magnification of the magnifier.
Search	Set the line width, point size, and channel color of the search results displayed when searching a waveform.

C. Data Source Design Window

Click menu above "Config" button, then click "data source design".



Parameter	Description
First / Second level menu	When there is information in second level menu, the first level menu text is the type, the second level content is data source. When there is no information in second level menu, the first level menu is data source.
	“add” button, add information in first level or second level.
	“delete” button, deleted selected information. Note: axis parameter and register in first level can’t be modified.
	Up / down, used to sort.
Rerrange	Sort items of first level and second level according to characters from A to Z.

(3) Import / Export Config / Recent Conig

A. Import Configuration

Import parameters related to scope, including parameter configuration, observer configuration, data source design, channel parameter configuration (show, No., data source, offset, vertical scale). And the file format of the imported data is .ini.

You only need to click “config” – “import config”, then select which file, when imported, new file data will cover before parameters.

B. Export Configuration

Export parameters related to scope, including parameter configuration, observer configuration, data source design, channel parameter configuration (show, No., data source, offset, vertical scale). And the file format of the imported data is .ini.

You only need to click “config” – “export config”, then select folder to save it.

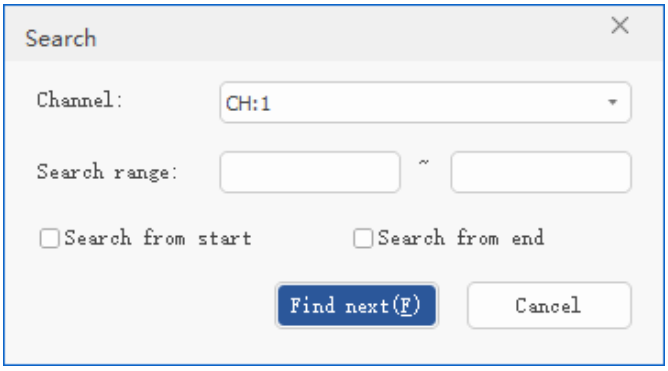
C. Recent Configuration

Import recent saved oscilloscope configurations in one click (max 5).

5.1.3. Accessibility

(1) Search Waveform

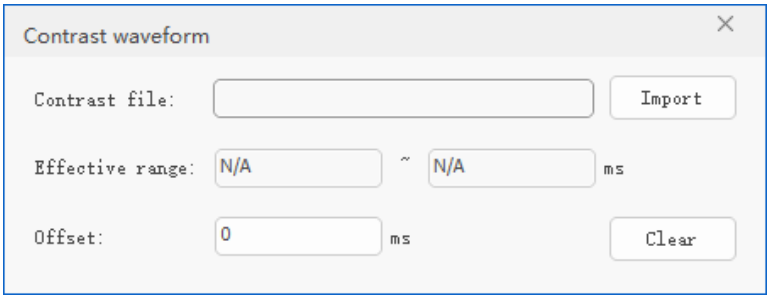
Click “scope” menu “accessibility”, then click “search waveform”, below window will appear. It is used in YT mode.



Parameter	Description
Channel	Which channel you want to know. Valid under YT mode.
Search range	Needed value range of waveform’s Y direction. After that, click “Find next”.
Search from start / end	How to search? From start / from end.

(2) Contrast Waveform

In scope, it can contrast multiple channel waveforms. Click “scope” menu “accessibility”, then click “contrast waveform”, below window will appear. Please note scope should be in stopped state.



A. Contrast file: Import the waveform data file that needs to be compared. When loading the contrast file, it will determine whether the contrast waveform is consistent with the number of current waveform points. The points of each channel are allowed to be inconsistent. (When the number of points of the contrast waveform > the number of points of the original waveform, the waveform will be truncated and displayed. When the number of points of the contrast waveform < the number of points of the original waveform, the waveform will be automatically completed and displayed, the automatic completion is to horizontally complete the points at the position of the last point.) After loading is completed, the parameters of the comparison channel will be updated (whether to show, No., data source, offset and vertical scale). The format of the comparison file is .txt.

B. Effective range: The point range of the waveform in the contrast file. After importing the contrast file, the system will automatically obtain the valid range. The value of the valid range is always positive. The starting value of the valid range defaults to 0, and the end value is the maximum point numbers of the comparison waveform, but it cannot be greater than the maximum number of points of the original waveform.

C. Offset: Set the time position of the waveform offset. When the offset is set, the waveform will move the corresponding offset range. At this time, the effective range will correspond to the number of points that increase/decrease the offset size. (When set to a positive value, it will shift to the right, and when set to a negative value, it will shift to the left)

For Example:

- If the max points of contrast waveform is 5000, the max points of original waveform is 6000, then effective range will be 0-5000. When contrast points < original points, contrast waveform automatically complete points and show.

  When offset of contrast waveform is 200, effective range will become 200-5200.

When offset of original waveform is 1200, effective range will become 1200-6000 – because after offset, the end value will be 6200, but it is bigger than 6000, exceeding 200 will be cut.

When offset of contrast waveform is -200, effective range will become 0-4800 – because effective value always is positive, after offset, it moves left 200ms, then a part can't be shown.

- If the max points of contrast waveform is 6000, the max points of original waveform is 5000, then effective range will be 0-5000. When contrast points > original points, exceeding part will be not shown.

When offset of contrast waveform is 200, effective range will become 200-5000 – because max value of effective range's end value is the max value of original waveform, after offset, exceeding part is cut.

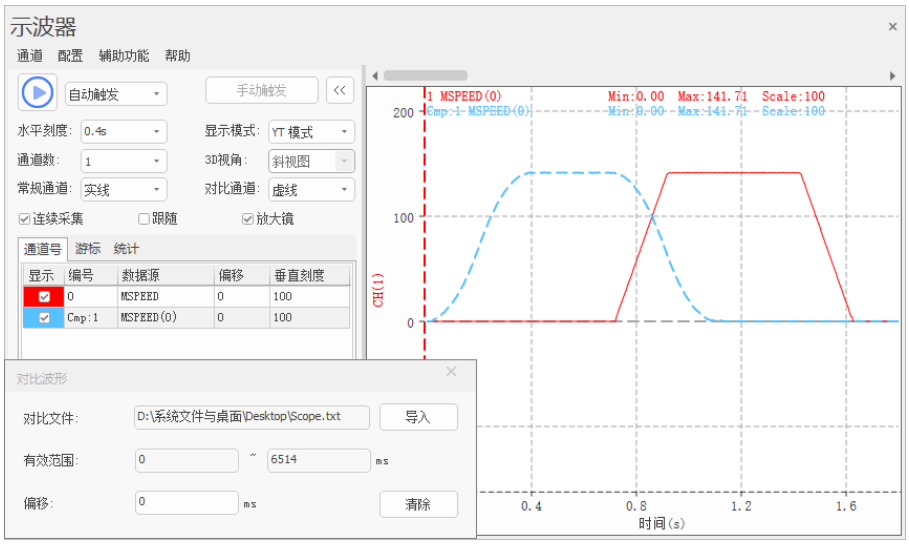
When offset of contrast waveform is -200, effective range will become 0-5000 – because effective value always is positive + max value of effective range's end value is the max value of original waveform (6000 is the max), after offset, it becomes 5800, then exceeding part is cut.

When offset of original waveform is -1200, effective range will become 0-4800 – because effective value always is positive, after offset, it moves left 1200ms, a part will not be displayed, at this time, max number of contrast waveform becomes 4800 (6000-1200), which is less than original waveform's max value.

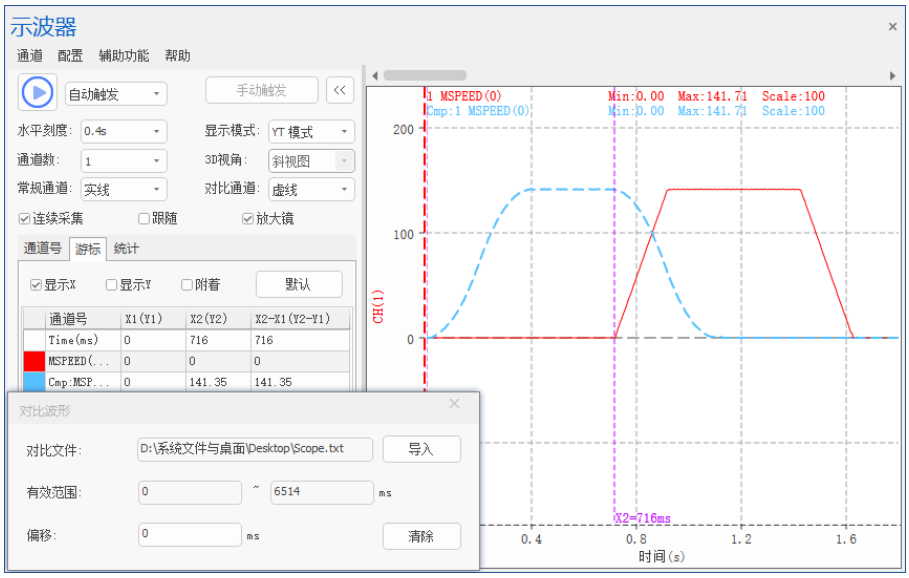
D. Clear: clear imported contrast files.

E. How to Contrast: click "import", then select corresponding data file and open it, at this time, waveform will be shown. Use cursor to measure horizontal time difference value between them, then according the difference value to set offset. Next, press "Enter", it starts to compare.

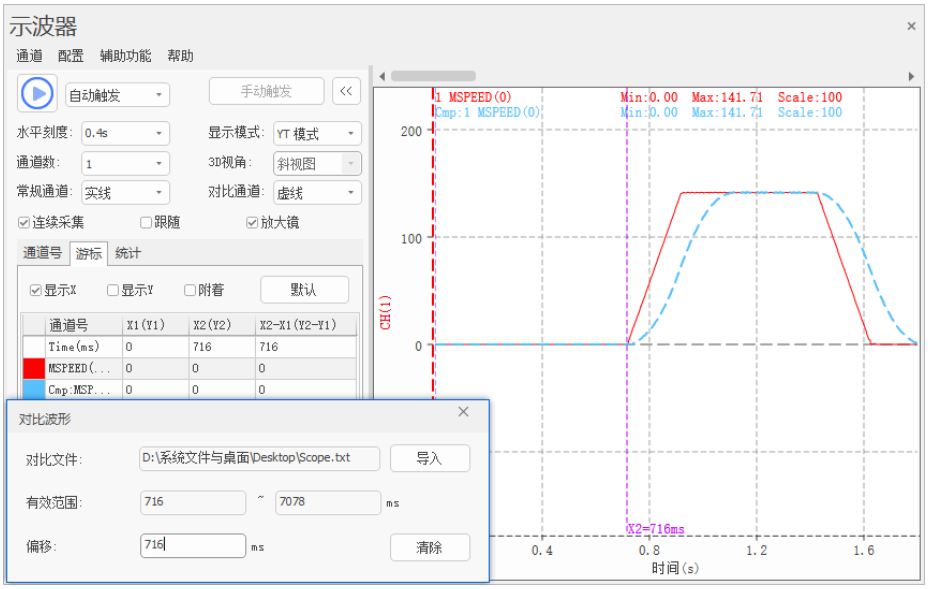
For example: original waveform is red, contrast file is blue, after contrasting, system automatically obtain effective range of contrast file.



Use the cursor tool to compare waveforms as needed to obtain the horizontal distance between them, that is, the difference between the two X cursors.



In “contrast file”, you can set offset, after setting, click “Enter”, offset contrast waveform will be shown. And at this time, it may add or reduce in effective range, please refer to above “offset”.



(3) Import / Export Waveform

A. Import Waveform

It is used to import parameters related to scope waveform, including whether to show, No., data source, offset, vertical, time interval of each point., please note waveform only can be imported when scope is stopped. And the format should be .txt file.

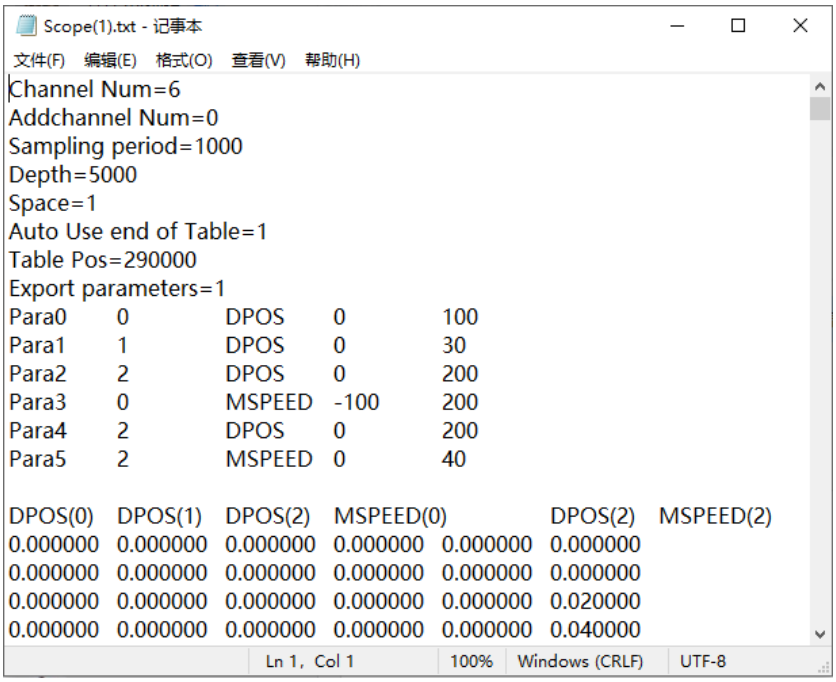
Click “accessibility” – “import waveform”, and select needed file. After that, new waveform will recover before waveform (when there are parameters in imported file, please select whether to import, if yes, it will cover, if not, current parameters are still).

B. Export Waveform

It is used to export parameters related to scope waveform, you can select the exported range (all, showing range, cursor range). Please note waveform only can be imported when scope is stopped (contrast channel is not exported). And the format should be .txt file.

Click “accessibility” – “export waveform”, and select folder to save it.

Below shows the file that is with parameters.



(4) FFT Calculation

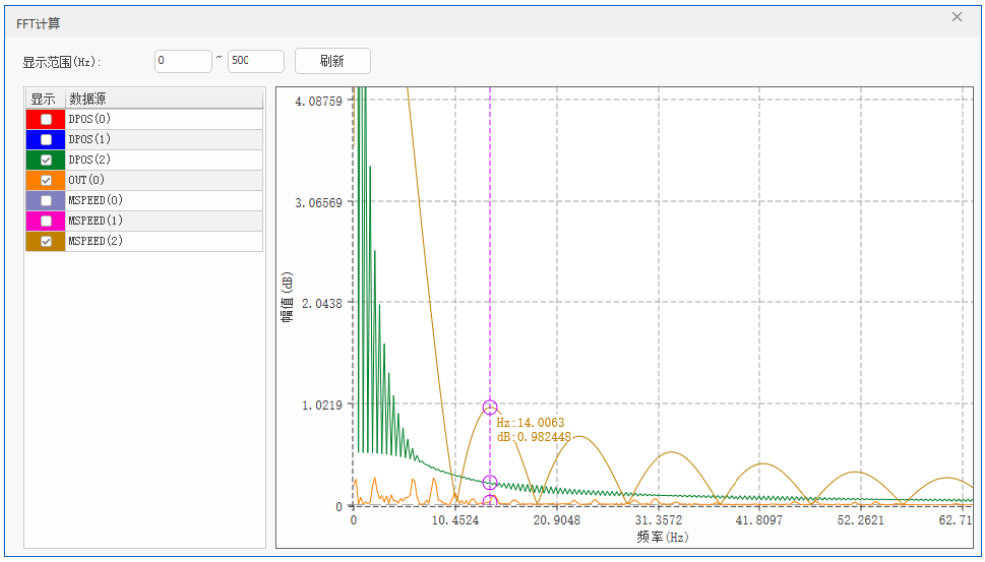
FFT is an algorithm for discrete Fourier transform, which converts the signal from the time domain into the frequency domain so that you can perform frequency domain analysis.

--How to Use--

Use X1 and X2 cursors in the observer to select one part of waveform. Then, this selected waveform will be done FFT calculation (if not open cursor, directly open FFT calculation, it will automatically show and FFT calculate the position where the X1 and X2 cursor is).

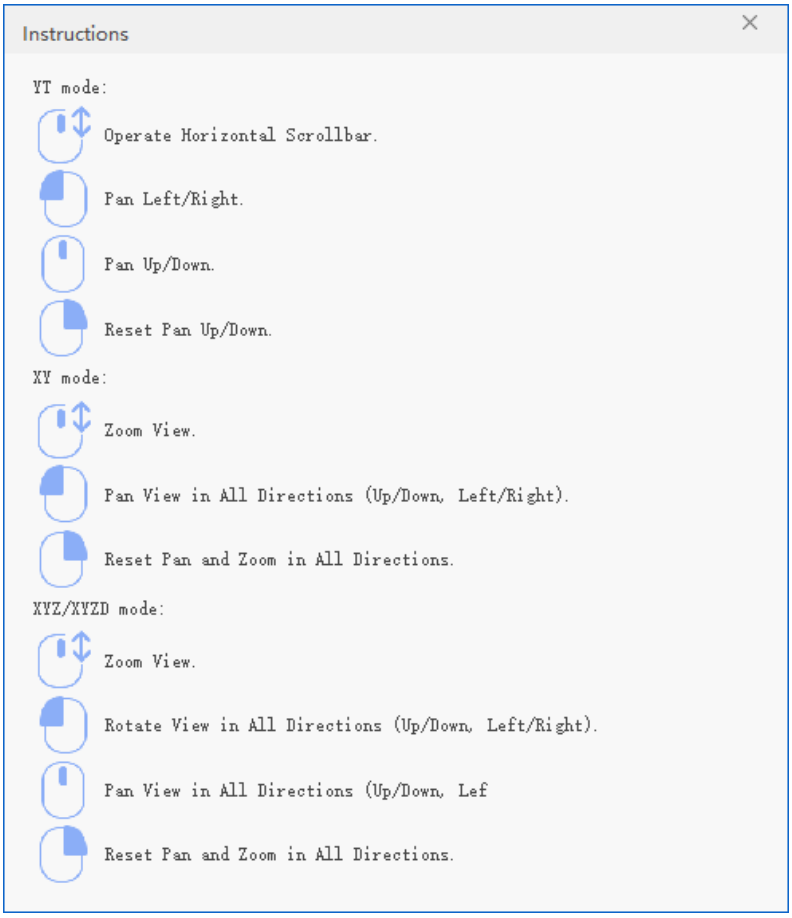
It is opened by scope "accessibility" – "FFT calculation", after that, it will show FFT calculation window and corresponding range waveform will be converted to the frequency and amplitude curve, like below shown (if the waveform is too small to be watched clearly, you can select the area and zoom in it, the usage operations is same as YT mode).

In FFT calculation window, you can choose showing range and choose data source. And when you want to know one point's value, just put the mouse near the waveform and attachment point, which can show the frequency and amplitude.



5.1.4. Help

In “help” button, it shows “how to use mouse”. In different modes, mouse usages are different.



5.1.5. How to Use Scope

(1) How to Sample by SCOPE

a. Open project, connect to controller or simulator, then open the oscilloscope window (note: before operating the oscilloscope, please connect to controller / simulator at first).

b. Click “Parameter Config” in oscilloscope window, select sampling period, max sampling period, sampling space, whether to use TABLE end data, Table position and showing type, etc. After configured, click OK to save it.

c. Select sampling Index and Source, then select auto-trigger / manual-trigger, click button. if you select condition-trigger, please see step D.

d. If you select condition-trigger, please set conditions in advance, and select trigger mode (corresponding conditions & values), then click button, download program into controller. When the set condition is met, scope starts to sample.

e. Download program into controller. When it is auto-trigger, sampling immediately after clicking button. When it is manual-trigger, click button first, then click “manual-trigger”, at last, download to RAM/ROM, or if there is “TRIGGER” command in the program, you can click and download directly to wait for BASIC to trigger sampling.

f. If the waveform accuracy is not high or the display is incomplete, click the "" button and then open the "Parameter Config", adjusting the sampling space and sampling depth, and perform the above sampling process again.

g. Notes

If the sampling time is long, start “Continuous acquisition” function. At this time, no relation between sampling time and max sampling period.

TRIGGER command can help check the waveform and find problem in an easy and flexible way.

- How to Calculate Scope Sampling Time:

For example, max period: 1000, space: 5

If system cycle SERVO_PERIOD=1000, which means it is 1ms trajectory planning cycle. Space 5 means sampling one data point per 5ms. Total sampling data number is 10000, so sampling time length is 50s.

- How to Calculate TABLE End Space:

Set the position where the captured data is stored. Generally, the space at the end of the TABLE data is automatically used by default, now starting space address is calculated automatically according to captured data space.

Calculation method: captured data space = channel numbers * max sampling periods

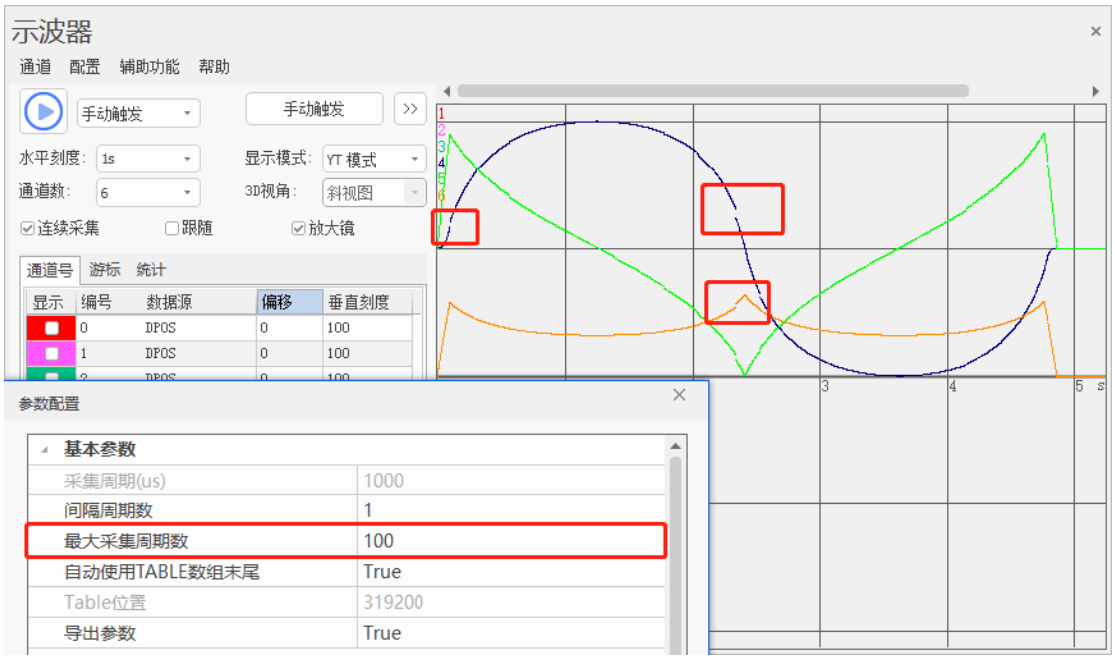
For example, if TABLE space of controller is 320000, there are 4 sampling channels, max sampling periods is 30000, each sampling point occupies one TABLE, so it will occupy 4*30000=120000 TABLE positions. 320000-120000=200000, which means starting position of TABLE is 200000.

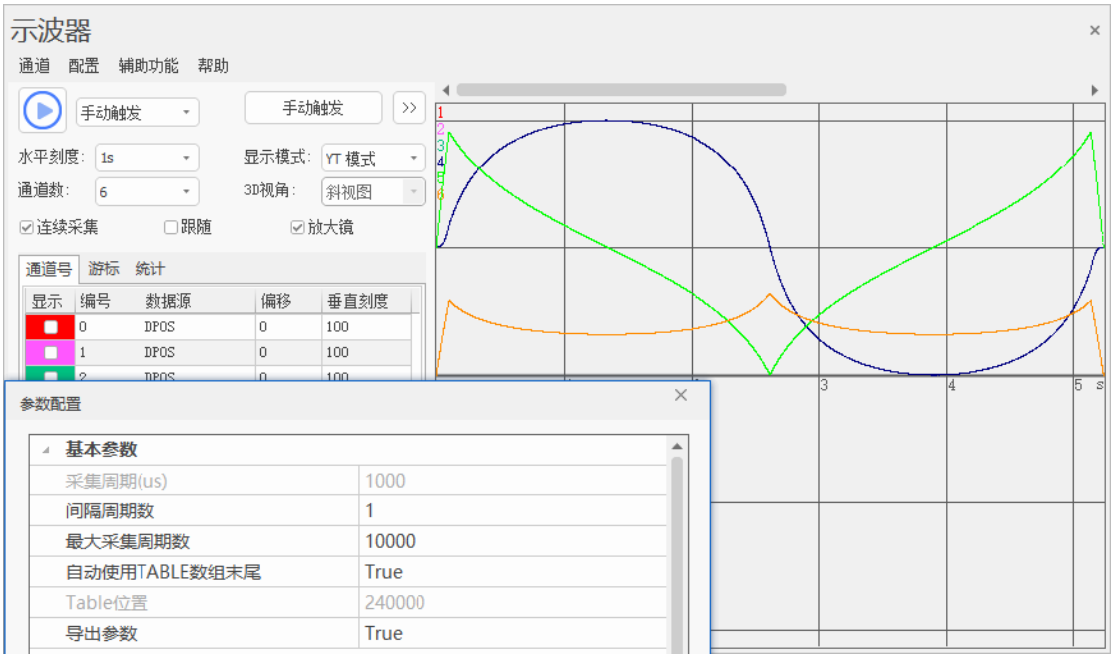
If you don't use TABLE end space, you also can self-define. Same condition as above, starting TABLE position can't be more than 200000, because this space can't be same as TABLE space used in program, otherwise, no way to run.

- How to Solve "Point Loss" Problem:

Generally, the "max sampling periods" is too low, "point loss" may appear. Then, you can set a bigger value.

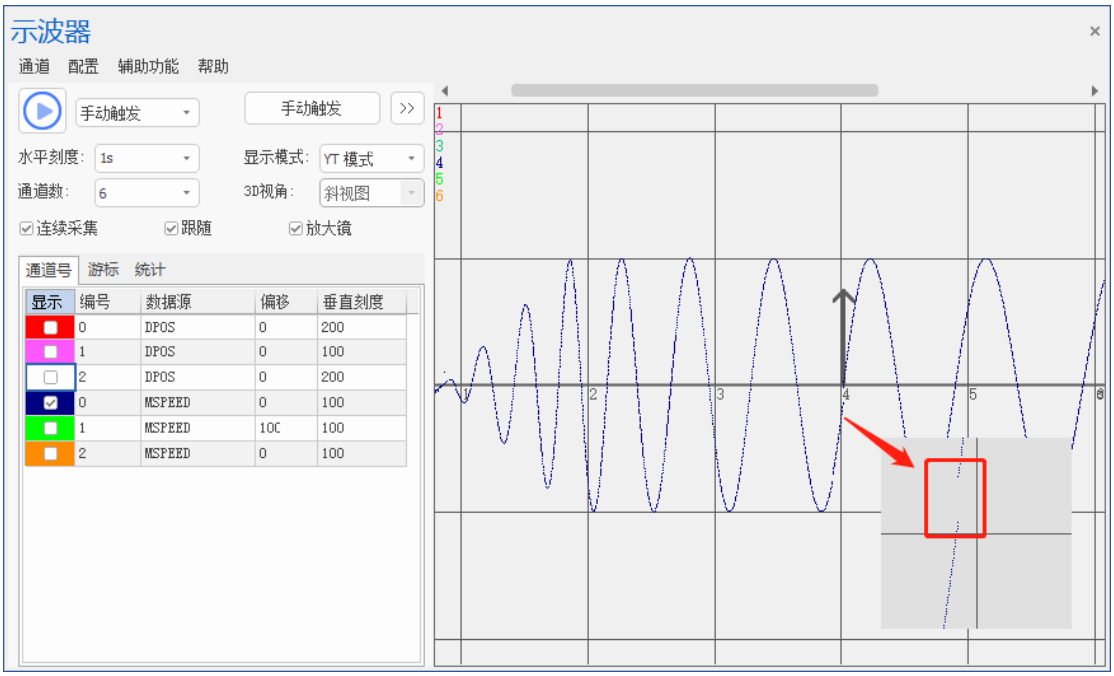
For example, when it is 100, it may lose data, but when it is 1000, it may not occur point loss. Please check their waveform:



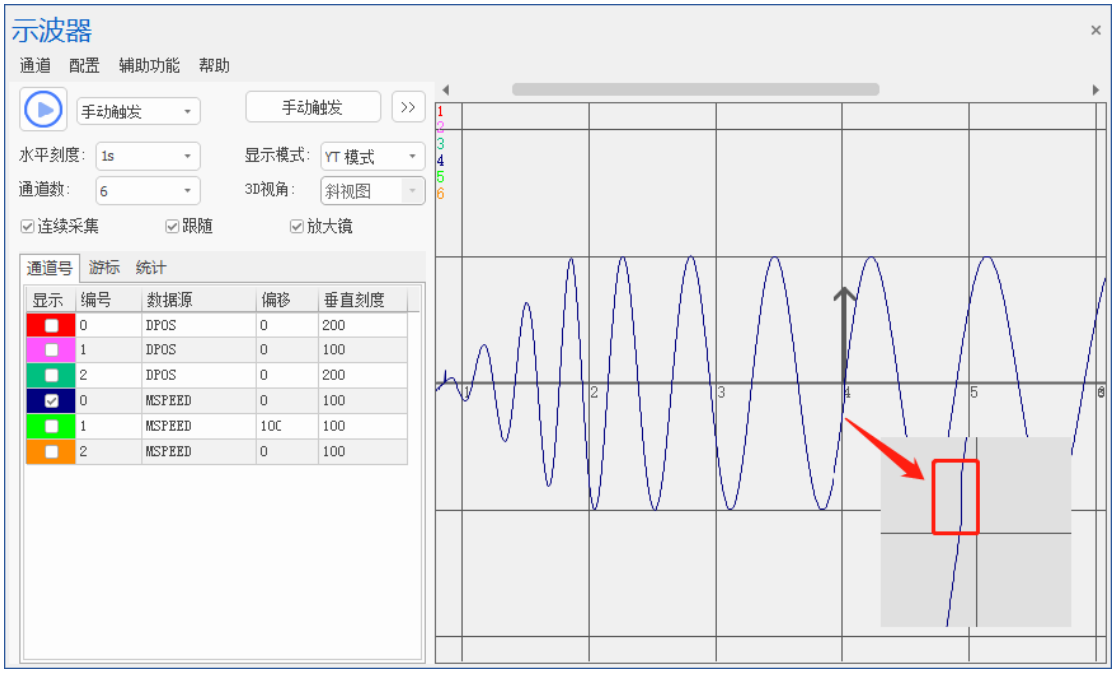


- How to Solve “Polyline” under “Continuous Acquisition”:

Related to “max sampling periods”. Actually, the problem is “point loss”. Because it will trigger again when at max period 80% after opening the continuous acquisition.



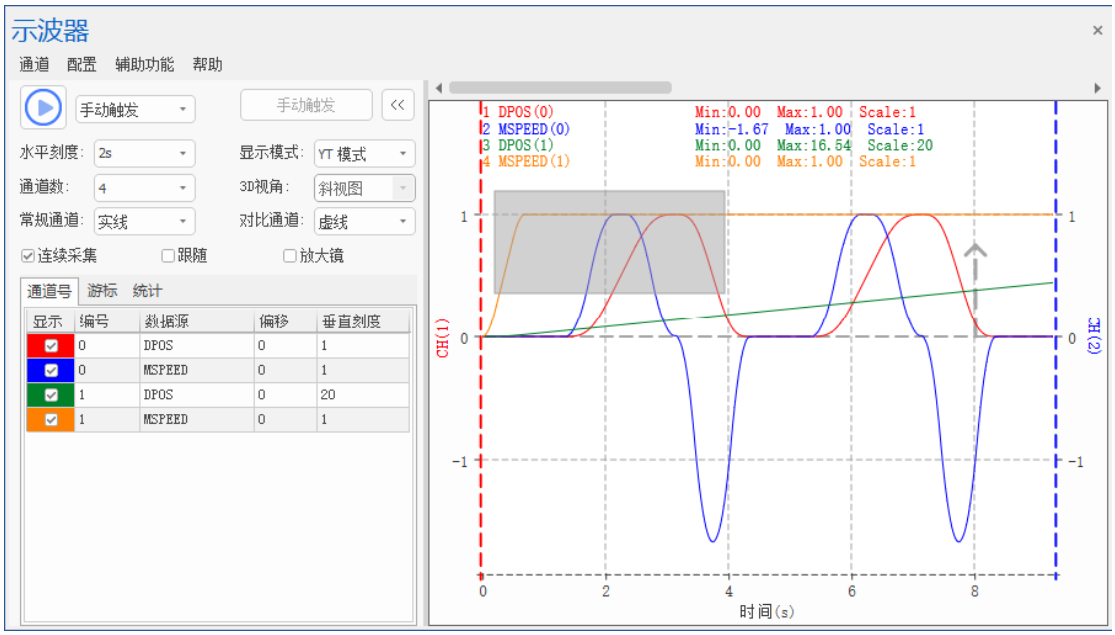
When the line type is solid type, it will appear “Polyline”.



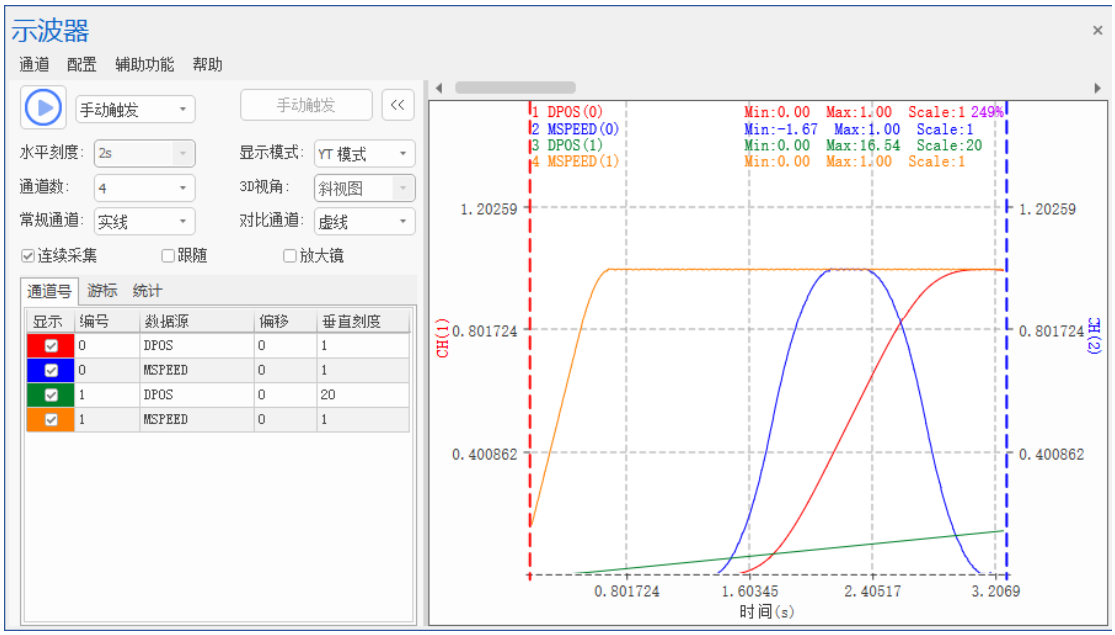
(2) How to Zoom in Waveform Locally under YT Mode

Under YT mode, press “Shift + mouse left” to select the area you want to scale in, the multiple is calculated automatically according to selected range. After that, the observer’s vertical and horizontal vertical will be changed also. If you want to resume it, right click in showing area.

--Select the Area--



--After Zooming in--



(3) Scope Usage Routine

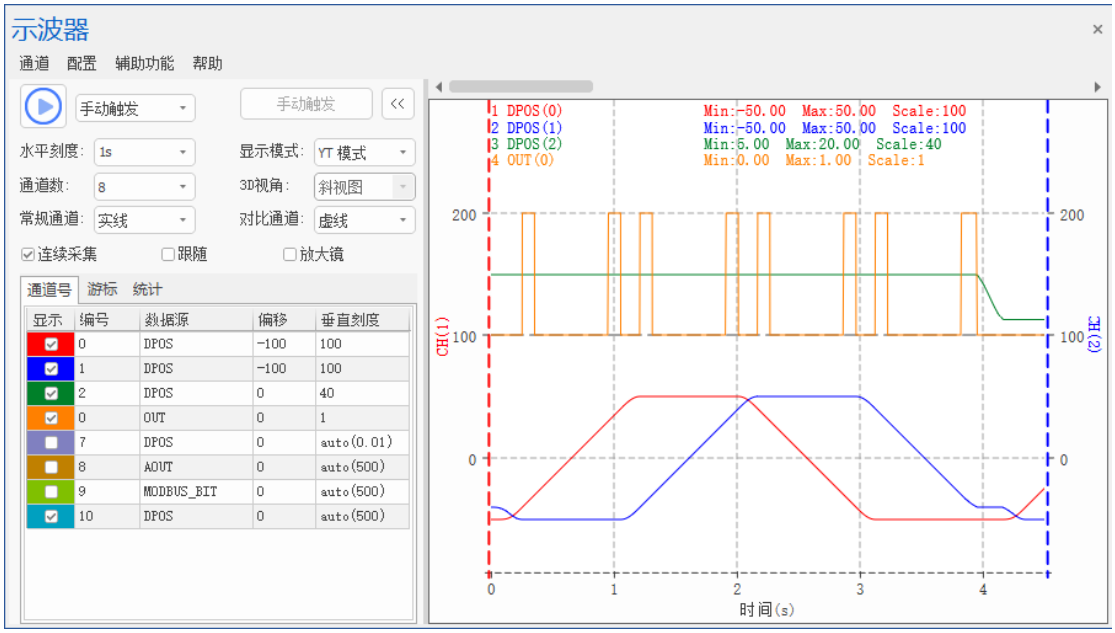
For example, the routine of "3-axis interpolation".

```
BASE(0,1,2)
ATYPE = 1, 1, 1
UNITS = 100, 100, 100
DPOS = -50, -40, 20
SPEED=100, 100, 100
ACCEL=1000, 1000, 1000
DECEL=1000, 1000, 1000
SRAMP=100,100,100

MERGE=ON
CORNER_MODE=2,2,2
DECEL_ANGLE = 15 * (PI/180)     'set starting deceleration angle
STOP_ANGLE = 180 * (PI/180)     'set end deceleration angle
FORCE_SPEED=100                 'take effect during geometric deceleration

TRIGGER
```

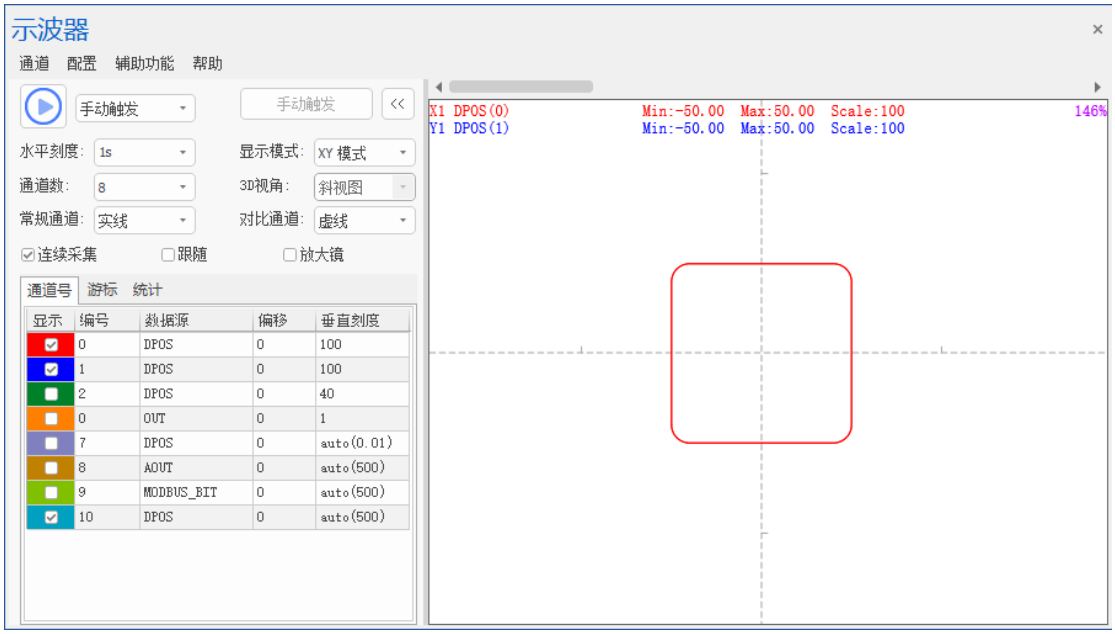
```
DPOS=-50,-40,20


MOVEABS(-50, -40, 20)
MOVECIRCABS(-40, -50, -40, -40, 0)
MOVE_OP(0, ON)
MOVEABS(-30, -50, 20)
MOVE_OP(0, OFF)
MOVEABS(30, -50, 20)
MOVE_OP(0, ON)
MOVEABS(40, -50, 20)
MOVE_OP(0, OFF)
MOVECIRCABS(50, -40, 40, -40, 0)
MOVE_OP(0, ON)
MOVEABS(50, -30, 20)
MOVE_OP(0, OFF)
MOVEABS(50, 30, 20)
MOVE_OP(0, ON)
MOVEABS(50, 40, 20)
MOVE_OP(0, OFF)
MOVECIRCABS(40, 50, 40, 40, 0)
MOVE_OP(0, ON)
MOVEABS(30, 50, 20)
MOVE_OP(0, OFF)
MOVEABS(-30, 50, 20)
MOVE_OP(0, ON)
MOVEABS(-40, 50, 20)
MOVE_OP(0, OFF)
MOVECIRCABS(-50, 40, -40, 40, 0)
MOVE_OP(0, ON)
MOVEABS(-50, 30, 20)
MOVE_OP(0, OFF)
MOVEABS(-50, -30, 20)
MOVE_OP(0, ON)
MOVEABS(-50, -40, 20)
MOVE_OP(0, OFF)
```

```
MOVEABS(-50, -40, 5)
MOVEABS(-50, -40, 5)
MOVECIRCABS(-40, -50, -40, -40, 0)
MOVEABS(40, -50, 5)
MOVECIRCABS(50, -40, 40, -40, 0)
MOVEABS(50, 40, 5)
MOVECIRCABS(40, 50, 40, 40, 0)
MOVEABS(-40, 50, 5)
MOVECIRCABS(-50, 40, -40, 40, 0)
MOVEABS(-50, -40, 5)

END
```

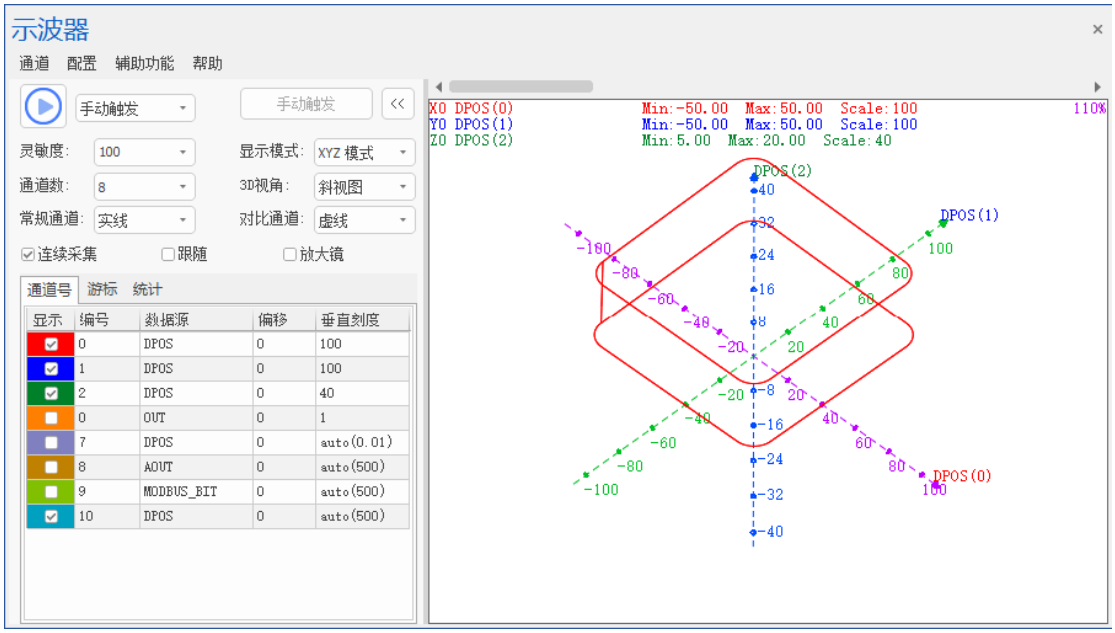
--below shows waveforms of position and speed of axis 0, axis 1, axis 2--



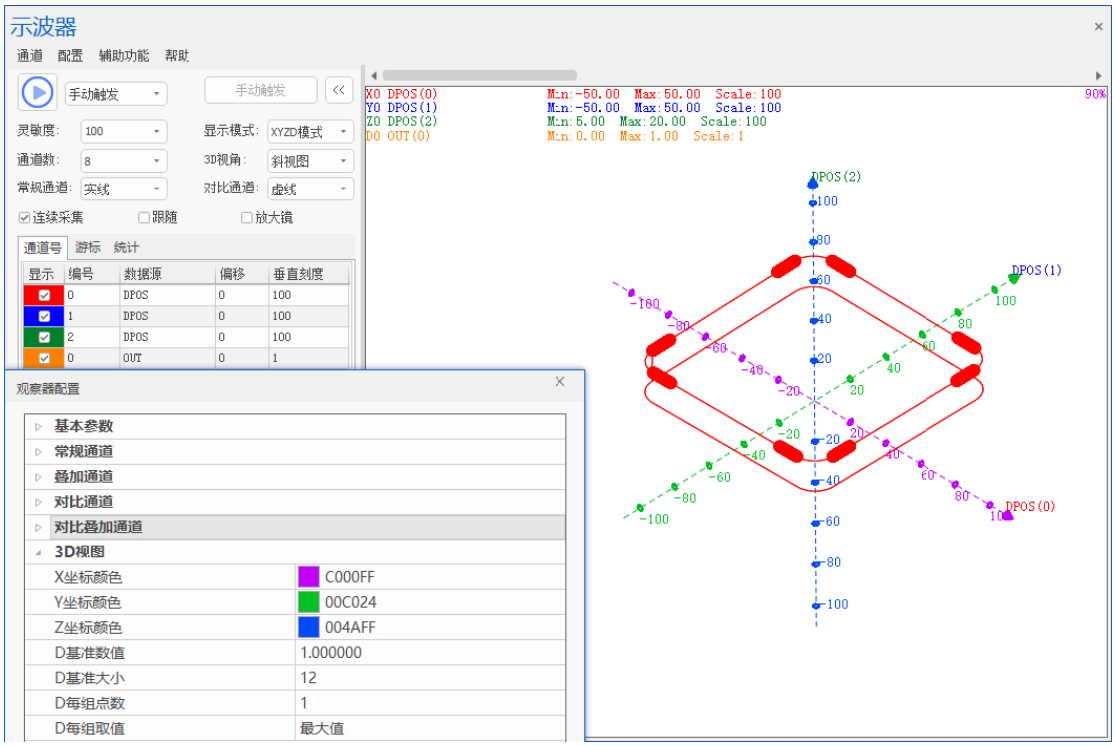
--two axes interpolated merged trajectory under XY mode--



--three axes interpolated merged trajectory under XYZ mode--



--four axes merged trajectory under XYZD mode, and the diameter of the point can be modified by modifying the D reference value and D reference size in the "Observer Config" window. When the channel line type is set to point, the oscilloscope only displays dots. When the channel line type is set to solid or dashed, the oscilloscope will use solid or dotted lines to connect the dots. The oscilloscope display and observer configuration are shown in the figure below--



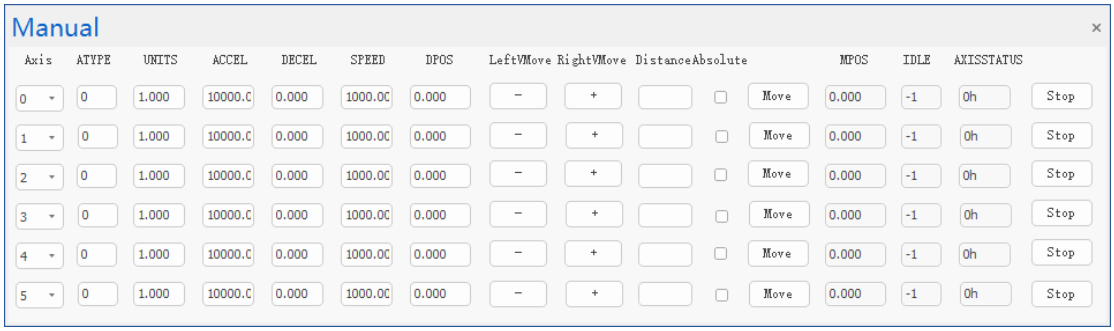
5.2. Manual Motion

"Manual" is used to operate the motor manually. It can be opened through the menu bar "Tool" → "Manual".

--How to Operate it--

Connect to controller and motor, then open this tool “manual”. You can enter/modify the axis-related parameters in real time on the left, select the axis No. (can be selected in the drop-down list), and after setting the relevant parameters, press and hold the "LeftMove"/"RightMove" button, the motor will continue to move left or right, and release the button to stop the movement. "DPOS" displays the current demand position’s movement distance (in units). Fill in the "Distance" parameter and click "Move". When "Absolute" is checked, the motor moves to the absolute distance parameter position. When "Absolute" is not checked, click "Move" and the motor moves according to the relative distance parameter.

"MPOS"/"IDLE"/"AXISSTATUS" (feedback position / motion state / axis state) are used to monitor the feedback axis movement status. And these three parameters are read-only and cannot be modified. Press the "Stop" button to stop the axis movement immediately.



5.3. IN

Used to detect the state changes of the controller input port in real time. The input state of this window changes with the state of the controller IN. Open it through the menu bar "Tool" → "IN".

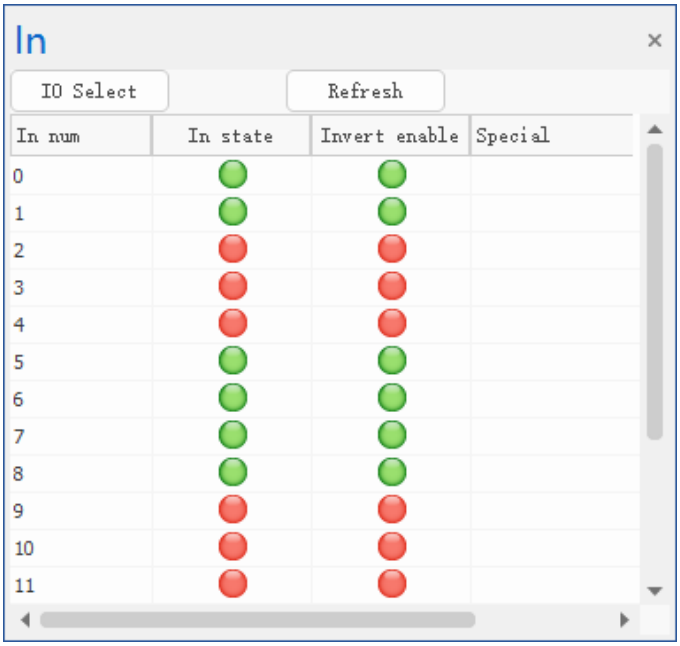
Use the INVERT_IN command in the program to invert input of a certain IN (the special input of the ZMC series needs to be reversed after the signal is defined, because for ZMC series, OFF is valid, and the ECI series does not need to be reversed). After setting the input inversion, the "Invert Enable" column is a constant green light, and if there is no inversion, it is dark red. At this time, when there is an input signal at the input port, the "In State" is gray, and if there is no input at the input port, it is green.

"Special Input" is used to display special signal prompts such as origin, limit, and alarm.

For Example: JOG Motion

```
RAPIDSTOP(2)
WAIT IDLE(0)

BASE(0)
DPOS = 0
ATYPE = 1
UNITS = 100
SPEED = 100
ACCEL = 1000
DECEL = 1000
CREEP=10
```

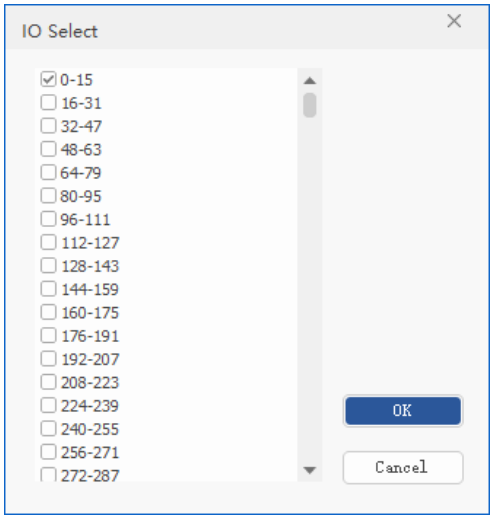
```
DATUM_IN = 5          'IN5 as origin switch
INVERT_IN(5,ON)    'invert IN5 electric level, common-ON signals to invert (ZMC Controller)
FWD_IN = 6             'set forward position limit switch
INVERT_IN(6,ON)     'invert the signal
REV_IN = 7             'set inverse position limit switch
INVERT_IN(7,ON)    'invert the signal
ALM_IN = 8            'set alarm signal
INVERT_IN(8,ON)    'invert the signal

JOGSPEED = 50       'JOG speed is 50
FWD_JOG = 0          'IN0 as forward JOG switch
REV_JOG = 1          'IN1 as inverse JOG switch
INVERT_IN(0,ON)    'invert the signal
INVERT_IN(1,ON)

DATUM(3)               'homing mode
```



For above, IN0, IN1, IN5, IN6, IN7, IN8 are enabled “Invert” (for ZMC, invert is valid when IN is off, for ECI, it is opposite). If there is on IN signal for IN5, after inverted, the led will be off.

“IO Select”: you can select IN to be shown, 16 inputs as one group.

“Refresh”: refresh each IN state and special IN.

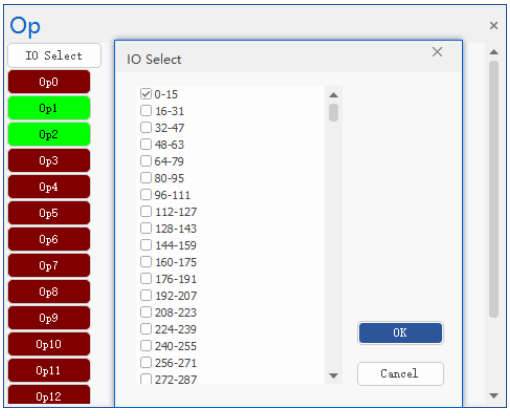


5.4. Op (OUT)

Used to watch controller OUT state, then operate it. Open through "Tool" → "Op". You can also view the status of the OUT port through the simulator, but only the status of numbers 0-11 can be displayed.

--How to Operate it--

Press the button to operate the OP port output, as shown in the figure below, OP1 and OP2 ports are turned on, and other ports are turned off. "IO Selection" selects the output to be displayed, each group of 16 outputs. Clicking the output port directly can switch its output status.



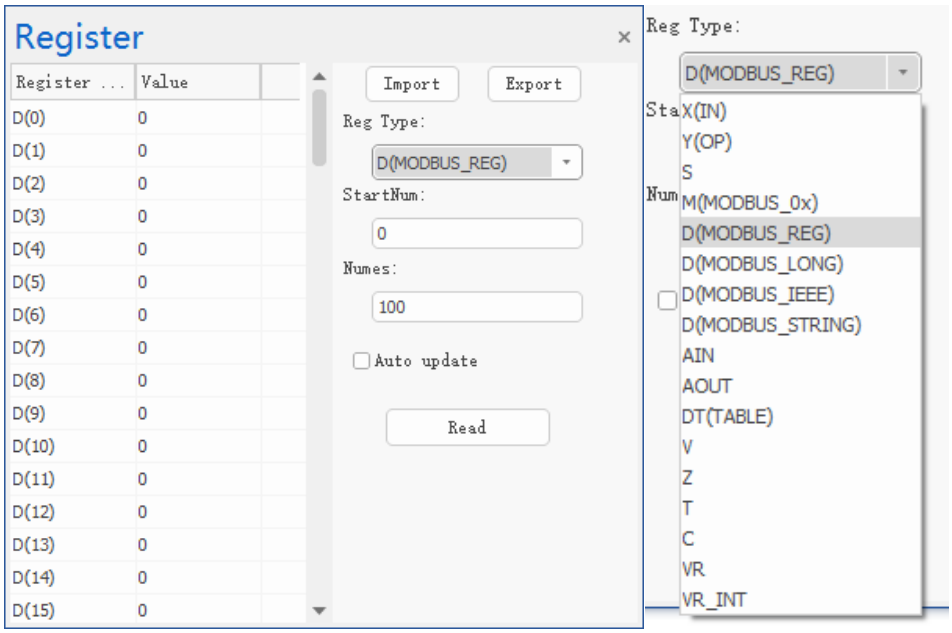
5.5. Register

Used to view the values of controller registers in batches. You can choose to view different types of registers (only controllers that support PLC functions support this function). This window can be opened through the menu bar "Tool" → "Register".

--How to Use--

Select the type, starting No., and number of registers to be read, and then click "Read" to display the data in the window. Check the automatic refresh function to automatically collect and display the changes in register values in real time. Otherwise, you need to click Read again to obtain the changes in the register values.

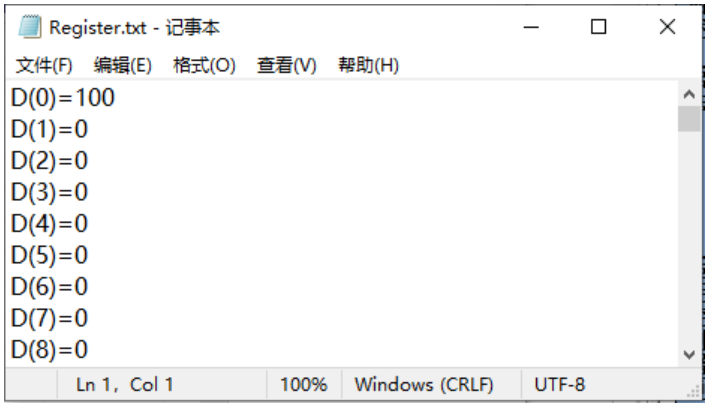
Note: Do not read the number of registers beyond the register range, otherwise an error will be prompted.



Register Name	Register Type
X (IN)	Input relay, driven by external signal, related to IN.
Y(OP)	Output relay, it can drive external load directly, related to OP.
S	state relay, used to control stepper.
M	Auxiliary relay, can't drive external load directly, related to MODBUS_BIT.
D(MODBUS_REG)	16-bit integer data register, MODBUS area data, related to MODBUS_REG.

D(MODBUS_LONG)	32-bit integer data register, MODBUS area data, related to MODBUS_LONG.
D(MODBUS_IEEE)	32-bit floating data register, MODBUS area data, related to MODBUS_IEEE.
D(MODBUS_STRING)	1 byte character string data register, MODBUS area data, related to MODBUS_STRING.
AIN	Analog input
AOUT	Analog output
DT(TABLE)	32-bit floating register, related to TABLE.
V	Index register, 16-bit.
Z	Index register, 16-bit.
T	Time, unit is ms.
C	Counter
VR	Power failure storage register, 32-bit floating type.
VR_INT	Power failure storage register, 32-bit integer type.

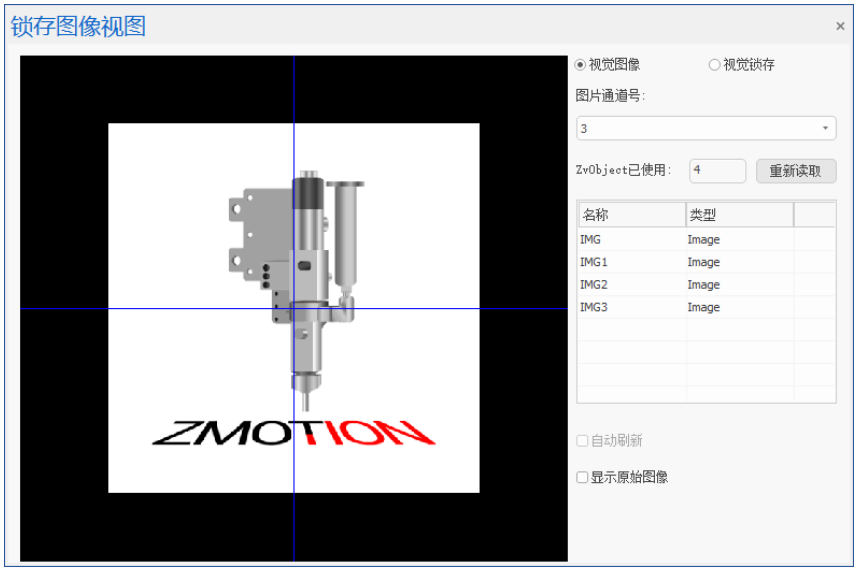
“Import” / “Export”: it can rapidly upload / download register data. Example of exporting is shown as below:



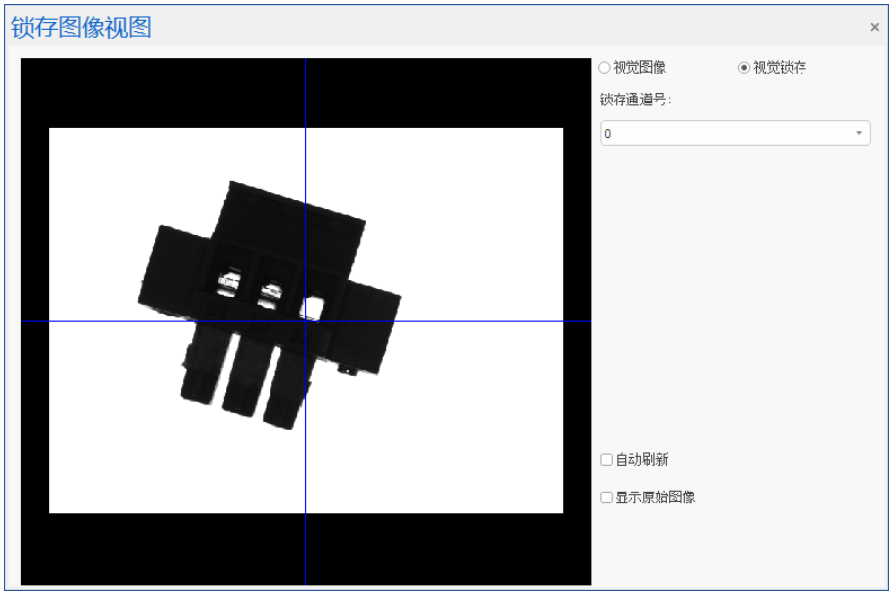
5.6. Latch Image

In the machine vision development environment, “Image View” (Tool – Image View) window can check the image. Please note it must capture the graphic, then the window can be used normally. Therefore, you can switch vision latch and vision image when all images are shown.

➢ Visual Image: display the image in the controller through visual instructions, including the acquired image and the processed image. And the image can be saved to the latched channel. If there is already an image in the latched channel, the original image will be replaced.



➢ Vision Latch: show image that is in latch channel. After selecting latch channel No., it will show the image of current latch channel. When the latch channel is empty, no image here.



Example of Getting Image from Latch Channel:

Below uses simulator, you need to put the image to simulator flash folder, then it can be read. If you use the controller, also, save the image into controller flash folder.

```
ZVOBJECT img,img1,img2,img3 'define variables
ZV_READIMAGE(img,"0.bmp",0) 'read the image in the format of original image
```

```
ZV_LATCH(img,0,0) 'show image of channel 0

ZV_READIMAGE(img1,"1.bmp",0) 'read the image in the format of original image
ZV_LATCH(img1,1,0) 'show image of channel 1

ZV_READIMAGE(img2,"16wei-02.bmp",0) 'read the image in the format of original image
ZV_LATCH(img2,2,0) 'show image of channel 2

ZV_READIMAGE(img3,"2.bmp",0) 'read the image in the format of original image
ZV_LATCH(img3,3,1) 'show image of channel 3
END
```

5.7. AD/DA

Used to view changes in analog input and analog output and corresponding scale values. This tool can be opened through the menu bar "Tool" → "AD/DA". (Note: To use this tool, the controller must support analog input and output functions)

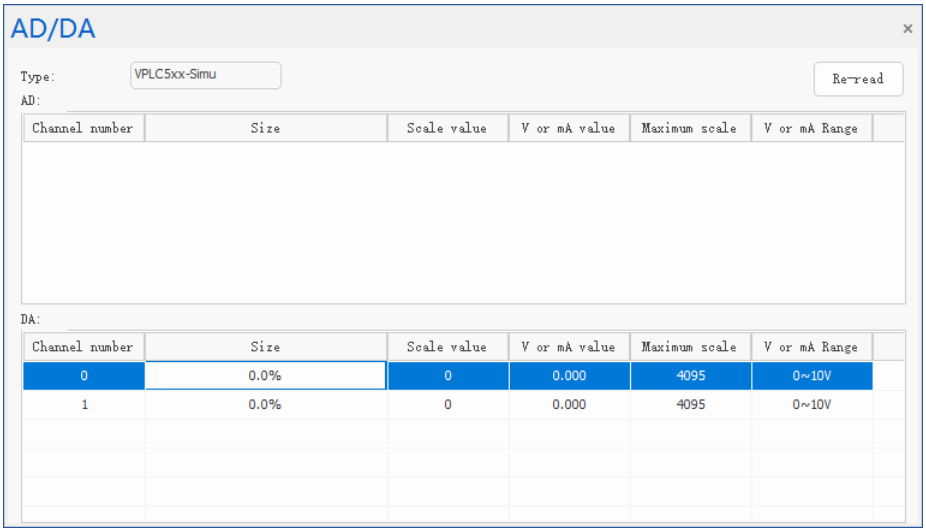
(1) How to Operate

Connect a controller that supports analog input/output, open the "AD/DA" tool window, and click "Re-read" to read the analog value of the current controller.

(2) Parameters Introduction:

- Channel number.: display the input and output ports corresponding to the AD/DA of the currently connected controller.
- Size: show AD/DA size of each channel port, displayed as a percentage.
- Scale value: show the analog scale value that has been input or output. (The "scale value" and "v or mA value" of DA can be modified.)
- V or mA value: display the value within the selected "voltage or current range".
- Maximum scale: depend on whether the resolution of the controller analog is 12 bits or 16 bits. For details, please refer to the corresponding user manual. For 12-bit, the corresponding scale value range is 0~4095. For 16-bit, the corresponding scale value range is 0~65535.

- V or mA range: voltage or current range, select the range according to the requirements and the range supported by the controller.

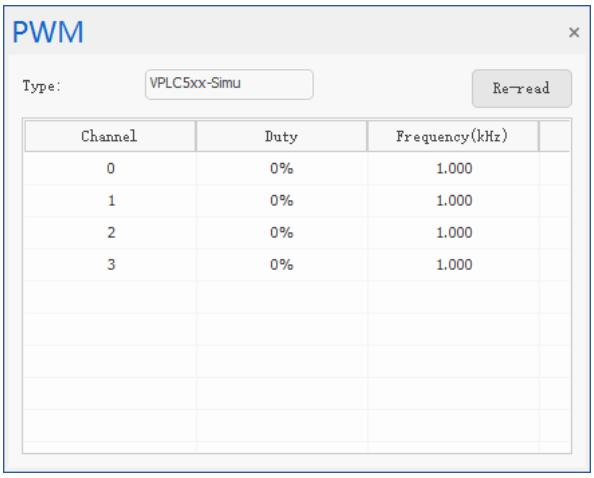


5.8. PWM

Read / set controller's PWM duty cycle, frequency and corresponding parameters. Different controllers are with different PWM channels, and numbers will be read automatically.

--How to Operate--

Connect the controller that supports PWM, then open "PWM" window (Tool – PWM), click re-read to get the parameter. At the same time, you can directly set parameter data in this tool.



5.9. SDO

Used to write/read EtherCAT data dictionary to the controller. This tool can be opened through the menu bar "Tool" → "SDO".

(1) How to Operate

a. Select Mode

Read / Write the data dictionary of the device according to the "Device and slot mode"/"Axis mode".

b. Select Device

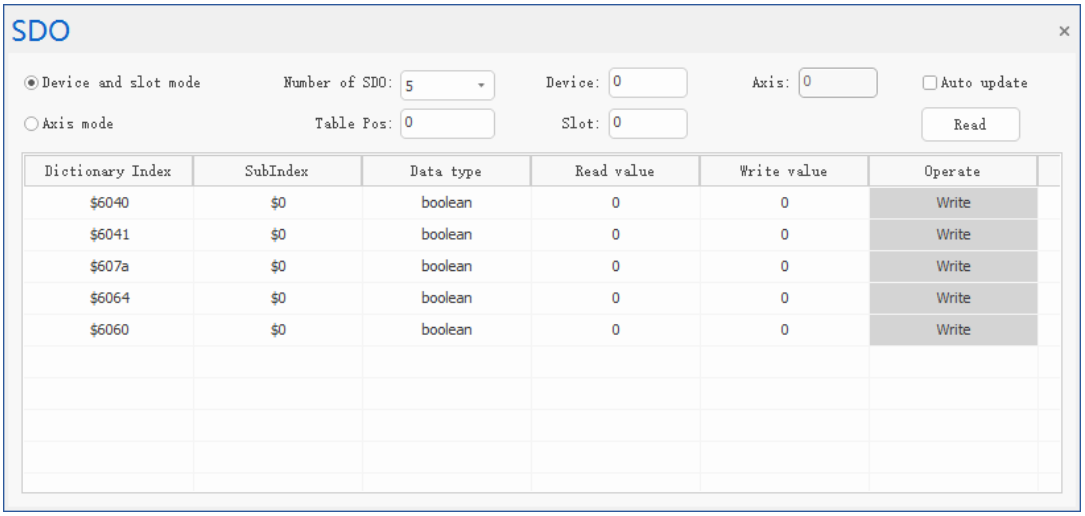
Select which device you need to read or write. If you selected “Device and slot mode” in step 1, please fill in device and slot. If you selected “Axis mode”, please fill in axis.

c. Set Data Dictionary

Select the "Number of SDO" to be read/written according to actual needs, and manually add the correct Dictionary index, SubIndex, Data type and Write value and other related parameters (please refer to the drive EtherCAT communication manual for details), click "Write", if the parameter setting is incorrect, the corresponding error code will be returned.

d. Read Data Dictionary

Set the Table Position where the data needs to be stored after reading, and then print the corresponding Table location to obtain the data. You can also add the data dictionary index and correct related parameters to be queried in this tool, and click "Read" to display it in the "Read value" column.



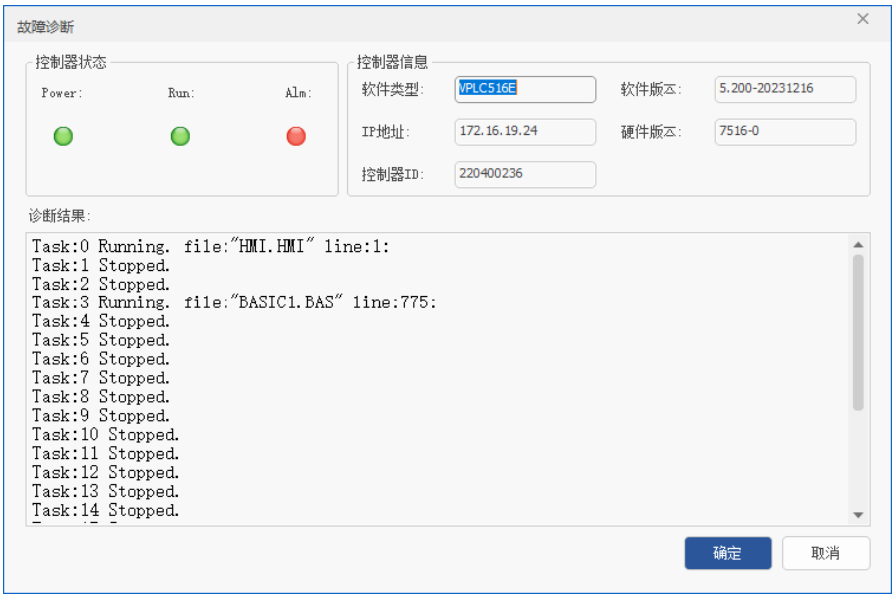
5.10. Troubleshooting

Used to check controller state and trouble details. Click Tool – Troubleshooting.

In this tool, you can see controller model, date, version No., task running state, and error information. Then you can know what is the trouble and program stopped position clearly.

In addition, LED Run and Alm can be opened and closed manually. In this way, you can find current connected controller easily when there are several controllers.

(1) Task Running State:



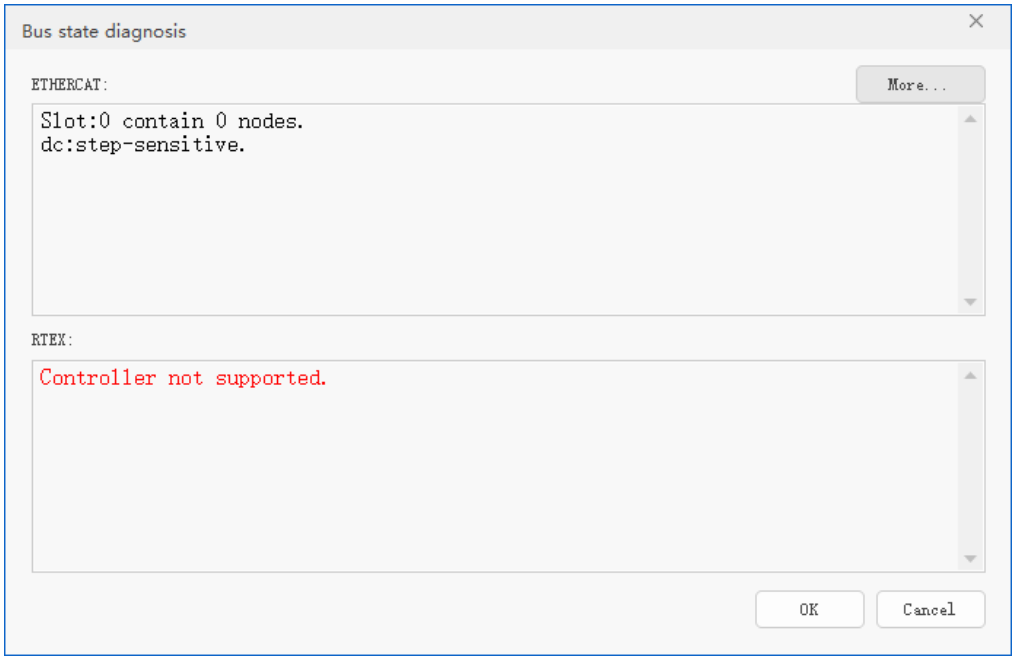
(2) Error Information:

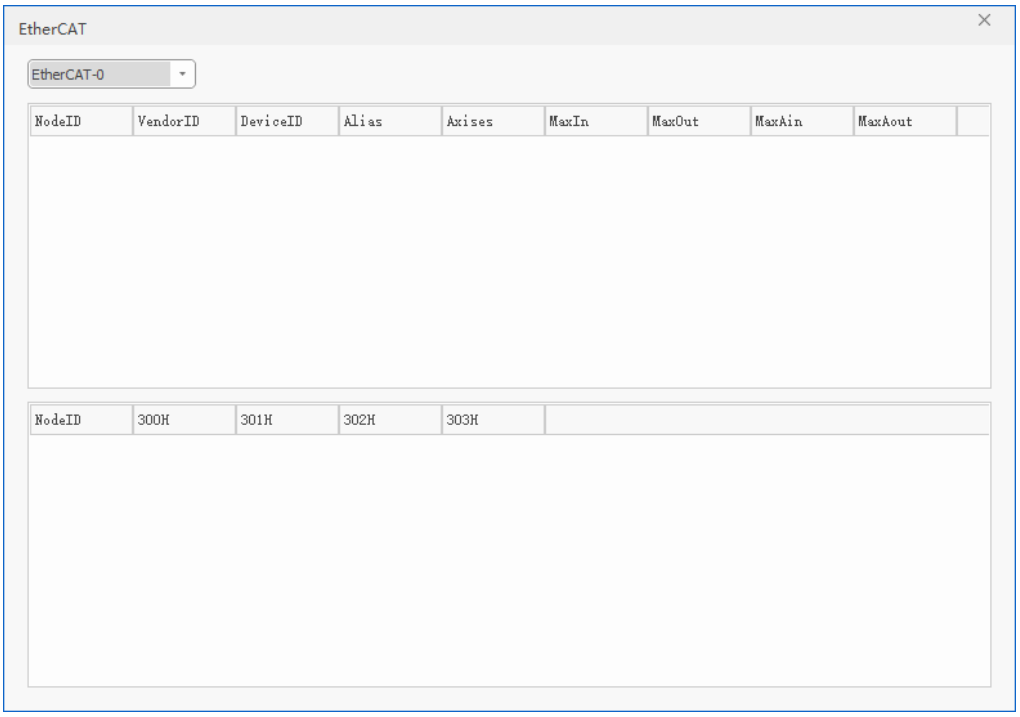
5.11. Bus State Diagnosis

Used to check controller supported bus, including all nodes devices information. Click tool – bus state diagnosis.

For output information in this command, please refer to “?*EETHERCAT” command for details.

“More…”: you can check device basic information connected on the bus, and package loss.



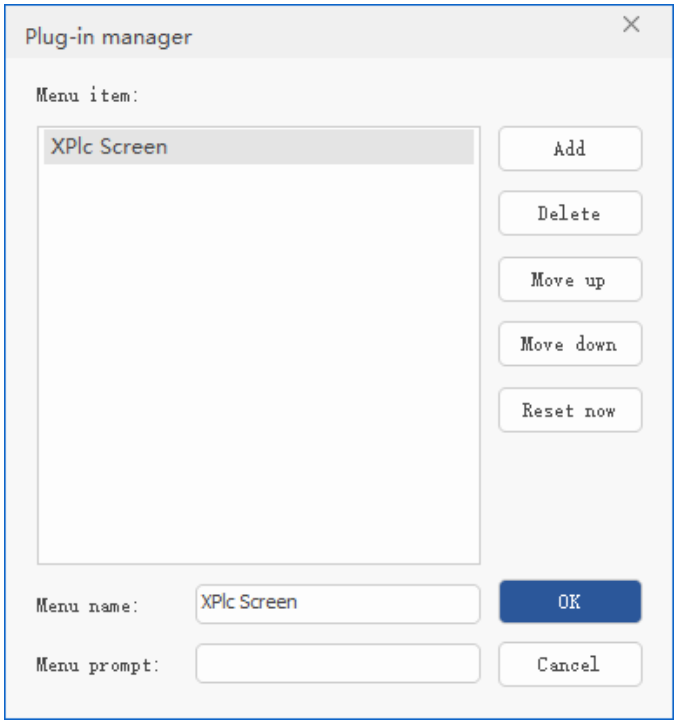


5.12. Plug-in

This is new added function for RTSys, “Plug-in”. Specifically, you can add small software and tools, formats can be .exe, ,zpl, and .bat. In this tool, you also can add / delete plug-in, sort the sequence (up / down), modify name, etc., open it by “tool” – “plug-in”. [for add “custom” one (format is .zpl), please refer to “VC Plug-in Tutorial”.

(1) Plug-in Manager

- **Add:** add plug-in program (.exe, .zpl, ,bat)
- **Delete:** delete selected plug-in
- **Move up / down:** sort selected plug-in
- **Reset now:** reset plug-in with .zpl format as initial state (for example, xplc screen will automatically connects to controller when controller is connected to RTSys, if RTSys connects to other controllers, now, xplc screen can’t refresh in time, you can use this “reset now” function).
- **Menu name:** set the plug-in name.
- **Menu prompt:** When the mouse points to a plug-in in the plug-in drop-down menu, the corresponding text prompt is displayed.



“xplc screen” is inside RTSys by default, which is HMI simulation tool.

(2) Add the Plug-in Made by Yourself

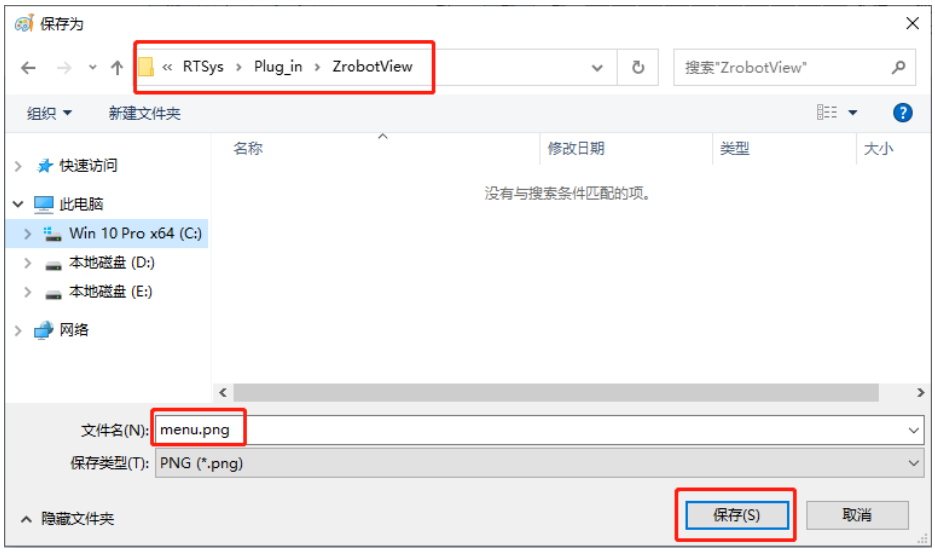
At first, own-made plug-in is .zpl program file, and it must be put one empty folder separately, then save corresponding configuration in this folder, at last, add into RTSys – plug-in.

- Open plug-in manager, then click "Add", below window will appear, then double click the file to add it into here, click OK.
- After that, new added plug-in will be shown, but please note, the icon can't be seen, unless you added corresponding file into RTSys – Plug_in folder.

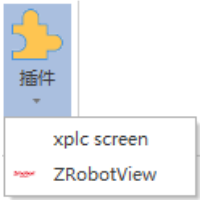
--How to Add Plug-in Icon--

a. Determine the image, and the pixel should be 16x16.

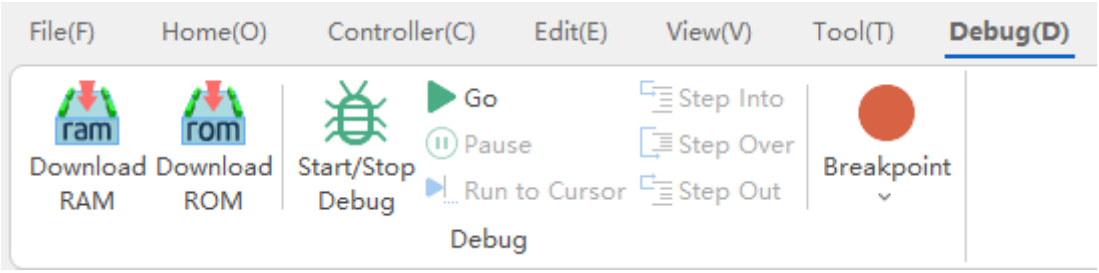
b. Save it in .png format, and rename it as menu. Then save it into RTSys \ Plug_in folder \ corresponding plug-in. For example, if the plug-in you made is "ZRobotView" (Note: it must be .png, and the name must be menu).



c. after saving, return to RTSys – plug-in manager, click OK again. At this time, drop-down the menu, new made plug-in and the icon are shown.



Chapter VI Program Debugging

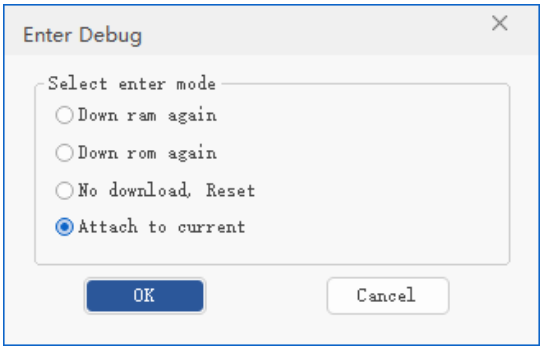


--how to operate quickly--

Name	Image Mark	Description
Start/Stop Debug		Start / stop debugging program and task.
Go		Run the program that has already opened “debug”.
Pause		Pause the program that is debugging.
Run to Cursor		Set the program run to which line.
Step Into		Jump to next command.
Step Over		Jump over next command.
Step Out		Jump out SUB subprogram.
Breakpoint		Add / delete breakpoint in Basic program.
Emergency Stop		Stop all axes’ motions.

6.1. Start / Stop Debug

The debugging function is used to track the program running. You can enter the debugging mode from the debugging menu and choose different entry methods. When you need to view the running status of the current controller program, please select “Attach to current”. After that, the file is in read-only mode by default. If you need to modify the program during debugging, you need to exit the debugging mode. Enter the program debugging state through the menu bar "Debug" → "Start/Stop Debug".



A. Down ram again: download project to ram again and start to debug, not saved when power off.

B. Down rom again: download project to rom again and start to debug, support power failure storage.

C. No download, Reset: it will not download the program, run the program that is downloaded again, and task window will be opened to show current operation status.

D. Attach to current: it will not download the program, only open the task window to show current operation status.

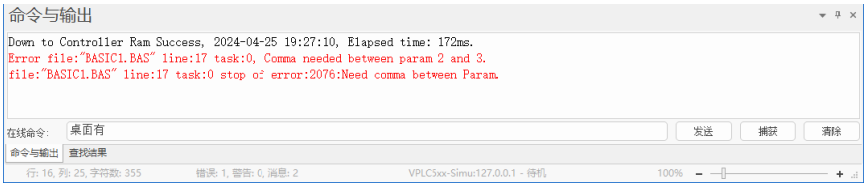
When errors come in operating project, RTSys will show error information, double-click the error information, it will locate program position, which means take it for reference to help debugging. If there is no error information showing, we can also view errors by typing in command:?*task, or open “Troubleshooting” window,.

For Example:

```
RAPIDSTOP(2)
WAIT IDLE(0)
WAIT IDLE(1)

BASE(0,1)                ‘select axis 0 and axis 1
ATYPE=1,1
UNITS=100,100
SPEED=100,100            'motion speed
ACCEL=1000,1000
DECEL=1000,1000
SRAMP=100,100            'S curve
MERGE=ON                 'open continuous interpolation
TRIGGER                  'trigger oscilloscope automatically
```

```
DPOS=100,0                  'coordinate offset


MOVE(-50,100)               'relative motion 1
MOVE(-100,0)                'relative motion 2
MOVE(-50,-100)              'relative motion 3
MOVE(50,-100)               'relative motion 4
MOVE(100,0)                 'relative motion 5
MOVE(50,100)                'relative motion 6
END
```

Then, for this code, in “output” window, it shows there is an error of line 17. Please solve it, then download again.



Also, error information can be checked through “Troubleshooting” window.

6.2. Debugging Tools

Debug tool bar is only valid in debug mode.

a. Go: run the controller.

b. Pause: pause controller, all tasks will be paused.

c. Step into: run into SUB subprogram in single step, if no subprogram, enter next line program.

d. Step over: run to next program. If it runs to SUB subprogram calling, it will not enter but execute SUB program.

e. Step out: jump out SUB subprogram operation

f. Run to Cursor: run to line assigned by the cursor.

Notes:

--when the program is not consistent with controller program, or when the program is not downloaded after modification, there will produce offset for assigned debugging line--

--when the program is paused, motions that have entered in motion buffer will not be paused--

6.3. Breakpoint

We can add the breakpoint in program to find out and pause the program running.

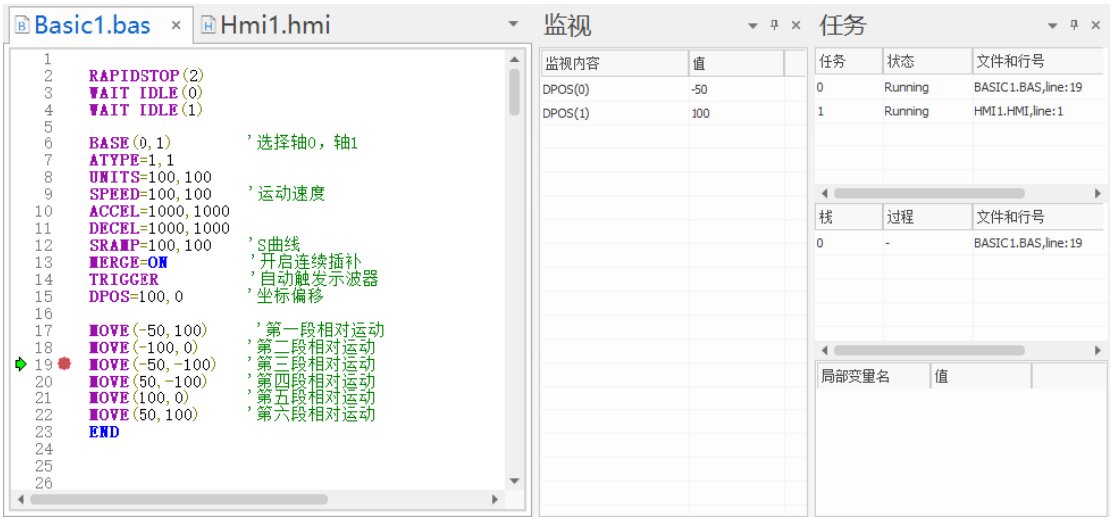
Using breakpoint to debug, detailed process can be known. Mainly used to check program logic errors. Then check the effects of each step program execution on register, variable, array by matching with “watch” and “axis parameter” changes. When the program stops at the breakpoint, you can do step-by-step debug, the shortcut button is F11.

(1) How to Add / Delete Breakpoint

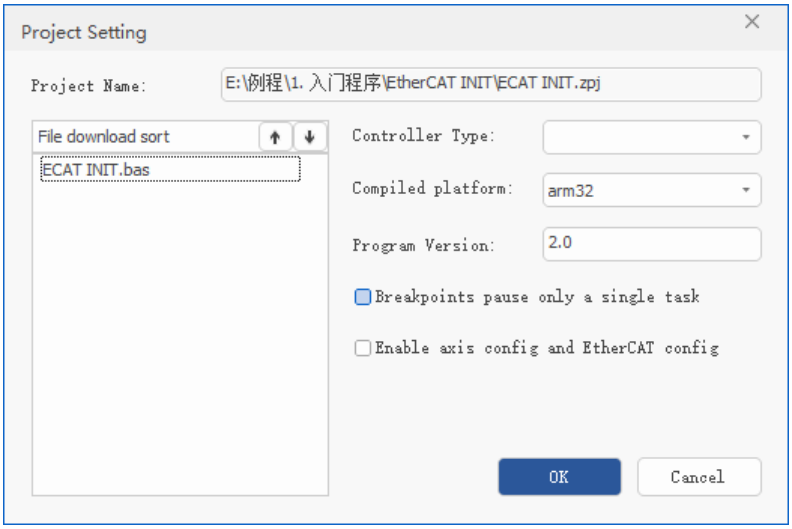
Breakpoint shortcut key F9 is used to add/delete, or "Debug" → "Toggle Breakpoint" in the menu bar. Multiple breakpoints can be added. "Debug" - "Kill All Breakpoints" in the menu bar can clear all breakpoints in the project file at once.

After adding a breakpoint, the program will stop at the breakpoint. At this time, the instructions in the line where the breakpoint is located will not be executed, and the program functions that have been scanned before will not be affected. As shown in the figure below, the 18th line MOVE (-100,0) has been executed. Now, the second interpolation movement is completed, and the position of the axis is (-150,100), the third movement (line 19) is not executed, and the position of the axis remains unchanged.

Note: If the breakpoint is set in a loop, the next time the loop runs to the breakpoint, the program will still stop.

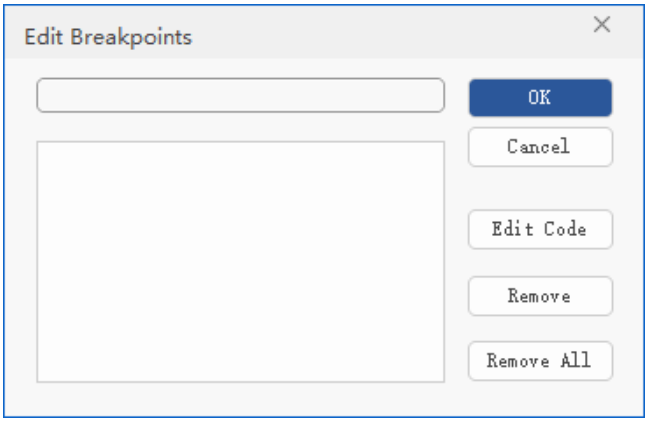


After setting a breakpoint, all tasks will be paused when the program runs to the breakpoint. If you only need to pause the current task, you can go to the menu bar "Controller" → "Settings", check "Breakpoints pause only a single task". After that, the breakpoint added will only pause a single task (Note: before checking, all tasks will be paused by added breakpoint by default).



(2) How to Edit the Breakpoint

Through the breakpoint editing window, you can view all the breakpoints added in the current file and edit all the lines where the breakpoints are located. Double-click the breakpoint information to jump to the breakpoint line. You can choose to remove one or more breakpoints. After removing, click "OK" to take effect.



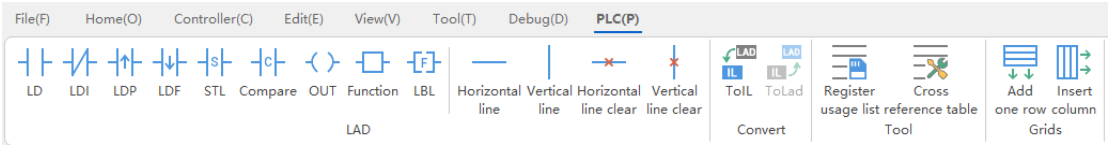
After the program is debugged, you need to clear all breakpoints or turn off the debug mode before downloading it to the controller for running. If you do not turn off the debug mode and clear the breakpoints, the following warning message will be printed in the command and output area: Warn file: "Basic1.BAS" line: 11 task: 0, Paused.

6.4. Emergency Stop

Emergency stop can immediately stop the program and the motion of all axes.

During the motion control debugging process, in order to avoid emergency situations such as loss of control, the emergency stop function is configured in the RTSys software. The function button can be found in the menu bar "Common" → "Emergency Stop" (or "Debug" → "Emergency Stop").

Chapter VII PLC Menu



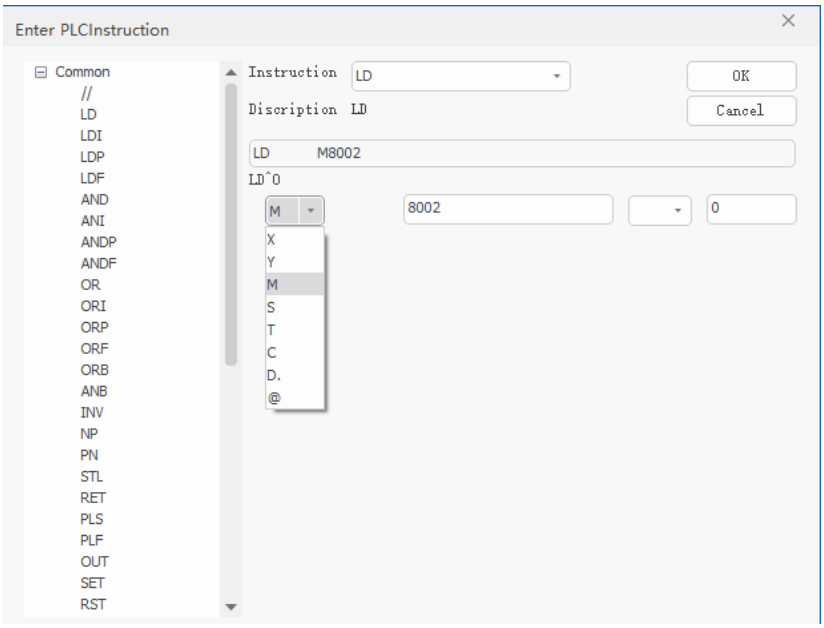
--how to operate quickly--

Name	Image Mark	Description
LAD		
LD		Usual-on contact to connect with bus line.
LDI		Usual-off contact to connect with bus line.
LDP		Used to detect the rising edge of the normally open contact connected to the busbar. It connects a scan cycle only when the rising edge of the specified bit soft element (when it changes from OFF to ON)
LDF		Used to detect the falling edge of the normally open contact connected to the busbar. It connects a scan cycle only when the falling edge of the specified bit soft element (when it changes from ON to OFF)
STL		Starting command of program that uses stepper LAD command.
Compare		Compare two data, that is, compare operand S1 with operand S2 according to assigned condition. When the condition is met, contact is conducted, if not met, contact is off.
OUT		Command that drives soft element coil.
Function		Open PLC command input list, select the command.
LBL		Build PLC subfunction, which is used as entry of subfunction.
Horizontal line		Add LAD horizontal line.
Vertical line		Add LAD vertical line.

Horizontal line clear		Delete LAD horizontal line.
Vertical line clear		Delete LAD vertical line.
Convert		
ToIL		Convert LAD to IL (instruction list)
ToLad		Convert IL to LAD (ladder of diagram)
Tool		
Register usage list		Check registers usages and notes under current project.
Cross reference table		Check how to use types of registers and position for current project.
Grids		
Add one run		Insert one row above the selected grid.
Insert column		Expand one column on the left side of selected grid.

7.1. PLC Shortcut Tools

Take “LD” command as the example, click it, then below window will show, in “S”, select the operand, click OK, then, in PLC, it forms as one common-on contact.



7.2. Codes Transformation

This function is only used in PLC programming.

There are two programming methods: ladder diagram (LAD) and instruction list (IL). LAD is convenient for visual programming, and IL programming is for users who are familiar with instruction syntax and programming logic.

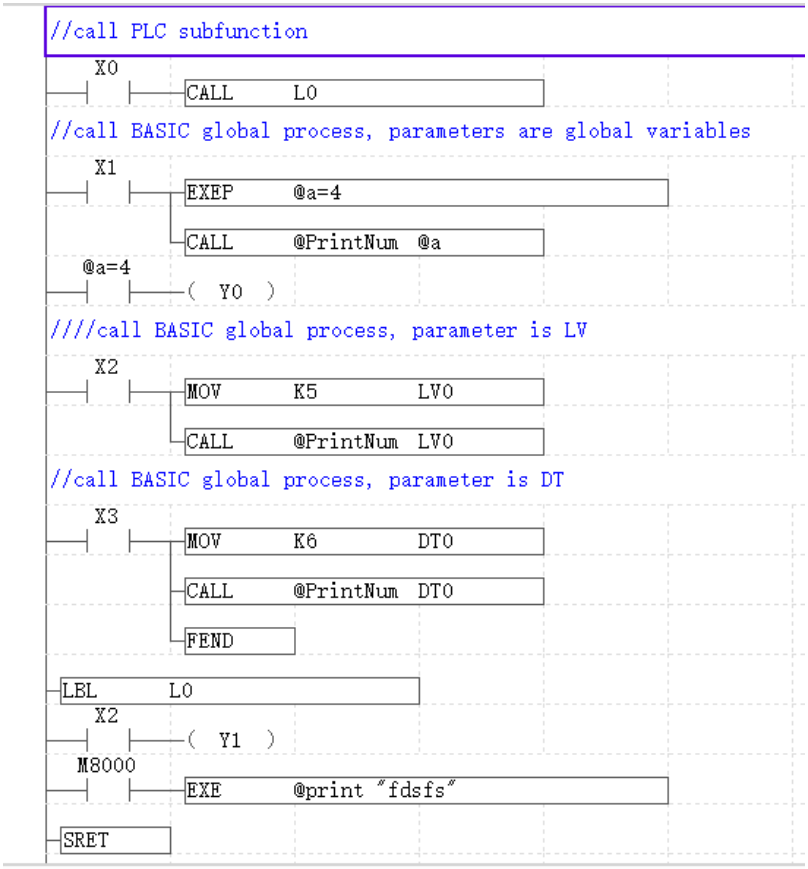
7.2.1. ToIL (LAD to IL)

Please see below example, and note the syntax should be correct.

```
//call PLC subfunction
ld x0
call l0
//call BASIC global process, parameters are global variables
ld x1
exep    @a=4
CALL    @(PrintNum) @a
ld @a=4
out y0
////call BASIC global process, parameter is LV
ld x2
mov k5 lv0
CALL    @(PrintNum) lv0
//call BASIC global process, parameter is DT
ld x3
mov k6 dt0
CALL    @(PrintNum) dt0
fend
lbl l0
ld x2
out y1
ld m8000
exe @print "fdsfs"
sret
end
```

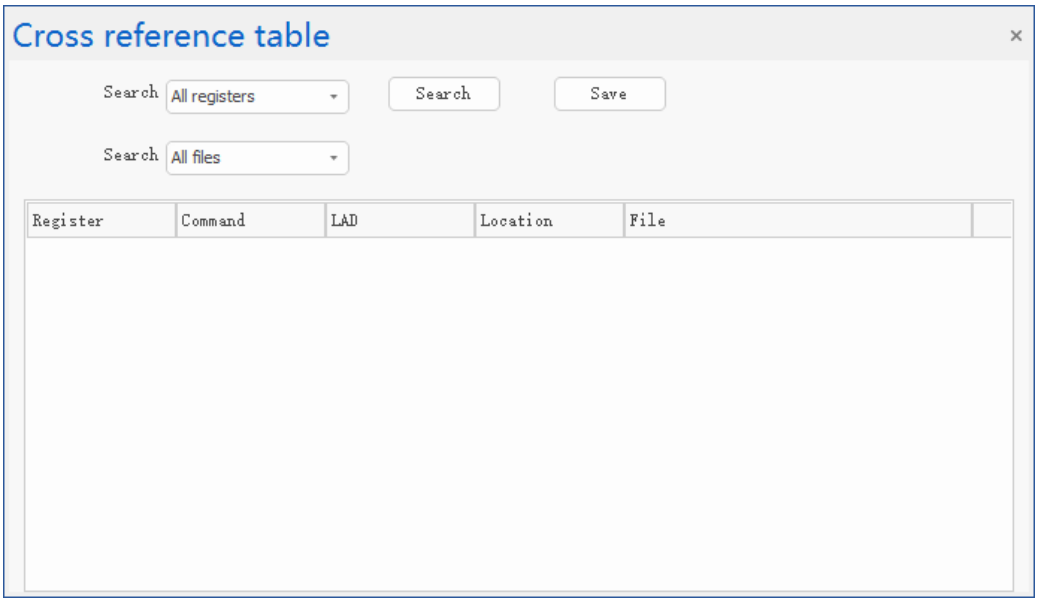
7.2.2. ToLAD (IL to LAD)

Convert above IL to LAD (also syntax must be correct):



7.3. Cross Reference Table

Open the window shown below through the menu bar "PLC"-"Cross Reference Table".

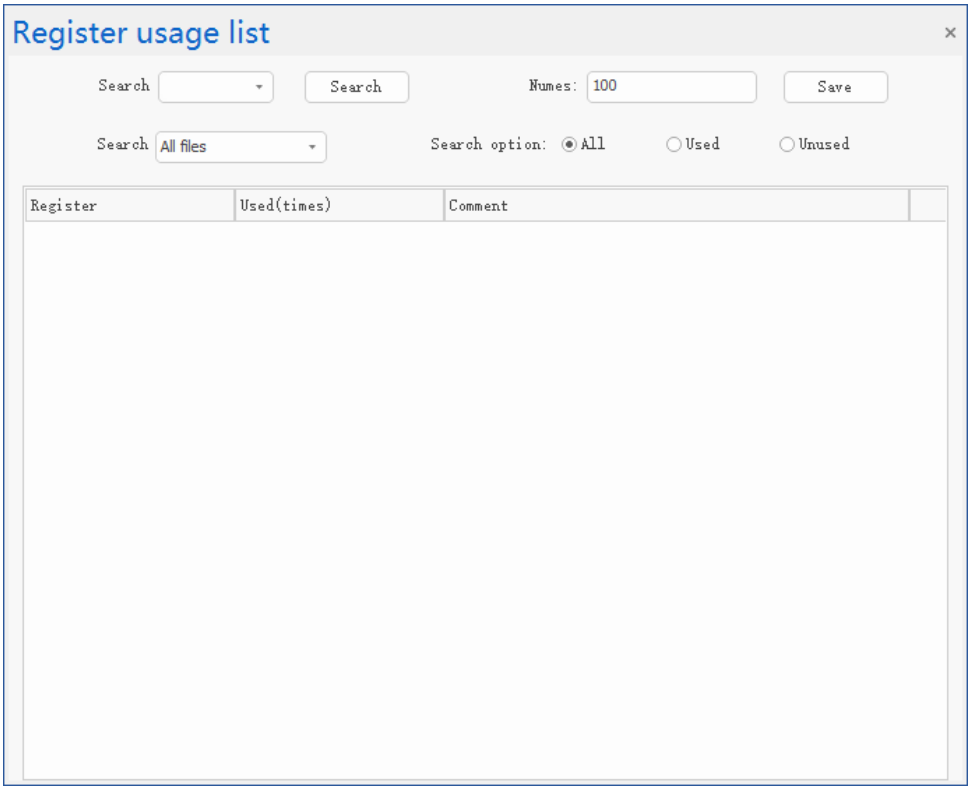


The "Cross Reference Table" is only available for PLC programming mode. Check the specific information of the used registers. Double-click the line to jump the cursor to the corresponding pane of the register in the program.

The "Save" button in the upper right corner is used to save the searched data in csv format.

7.4. Register Usage List

Open the window shown below through the menu bar "PLC"-"Register Usage List".



"Register Usage List" is only applicable to PLC programming mode. It is convenient to check the usage of registers, and easy to know the number of times the registers are used in the program. What's more, it can show or edit comments in the comment column. To display comments in the ladder diagram programming interface, you need to check "Notes Show " in the right-click menu.

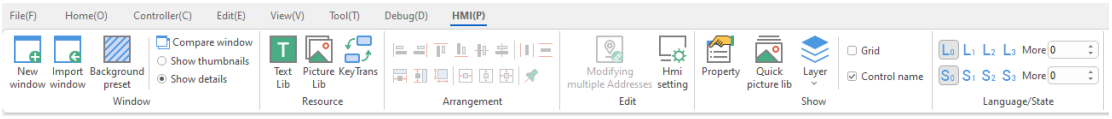
--How to Use--

Select the register to be searched, enter the number, the PLC file to be searched under this project, etc., click "Search", and the search results will be displayed in the window.

Double-click the left mouse button on the "Register" or "Used (times)" lines to pop up the "Cross Reference Table" window.

The "Save" button in the upper right corner is used to save the searched data as csv format.

Chapter VIII HMI Menu



--how to operate quickly--

Name	Image Mark	Description
Window		
New Window		New build one Hmi window.
Import Window		Import existed Hmi window (only can be .hmi form)
Background preset		Preset global window background and element form.
Compare Window		
Show thumbnails	/	Show as window thumbnail.
Show details	/	Show as window & element details.
Resource		
Tool Box		Open/hide “tool box” window by “View”, it can add and call all HMI components.
Text Lib		Set texts with multiple languages at once and save them to call in element.
Picture Lib		Add the picture to picture library, and support calling. There are system picture library and user picture library, pictures are used only for Hmi.
KeyTrans		Bind functions of physical keys and virtual keys.
Arrangement		
Arrangement	/	Sort multiple elements, there are many options.
Edit		
Modifying multiple Addresses		Modify multiple register addresses.
Hmi settings		Preset Hmi system, including starting window, resolution, etc.
Show		

Property		ON / OFF "property" window, and it can check / set HMI element / window property information.
Quick picture lib		ON / OFF "shortcut picture lib" window, HMI pictures can be checked, and can be used or deleted quickly.
Layer		Show / hide elements of top, middle, bottom parts.
Grid	/	Show / delete the grid.
Control name	/	Show / hide element name in Hmi window.
Language / State		
Language		Switch the language in text library
State		Switch element state.

8.1. How to Set Component

There are many elements in Hmi programming. You can call objects from the "Tool Box", and can set objects' properties.

For more, please refer to "Tool Box". And see "RTHMI Programming Manual".

(1) How to Add Element (Component):

Step 1: New build one HMI file / open one HMI file

Step 2: open HMI window / build one HMI window

Step 3: open "Tool Box"

Step 4: click the object in the tool box, and move it into HMI window.

(2) How to Set Component:

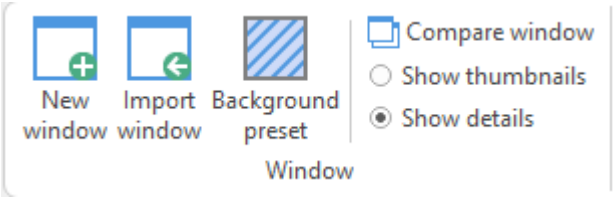
Property	Function	Description
Basic Feature		
Object ID	It can modify the No.	/
Object Name	It can modify the name.	/
Layer	Select component display layer	➢ TopLayer: the surface, it shows the most external layer, and covers below components.

		➢ MidLayer: the middle layer ➢ BottomLayer: the bottom layer (default)
IfValid	Object is shown or not	➢ Show: Objects will be shown and can be called after downloading. ➢ Hide: not show after downloading ➢ Show & Disable: show but can't use after downloading.
Valid Control	Determine object is shown or not through register	Default is False. If TURE, below 3 parameters will be shown.
Valid Device	Device No.	Default is local
Valid regtype	Select register type	Select from the list
Valid regnum	Select register No.	It does not show when register value is 0, it is used when not 0.
Safe timems	The leaset button time	Unit is ms.
Bound Virtual Keys	Select virtual codes to be bound	Not used by default.
Bound Physical Keys	Select pysical codes of HMI (teach pendant) to be bound.	Button codes, please refer to "vitrual buttons".
Appearance		
Use Picture Lib	BackPicture / Back Picture	Select from picture library or backgroud
Back Picture Lib	Select background picture	After select background picture for picture source, then add
If Draw edge	Draw the edge?	/
If MakePic	Change object to image?	Default is False
Label		
Text Lib	The name of text library	If empty, indicates the use of a text label
Format text	Open the Format Text Settings window to set the text to be displayed by the component	Default is text 0, press it, it will show information set in text 1.
Action		

Action	Motion to be executed when button	Please refer to “action”.
Action when up	Select execution action when press or release.	Default False: the action when pressed, True: the action when released
Action sub name	SUB function to be called after pressing	Select global SUB function of Basic
Position and size		
Top	Vertical starting position	Can’t exceed vertical resolution
Left	Horizontal starting position	Can’t exceed Horizonical resolution
Width	The width of element	/
Height	The height of element	/

Note: different objects are with different properties, more, please refer to RTHMI Programming Manual.

8.2. How to Set Window



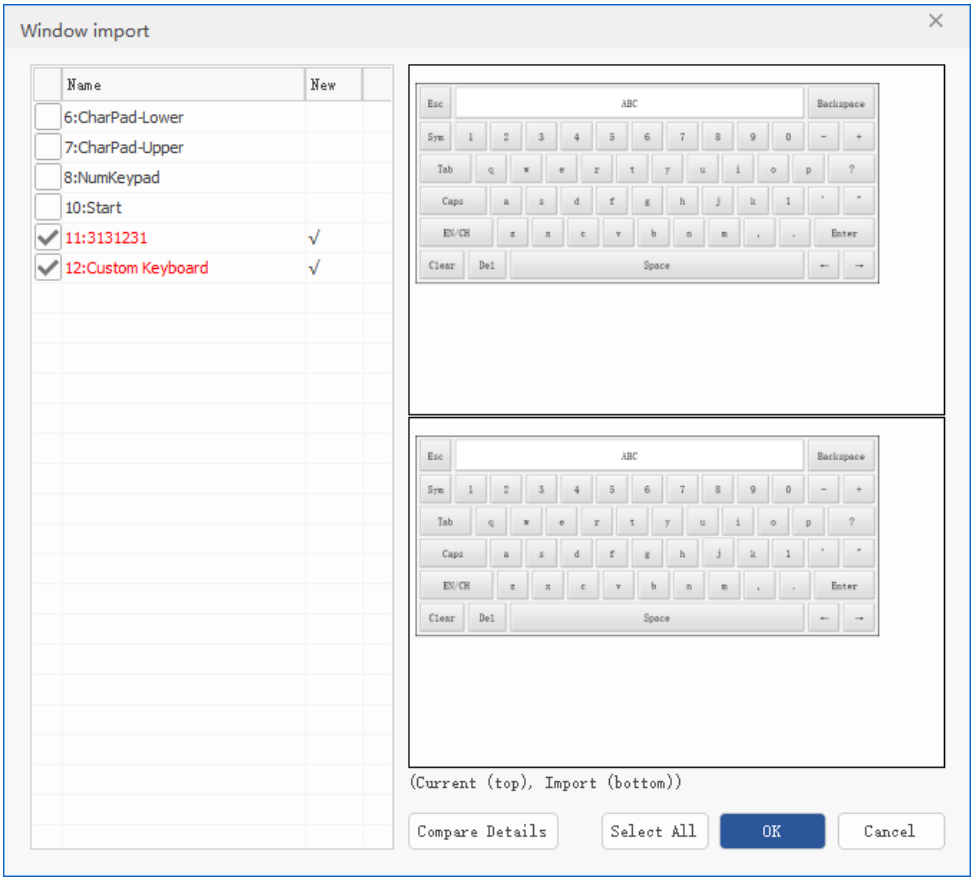
8.2.1. Create / Import Window

(1) New Window

In current project, new build one window or multiple windows.

(2) Import Window

In current project, import created HMI window from other projects into here (one / multiple). After that, below window will appear, here, you can select which you need. Please note “red” means current window No. and imported window No. repeat, choose replace or not.



8.2.2. Preset Background

This, you can set HMI window background and object's default formats / colors. Please note it is valid when building new window or object after saving the settings, for created windows and objects, original formats keep.

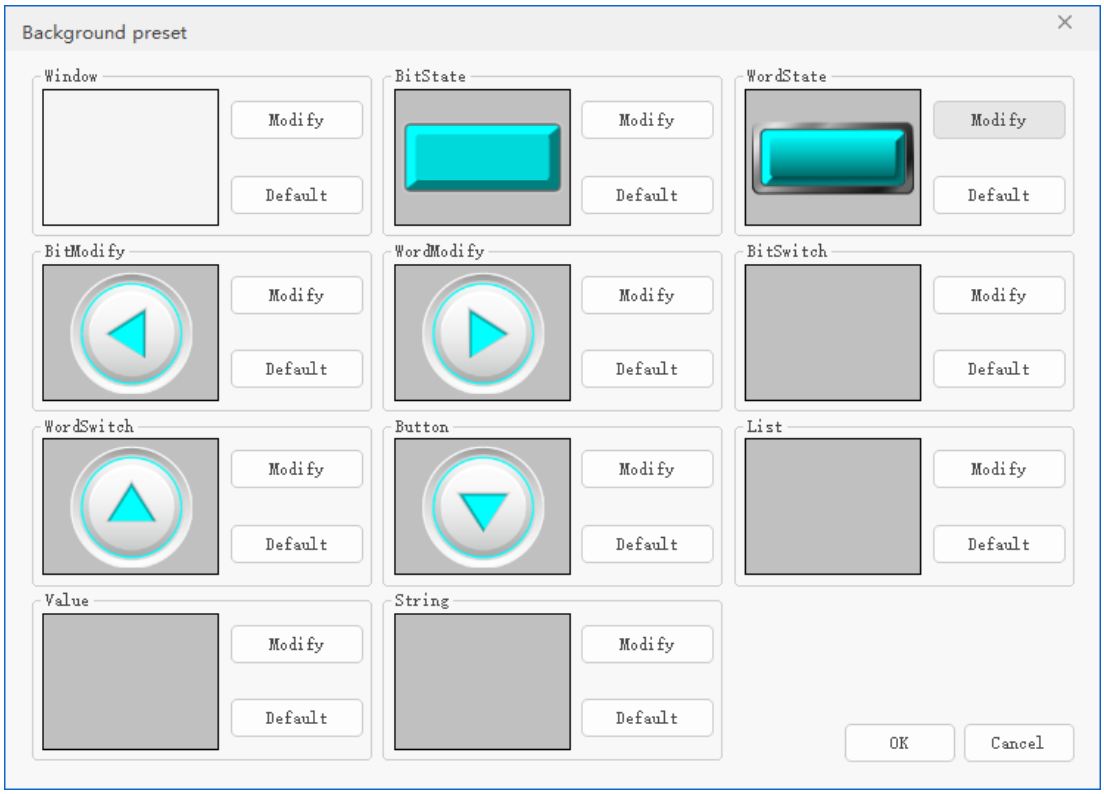
--How to Operate--

Select needed object / window, then click "modify", there are two format selections.

- Method 1: if you are ready to use picture library formats, import the picture from picture lib, and change the image ratio can adjust object showing size.
- Method 2: custom colors for the state 0 and state 1.

Between these two methods, only one takes effect. If "picture lib" is used, custom colors and formats will be covered.

For window background preset, use method 2, and set state 0. It takes effect after building new window.



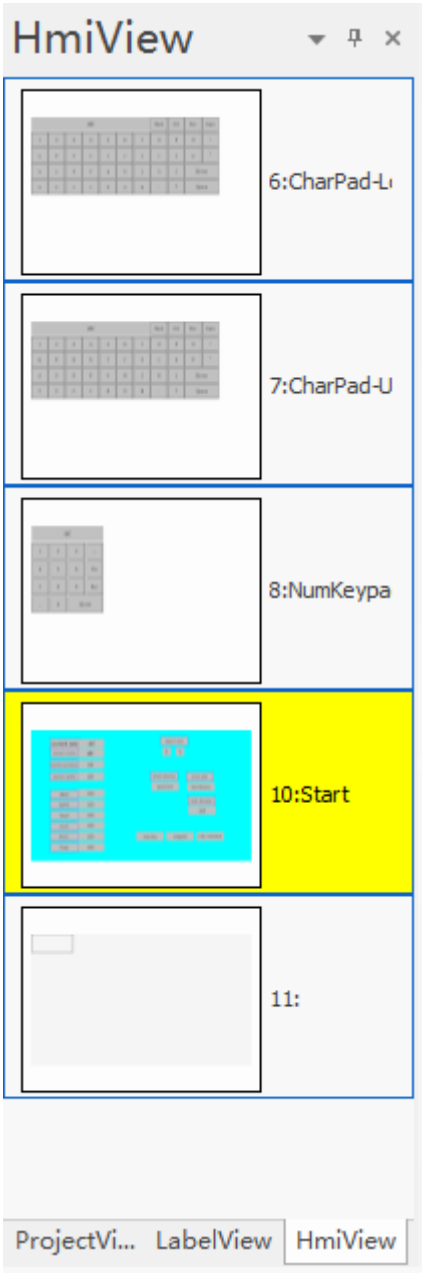
8.2.3. Show Thumbnails / Details

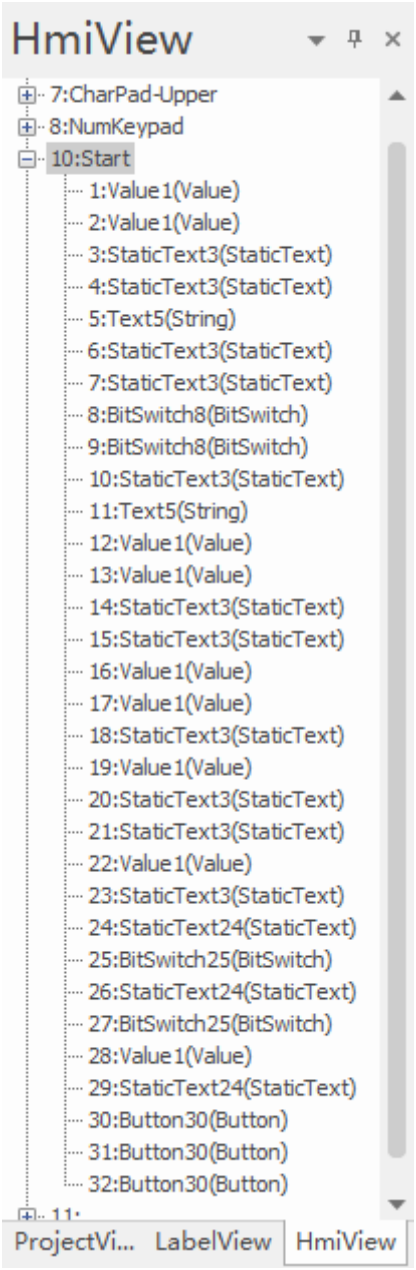
(1) Show Thumbnails

In HmiView, window thumbnails, window No., and window name will be shown (yellow background indicates currently opened window, you can switch through mouse clicking) [Left Image].

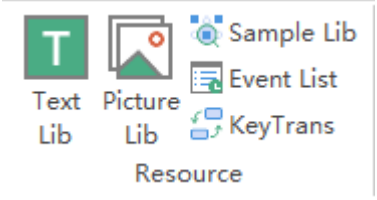
(2) Show Details

In HmiView, window information (window No., window name), and object information (created object No., object name) all will be shown [Right Image].



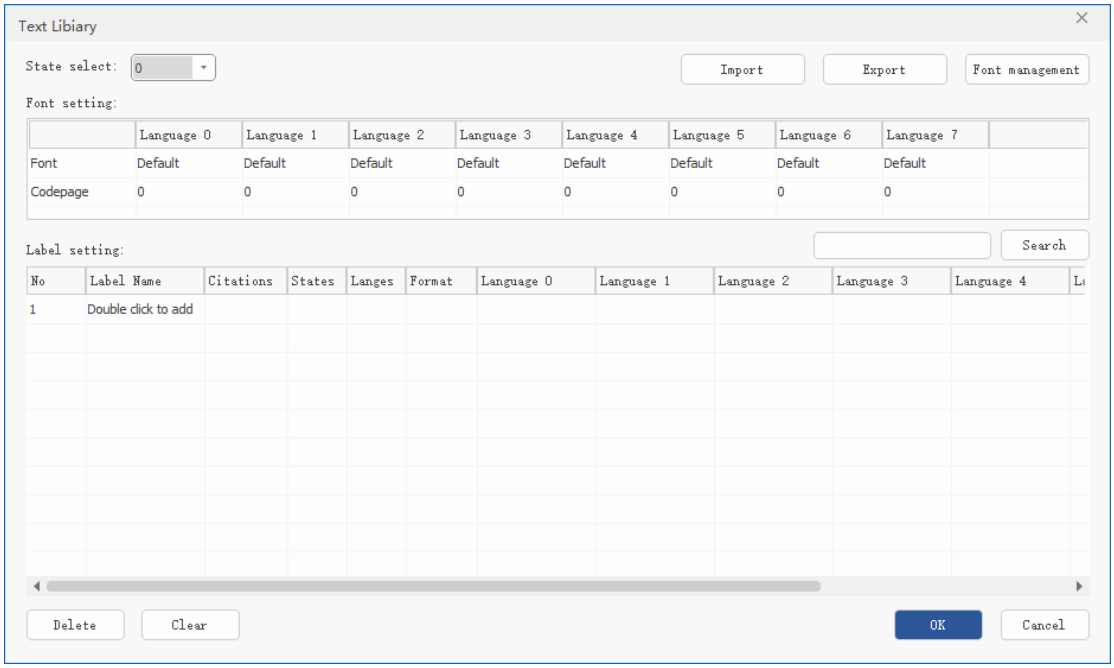


8.3. How to Use HMI Resources



8.3.1. Text Lib

Use labels to set multiple language texts in different states and the fonts corresponding to each text at one time and call it in the corresponding component. Different text content can be filled in each language. (One label supports up to 256 states, and one state supports up to 8 language texts)



--How to Operate--

- Double-click “label name” to add ("Label" is used to distinguish which text library the component calls).
- Set the required number of states and languages, and the format of the text.
- Select the state, and add the text content to be displayed to the corresponding number of languages in the selected state.
- After setting the label and language text, if no need to set font, click [OK] to save.
- If you need to set the font, you need to add the font file to the project first. After importing the font, you can set the font for each language in the [Font Settings], then call the corresponding text library and run it to display.

8.3.2. Picture Lib

A library that integrates and stores Hmi component style images or custom images. It includes system image library and user image library. This library supports modification of image style color, display content/color corresponding to different states, etc.

Note: The maximum number of image libraries is 512.

➢ System Picture Library

The system image library is the system default image and does not support deletion or addition. Here, rich component styles for you, you can select and use directly.

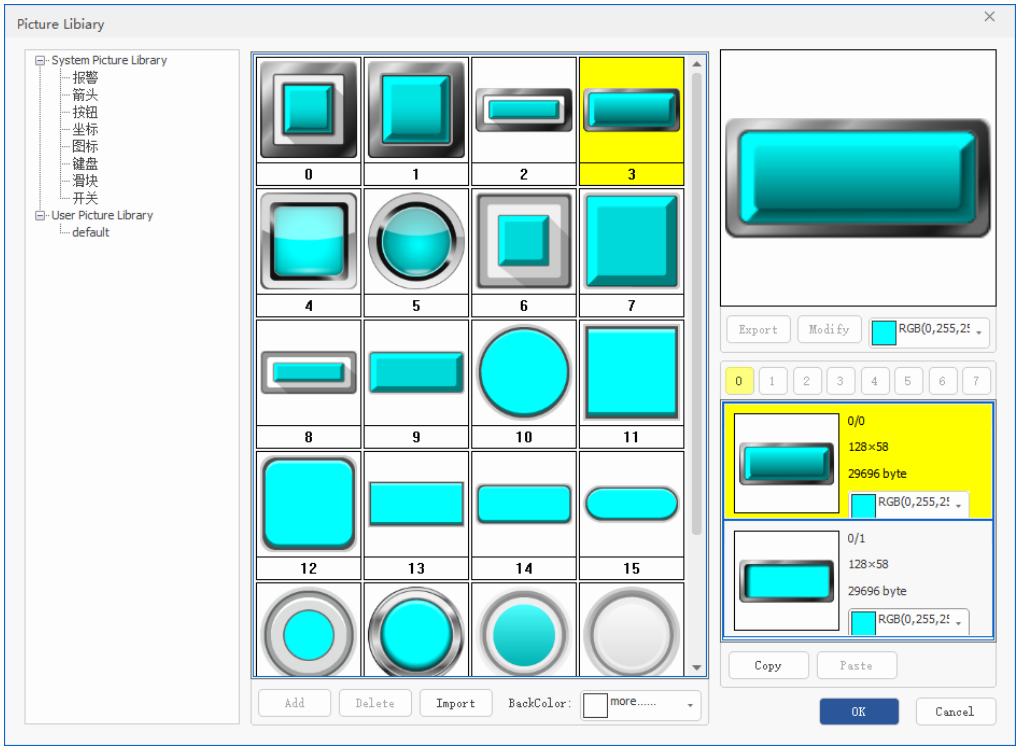
➢ User Picture Library

Here, you can create your own picture library, also can add external images.

--How to Use Pictures for Component in Picture Lib--

(1) Method 1: modify component style directly in “background preset”, for details, please refer to “preset background”.

(2) Method 2: add one object in HMI window, and in its “property” window – “use picture lib”, select “back picture lib”, then click “…” to open picture library, choose needed image, next click OK.



8.3.3. Sample Lib

It is used together with HMI "trend chart" tool, specifically, define the trend chart's properties through resource sample library, like, sampling, and do channel configurations, then, the system will capture according to this sampling method. Sampling methods include periodic, and triggered ways. What's more, it can add, delete, copy, insert, replace, and clear resource sampling items.

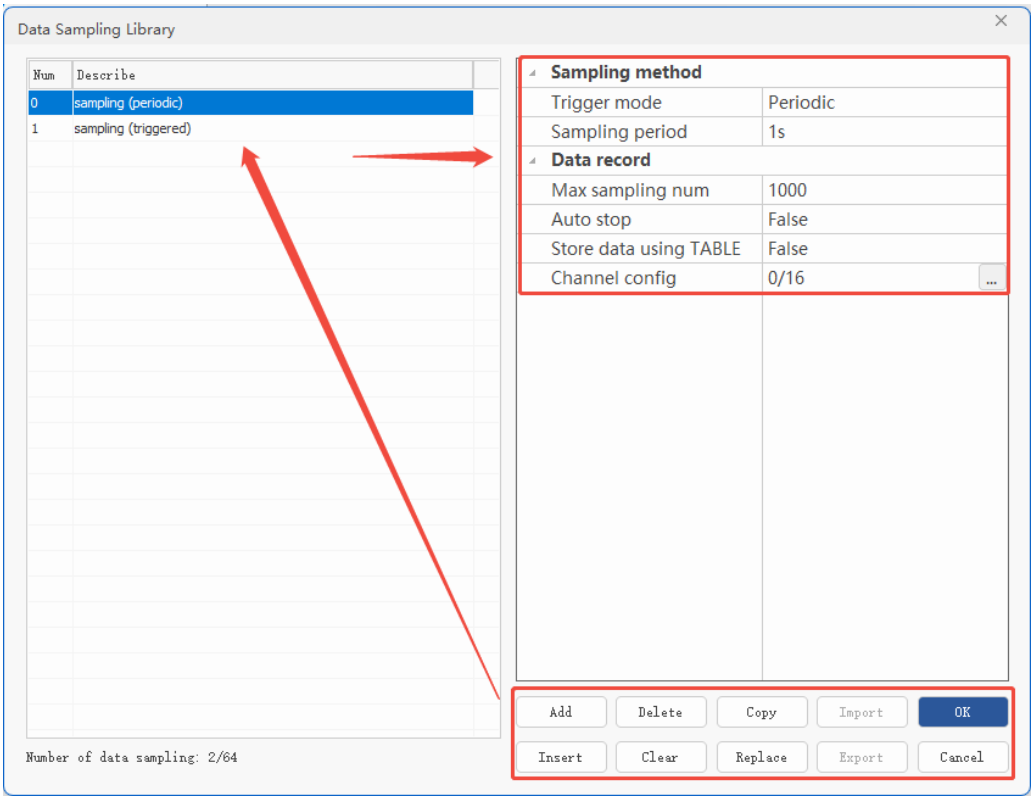
- **Basic Usage:**

1) Open "resource sampling lib" (sample lib), and click "add" to add needed sampling.

2) Double-click the content (describe) of the new added sampling, and enter information.

3) Then single-click to choose one, set its properties, like, right shown, sampling mode, sampling period, max sampling numbers, etc.

4) After that, please remember to click OK.



"For more details, please refer to RTHmi Programming Manual"

8.3.4. Event List

"Event List" is used to define trigger conditions of event and the contents. Then, the system will trigger the event when conditions are met, showing contents in "info display components" of "event show", "alarm show", "alarm bar".

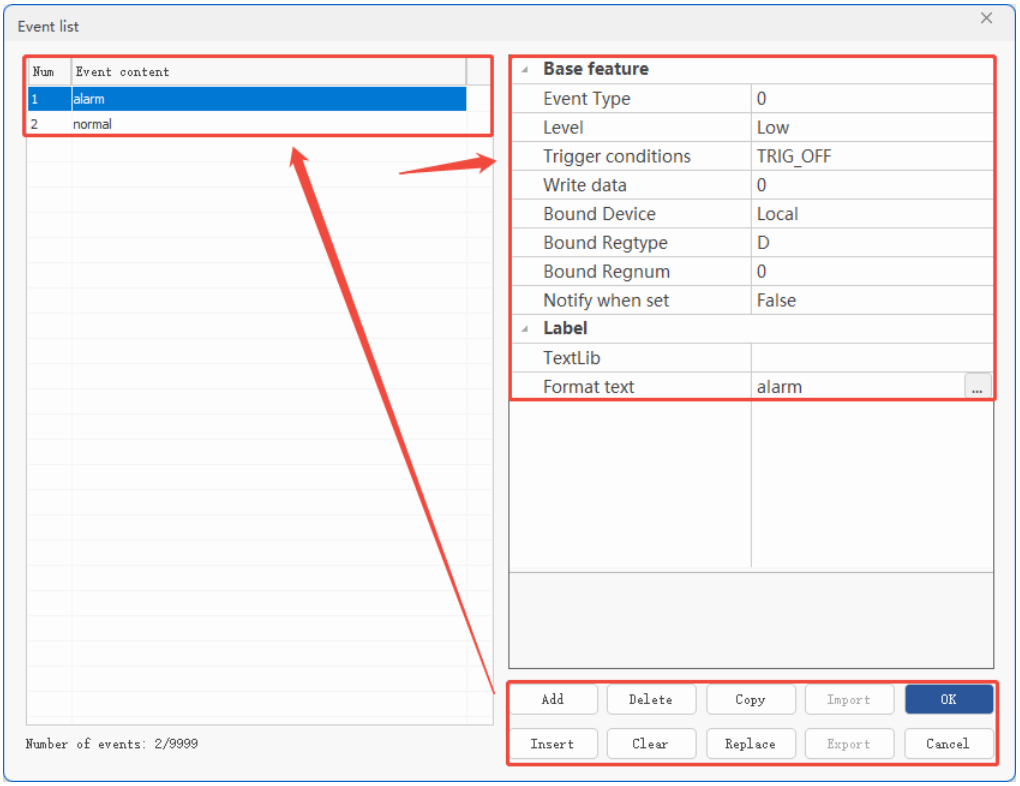
- **Basic Usage:**

1) Open "event list", and click "add" to add an event or events.

2) Double-click the event content, enter information.

3) Then single-click to choose one, set its properties, like, right shown, event type, level, trigger conditions, etc.

4) After that, please remember to click OK.



"For more details, please refer to RTHmi Programming Manual"

8.3.5. Keys Transformation

Bind the functions of physical keys and virtual keys, then operate physical keys to trigger virtual keys. Now, this tool has preset the key functions of ZHD300X and ZHD400X. In addition, it also supports exporting or importing the set key value content. ("HMI" → "Key Trans")

(1) Physical Keys:

A physical key refers to the actual key on an external device. Each key has a unique encoding value (such as the physical key encoding value in the figure below). When pressed, a message will be sent, which is the encoding value of the key.

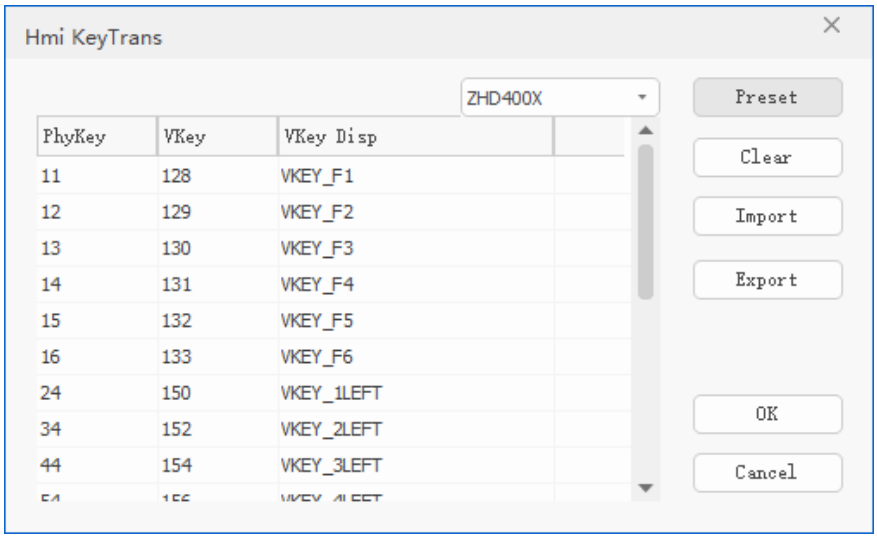
(2) Virtual Keys:

Virtual key encoding values 0-127 all correspond to the ASCII code table, and the corresponding functions after 128 can be customized. For the virtual key value comparison table, please refer to the "RTHmi Programming Manual".

Notes: The encoding value of the physical key is determined by the hardware and cannot be modified in the program. Therefore, different keys are with separate encoder values. Except this, in HMI, virtual keys' codes are encapsulated by bottom level, no way to modify in program.

--How to Operate--

- If you need to ZHD300X and ZHD400X key functions that have been set already: open "Hmi Keytrans" window – in drop-down list, select ZHD300X or ZHD400X – click "preset", at this time, corresponding key functions are shown in left – click OK (if you want to change, click clear to clear all, or double-click to modify).
- If you need to create your own key functions: open "Hmi Keytrans" window – double-click empty place to enter physical value and virtual value – click OK.



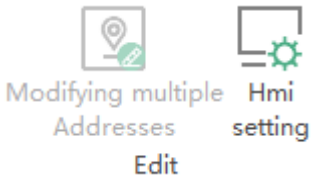
8.4. How to Sort Components

RTSys provides a variety of alignment and arrangement methods, arranging multiple components neatly according to certain rules, making the entire Hmi interface more beautiful and orderly. That is, you need to select multiple components at the same time to arrange them.

It includes left/right alignment, top/bottom alignment, horizontal/vertical center alignment, horizontal/vertical same spacing, same width/height/size, horizontal/vertical center display of the window, and locked components. (You can use it through the menu bar "HMI" → "Arrange")

The above arrangement methods all use the red box displayed in the selected component as the target component, and use the target component as the standard for arrangement and alignment. If you need to customize the target component, first select the target component, hold down the "crtl" key, click other follow-up components with the mouse, release the "ctrl" key after the component selection is completed, and then select the arrangement method.

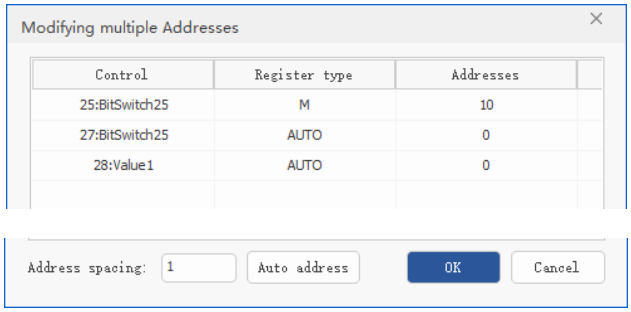
8.5. How to Edit HMI



8.5.1. Modify Multiple Addresses

You can modify several HMI components' register addresses as one register type at one time, including address space setting. Operate by "HMI" – "Modifying multiple Addresses".

Note: it is only valid in HMI components that binds with register.



--How to Operate--

- In HMI window, select several components that set register (recommend to select components that can use same register type).
- Open above window, then you can see related component name and register type, address.
- Select "register type", drop-down the list to choose.
- Set starting address No. (AUTO: automatically follow with the last one register type).
- If you need set address space, set by "address spacing". Default is 1. After that, click "auto address". Then, AUTO type will become needed one, and show according to the sequence of address. At last, click OK.

--Notes--

- The first one component can not be AUTO type.
- For the address, addresses can't be same.

- Showing sequence is determined by adding sequence. Therefore, select them who use same register type.
- After “auto-address” each time, if you modify the type and “auto-address” again, please manually modify and set as AUTO type.

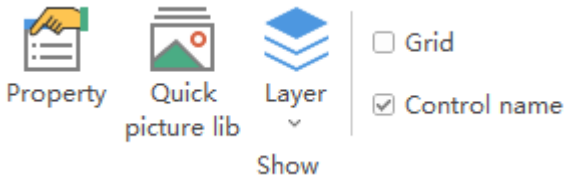
8.5.2. HMI Settings

Perform initial property settings for the HMI system, and modify properties such as the HMI window resolution and the initial basic window. For details, see the table below. You can open the property window of the HMI system settings through the menu bar "HMI" → "Hmi Settings".

Property	Function	Description
Lcd number	Set the LCD screen No.	While connecting to HMI, select which HMI file content is shown by this No.
Backlight time	The actual backlight time of the teaching box	/
Screen time	Set screen saver time	/
Startup base window	Set the HMI initial base type window	Windwo 10 is shown by default.
Starup top window	Set the HMI initial top window	/
Init Sub	Add HMI initial sub function	The sub fuinction is only called once after powered on, and it must be GLOBAL sub.
Period Sub	Add HMI period sub function	The sub fuinction is called in cyle after powered on, and it must be GLOBAL sub.
Compress pictures	Compress pictuers or not	Old Compress: low compress quality and can’t be resumed (that is, ZDevelop compress method). New Compress: compress and keep same quality No Compress: don’t compress
Picture quality	Select picture quality (when	Standard: low image showing quality,

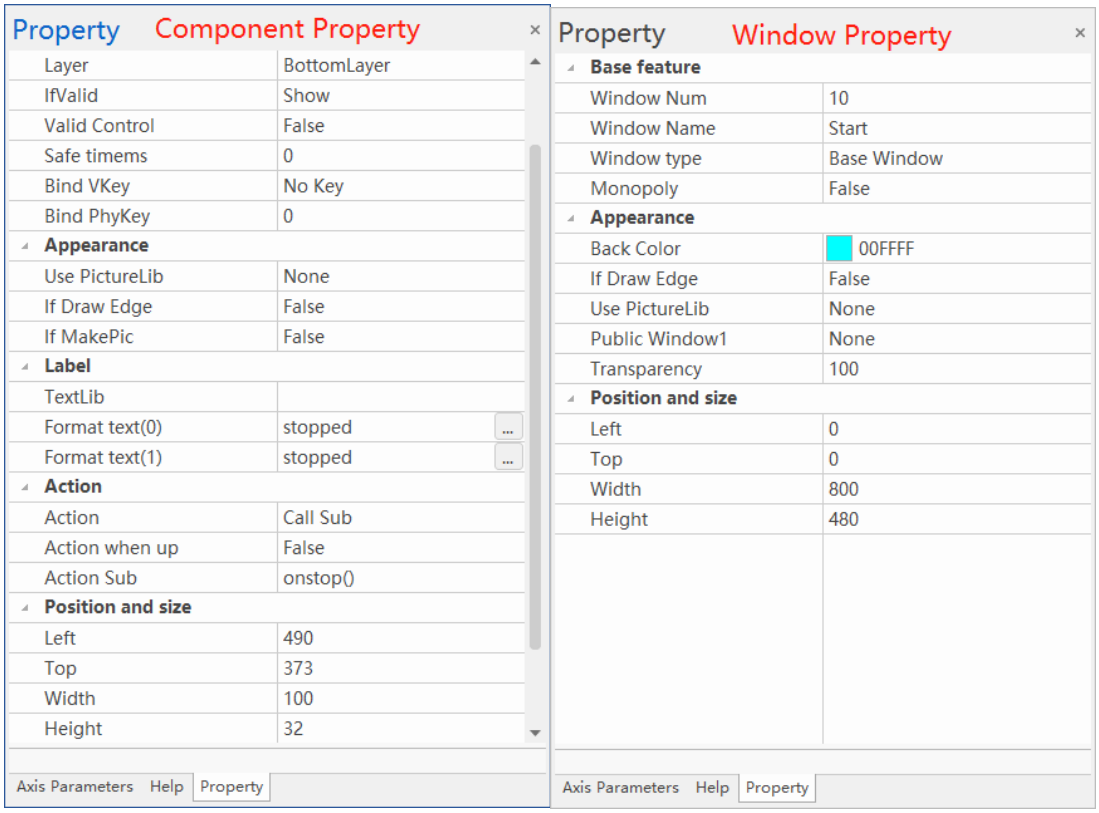
	not compressed, the quality depends on if it supports "anti-aliasing zoom"	but HMO performance is high. High: high image showing quality, but HMO performance is low.
Text adaptive sizing	Text adapts to component's size.	When the text exceeds component range, font will be zoomed out automatically, the smallest font will not be lower 50% than you set one.
Screen Width	Window display r/esolution	/
Scren Height	Window display resolution	/

8.6. How to Set Showing



8.6.1. Window Property

Used to show and set properties of window / component in HMI file. Before, new build / open one HMI file, and then click "HMI" – "Property".



(3) Component Property Introduction

Property	Function	Description
Basic Feature		
Object ID	It can modify the No.	/
Object Name	It can modify the name.	/
Layer	Select component display layer	➢ TopLayer: the surface, it shows the most external layer, and covers below components. ➢ MidLayer: the middle layer ➢ BottomLayer: the bottom layer (default)
IfValid	Object is shown or not	➢ Show: Objects will be shown and can be called after downloading. ➢ Hide: not show after downloading ➢ Show & Disable: show but can't use after downloading.
Valid Control	Determine object is shown or not through	Default is False. If TURE, below 3 parameters will be shown.

	register	
Valid Device	Device No.	Default is local
Valid regtype	Select register type	Select from the list
Valid regnum	Select register No.	It does not show when register value is 0, it is used when not 0.
Safe timems	The leaset button time	Unit is ms.
Bound Virtual Keys	Select virtual codes to be bound	Not used by default.
Bound Physical Keys	Select pysical codes of HMI (teach pendant) to be bound.	Button codes, please refer to “vitrual buttons”.
Appearance		
Use Picture Lib	BackPicture / Back Picture	Select from picture library or backgroud
Back Picture Lib	Select background picture	After select background picture for picture source, then add
If Draw edge	Draw the edge?	/
If MakePic	Change object to image?	Default is False
Label		
Text Lib	The name of text library	If empty, indicates the use of a text label
Format text 0 / 1	Open the Format Text Settings window to set the text to be displayed by the component	Default is text 0, press it, it will show information set in text 1.
Action		
Action	Motion to be executed when button	Please refer to “action”.
Action when up	Select execution action when press or release.	Default False: the action when pressed, True: the action when released
Action sub name	SUB function to be called after pressing	Select global SUB function of Basic
Position and size		
Top	Vertical starting position	Can’t exceed vertical resolution
Left	Horizontal starting	Can’t exceed Horizonical resolution

	position	
Width	The width of element	/
Height	The height of element	/

➢ **Window Property Introduction**

Property	Function	Description
Basic Feature		
Window Num	Current window No.	Under one project, window No. can't be same.
Window Name	Current window name	/
Window type	There are 5 window types	Refer to "window type".
Monopoly	Monopoly or not?	After the monopoly, the components below the window cannot be operated
Appearance		
Back Color	Select window background color	/
If Draw edge	Draw the edge?	After TRUE, edge color can be selected.
Use PictureLib	BackPicture / Back Picture	You must add a picture before you can select it. The picture name cannot exceed 26 characters.
Public Window1	Set current window's public window 1	The current window can display the controls of the public window. Up to 3 public windows can be set.
Transparency	Background transparency	Reserved.
Position and size		
Top	The X coordinate of the upper left corner of the window display	Can't exceed vertical resolution
Left	The Y coordinate of the upper left corner of the window display	Can't exceed Horizonical resolution
Width	Current window's showing width	/
Height	Current window's showing height	/

8.6.2. Quick Picture Lib

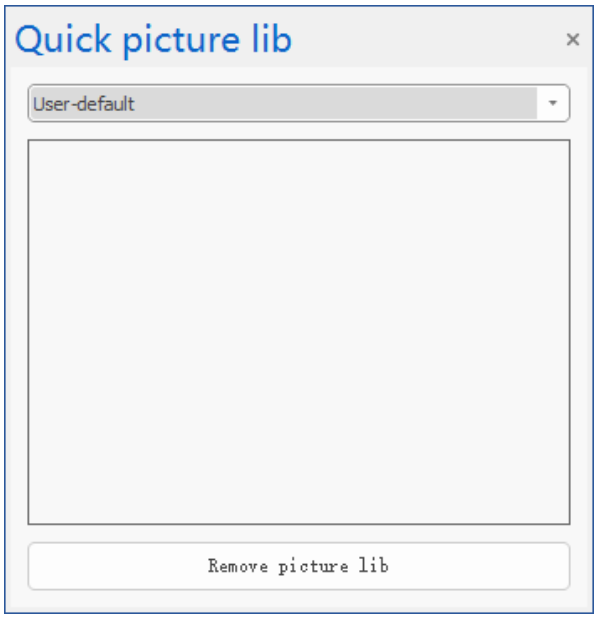
Used to quickly apply or remove styles from the pictuer library to HMI components. You can open this window through the menu bar "HMI" → "Quick Picture Library".

(1) Add Picture Library

Open HMI file, select one single component in HMI window, and open “quick picture lib”, find needed pictures, double-click it to use it (there are many classifications in “picture lib”, select from “drop-down” list).

(2) Remove Picture Library

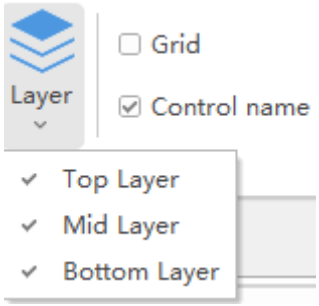
Choose the component, then remove it.



(3) Show / Hide Layer

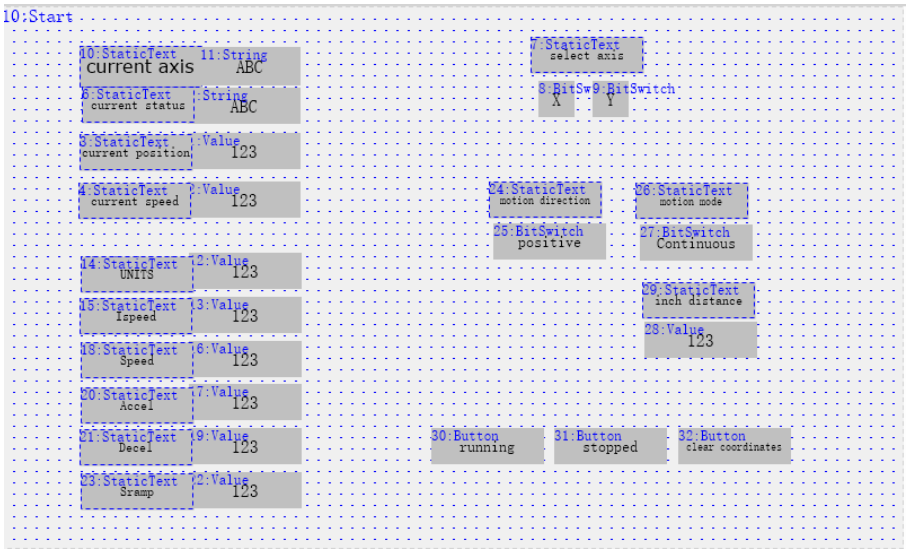
Show or hide components of different layers (HMI – Layer)

There are 3 layer operations, top, middle, bottom.

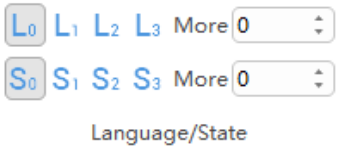


(4) Grid & Control Name

Show or not (grid, component name, window name). Both are checked, as like:



8.7. Language / State Switching



(1) Switch Language

Switch the language of the component that has called the text library. You need to set the content corresponding to each language in the current state in the text library first, call the text library in the component, then select the language to switch the text content, L0 corresponds to language 0, L1 corresponds to language 1, and so on. If there are more than that, you can enter the language number in More. A maximum of 8 languages can be set, that is, L7.

(2) Switch State

To switch the state of a function key or bit state/multi-state component, select state S to switch to a different state. S0 corresponds to state 0, S1 corresponds to state 1, and so on. A maximum of 256 states can be set, that is, S255.

Chapter VIIII RTSys File Types

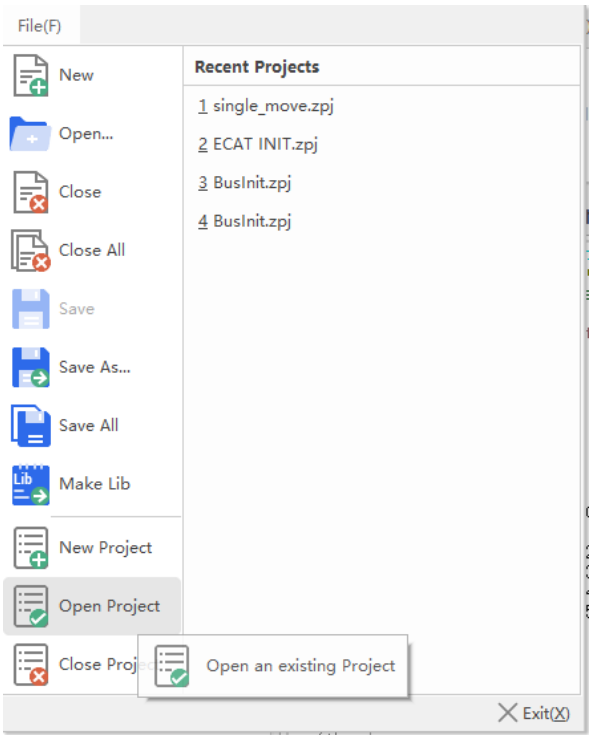
Using RTSys, you must understand “Project”. Build one folder that will save all corresponding program files. One project can include one file or several files.

9.1. Project File

File name of project file with suffix “.zpj”, files in program must be in the same folder as project file (namely, bas/plc/hmi file included by zpj file are saved in same folder).

Click “File”, and select “open project”, added project file (bas/plc/hmi file in this zpj) will open automatically, or drag zpj file to RTSys so that open directly.

If only open bas/plc/hmi file, related projects are not opened, then program can’t be downloaded, which means it can’t run.



9.2. Program File

Program files are files included in the project and they can be programmed, including Basic file, PLC file, HMI file (.bas, .plc, .hmi)

Note: you need to create or open one project file (.zpj) at first, then open / build program file.

9.3. ZAR File

ZAR file is a kind of encryption file, the suffix is .zar. After project item generated ZAR file, code can't be found. Now, you can download ZAR file into controller, the methods of downloading ZAR file refer to "ZAR Downloading".

9.4. Library File

Library file is generated from "Make Lib" and is saved for program protection, program won't be modified. Library file's name with suffix ".zlb", the methods of compiling Lib file refer to "Make Lib".

9.5. ZML File

ZML file is used to identify hardware device's functions. This is only for EtherCAT devices. And ZML file is generated by our Zmotion small tool, then save generated ZML file into controller in RTSys.

9.6. Font / Library File

In RTSys, you can customize the font, add needed font file into RTSys project. Font file name suffix is ".ttf" / ".zft".

Chapter X Download & Run Program

10.1. How to Download Program (RAM / ROM)

According to "Downloading" in Chapter II, there are 2 downloading methods, RAM & ROM. For details, please refer to chapter II corresponding information.

Here, let's conclude the **downloading steps**: create project – create file – select file type – set auto run task No. – open the file – edit the file – connect to controller – download RAM / ROM.

10.2. How to Run Program Automatically

When RTSys connects to the controller/simulator to run the program, you can set whether the program runs automatically or not at this time, that is, auto tun task No., right-click file name to set in "task number setting" window. And before that, please check how many tasks the controller supports in "controller state" window or in corresponding user manual. Because each controller with different task numbers.

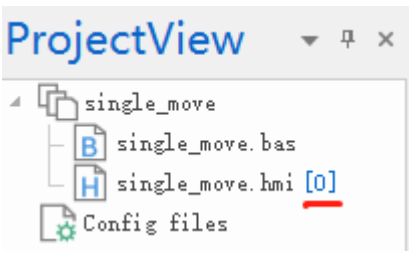
In one project, one auto run task No. must be set, if not set, program can't be run, and the print message is "WARN: no program set autorun."

Notes:

a. Generally, the auto-run task No. is set before downloading the program.

b. It is best to set only one auto-run task No. for a project file, and other tasks are opened using the RUNTASK instruction or the RUN instruction.

c. Please don't set same task No., and the task number has no priority and is only used as an identifier.



10.3. ZAR Downloading

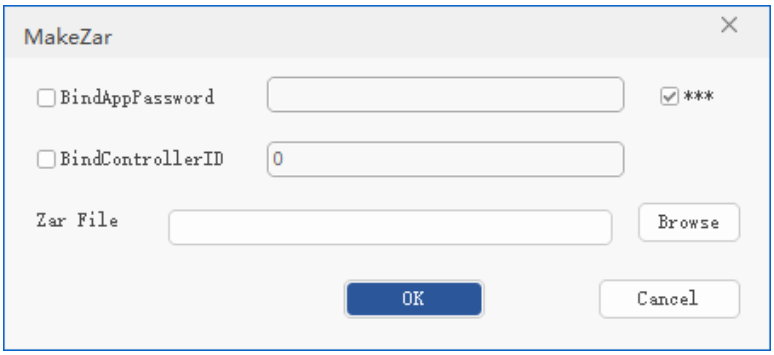
By generating a specified ZAR encryption file, you can achieve independent program downloading, so that you can pass the download file to the end customer without worrying about program leakage. The file suffix is .zar. What's more, in RTSys, you also can bind the controller ID (the controller ID is the unique serial number of the controller when it leaves the factory). After binding, the ZAR file is only used for this controller. And if there is .c file, please select compile platform / customize gcc compile options.

(1) How to Check Controller ID:

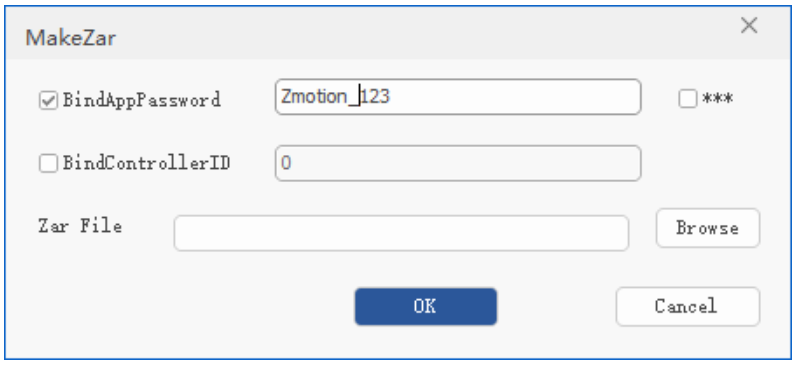
After connected to controller, check ControllerID in "controller state" window, or enter "?SERIAL_NUMBER" command in "command & output" window.

(2) How to Generate ZAR File:

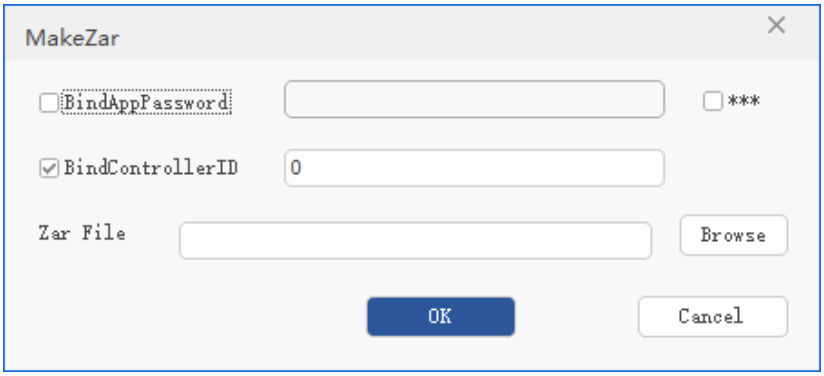
a. After debugging the program, open "MakeZar" file through "Controller" → "Make ZAR File". Then, you can choose to use password binding or controller ID binding to encrypt and generate (if you choose both, while downloading, two must be met).



b. If you check "BindAppPassword", please enter the password (the password supports letters, numbers, and some special symbols such as "_", and can be set up to 16 characters). If you are not sure whether the password you enter is consistent with what you want, you can uncheck the "***" after the input box.



c. If you check “BindControllerID”, please enter controller ID (The ID of each controller is different).



d. After setting the encryption method, click "Browse" on the Zar file item, select the save path for the Zar file, and click "OK".

Notes: Remember the password! Because it can’t be found if you forget it! For controller ID, it is unique and can’t be modified!

(3) How to Download ZAR File (2 Methods):

➢ Method 1: Download ZAR File in RTSys

The ZAR file encrypts and packages the entire project, so you do not need to download the project when downloading the ZAR file.

- Open RTSys, connect to correct controller (if the ZAR file is bound with controller ID, controller must be bound one).
- Use APP_PASS command to verify: in “output” window, enter “APP_PASS(your password)” and send it out.
- Click “controller” – “Down ZAR File”. Then select the position where puts the ZAR file, and choose it to open.
- If password or controller ID are correct, in “output” window, success information will be

shown (if encryption is made by “bind with controller ID”, while downloading ZAR file, it will verify the controller ID automatically, and only when the ID is correct, ZAR program can be downloaded successfully).

- If fail to download, there is one window that shows “download fail”. Please check password, controller ID.

➢ Method 2: Download ZAR File in U Disk

- Open RTSys, and connect to controller at first, and insert the U disk that saves ZAR file into controller U disk interface.
- Use APP_PASS command to verify the password. Same step 2 as above method
- Use FILE command’s “LOAD_ZAR” function to load U disk’s ZAR file: in “output” window, enter “FILE “LOAD_ZAR”, “filename”, and send it out (filename: ZAR file name, must be English characters).

10.4. Make Lib

This function, like ZAR, belongs to the category of program encryption. The "Make Lib" function can compile a program file into a library file and save it to facilitate program confidentiality or prevent modification. The library file's file name suffix is ".zlb" and the library file can only display global SUB definitions.

(1) How to Compile Lib File:

a. After debugging, click “File” – “Make Lib”, then choose one file to compile it as library file (can compile multiple files at the same time).

b. Choose compiled Lib file, and save it.

(2) How to Download Lib File:

a. Open / new build one project file (.zpj), and add compiled Lib file into the project again – in RTSys left “project view”, right click to choose “add to project”, and find the Lib file to add it..

b. Set auto run task No. for Lib file, click “Download ROM / RAM”, Lib file can be downloaded. After that, for Lib program, you can see the program definition statement at the beginning, but only global definition can be checked.

10.5. Compare Controller Program

Through this function, you can check current PC side program file and controller side program file, and compare them, whether they are consistent, for example:

It can be seen left part is PC side file, and the column 1 is file name and type information. Right part shows controllers files. The last column tells you whether they are consistent. YES means the same, NO means they are different.

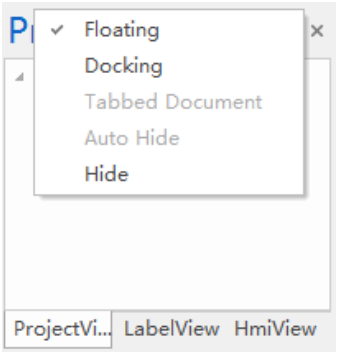
Note: controller files can’t be uploaded into PC for encryption.

Chapter XI Right-Click Shortcut Menu

11.1. RTSys Right-Click

(1) Right-Click "Project / Label / Hmi View" Window

In RTSys, right-click any view (ProjectView / LabelView / HmiView) window, you will see below window information.



a. Floating: switch current selected window to "floating", that is, it is floating, can be dragged to any RTSys position.

b. Docking: fix current selected window at default position in RTSys.

c. Auto Hide: hide the current selected window to the edge of the RT Sys interface and form a small label. When the mouse points to the label, the window pops up, and when the mouse is not pointing to the label, the window is hidden.

d. Hide: hide the current selected window, that is, close it (if you want to see it, please click this window again in "View".

(2) Right-Click Program File

In RTSys, when you opened the program file, you can right-click program name:

a. Closs: close current selected file.

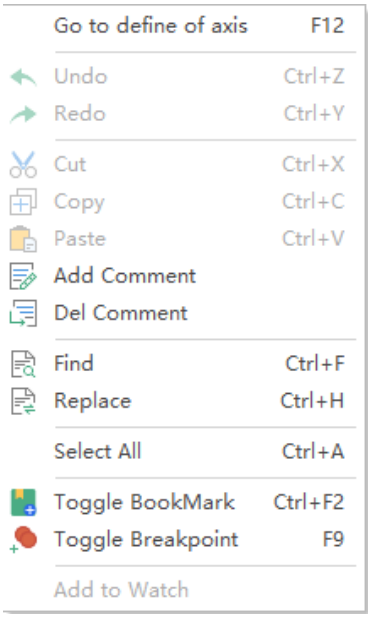
b. Save: save current selected file.

c. Closs All: close all current opened files.

Note: closed files can be opened again – double click the file again in Project View.

11.2. Right-Click Basic

In Basic editing window, right-click to open below window.



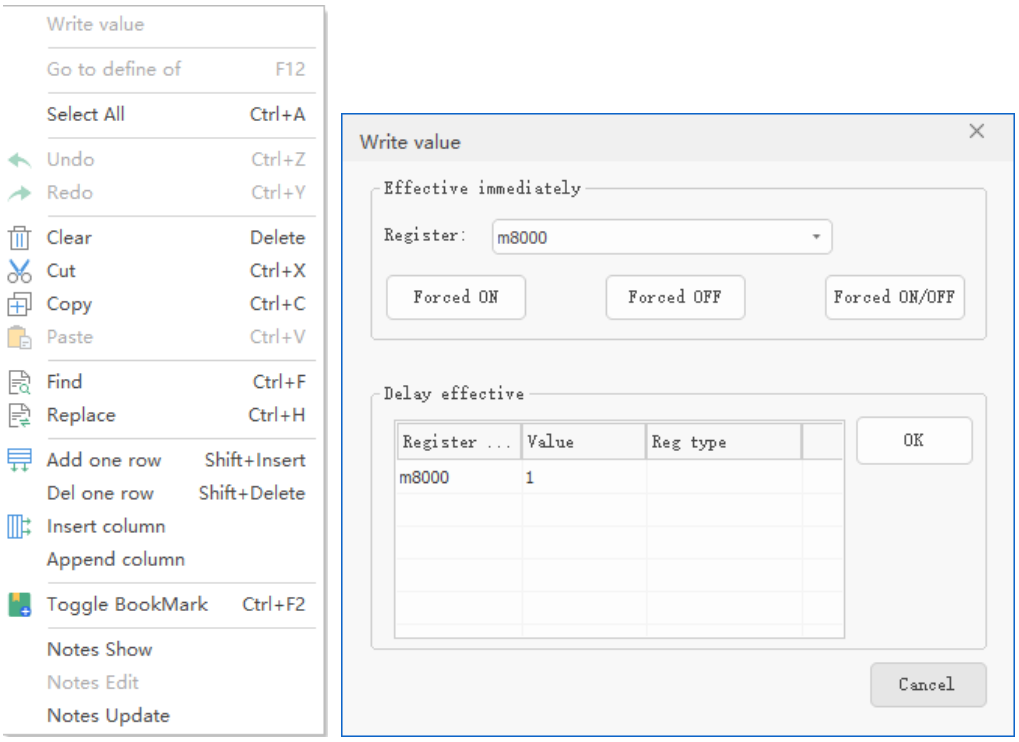
a. Go to define of axis: you can quickly go to corresponding SUB function position.

b. Select All: select all contents in current editing interface.

c. Add to Watch: after opening debugging, you can choose variable and add to “watch” window, real-time changes can be known.

11.3. Right-Click PLC

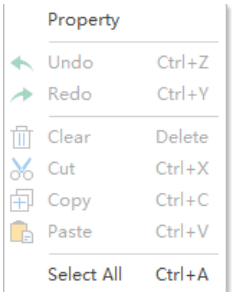
In PLC editing window, right-click to open below window.



- Write Value: after starting debugging, manually modify values of soft component or register.
- Add one row: insert one row above the current line.
- Insert column: insert one column at the right column.
- Append column: append one column at the end of PLC window (Dotted line)
- Note Edit: used to note PLC file's register.
- Note Show: show edited notes at the bottom of corresponding soft component.

11.4. Right-Click HMI

In HMI editing window, right-click to open below window.



Property: pop up current selected component's property window, you can check and modify information.

Chapter XII How to Set RTSys Showing

12.1. Status Bar

In status bar (View – Status Bar), there are 3 parts of contents: cursor positioning, message printing statistics, controller model / IP / state showing.

(1) Cursor Positioning

It will show information of cursor position, for example, for Basic, “Line:26 Col: 17, Chars: 788”.

(2) Print Information Showing

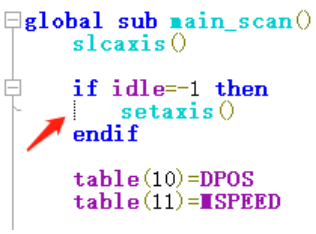
It will show now error, warn, message numbers in OUTPUT window.

(3) Controller Information Showing

It will show currently connected controller model, controller IP address, and controller running state. If ALM or ERROR, corresponding details will be shown also, and highlight in red background. If no controller connected, it will show “Unconnected”.

12.2. Indent Line

Indent lines are used to align program lines with the same indentation when there are multiple layers of nested indentation in a Basic program, making the program more hierarchical and standardized. By checking this function, you can choose whether to display alignment lines in the Basic program editing interface. The arrows in the figure below are indent lines.

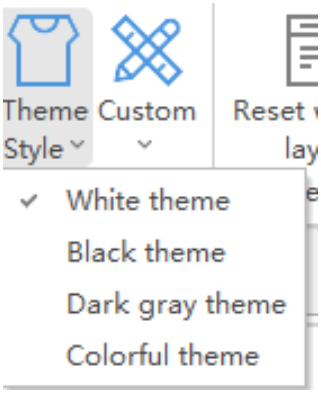


12.3. Auto LineFeed

"Auto Line Feed" means that when the RTSys software window is scaled, the Basic program content will wrap according to the window size. You can choose whether to check this function.

12.4. Theme Styles

RTSys supports switching theme style, there are 4 theme colors, white, black, gray, colorful. If you want to custom, please refer to.

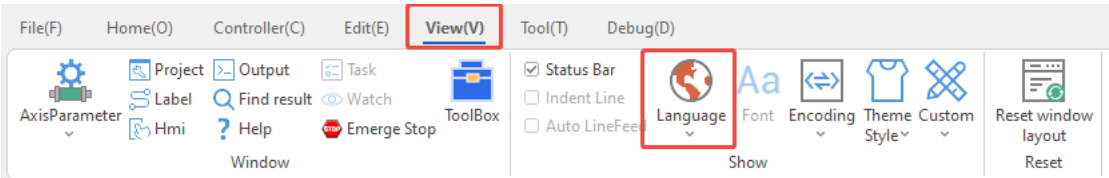


12.5. Switch Languages

RTSys supports switching software languages. There are Chinese and English. If you need to use other languages, you can add needed one.

(1) How to Switch the Language:

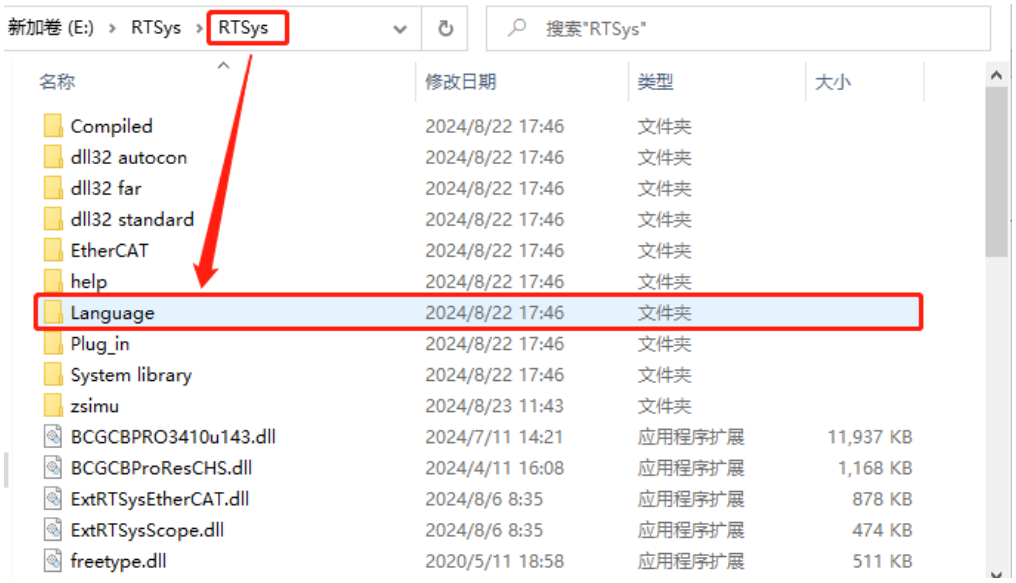
Click "View" – "Language", choose needed one, click it. Then it will tell you "restart" to take effect it, click OK. When you restart RTSys, language become you needed.



Video Description: RTSys Page (https://www.zmotionglobal.com/pro_info_282.html) – System Architecture – E.

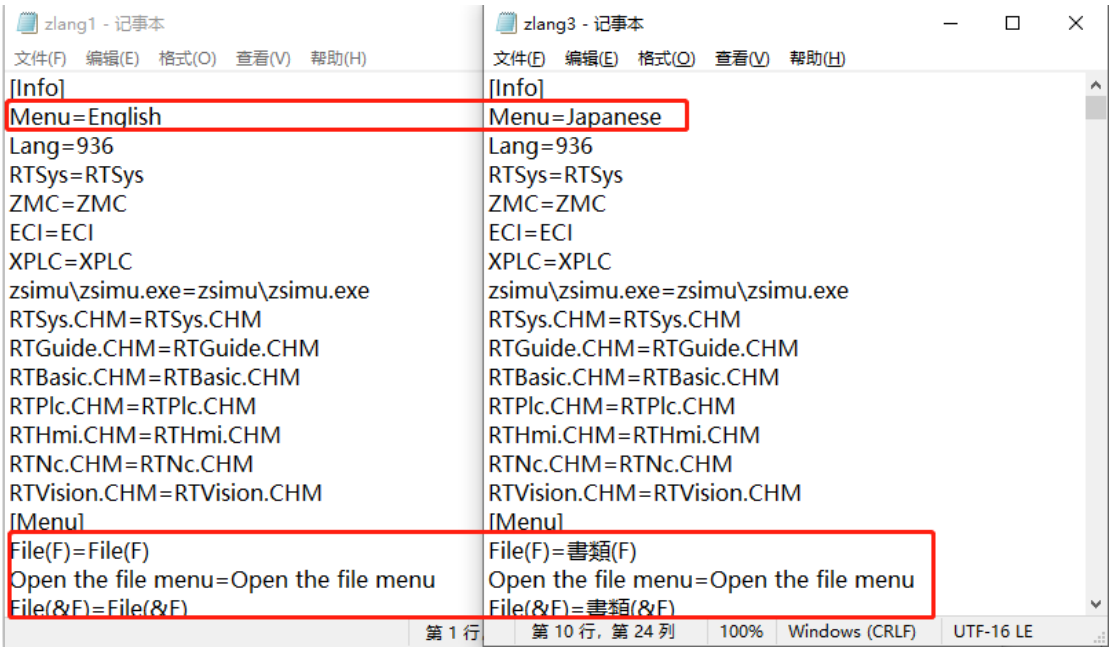
(2) How to Add New Other Languages:

a. Open RTSys software installation path, and find “Language” folder.

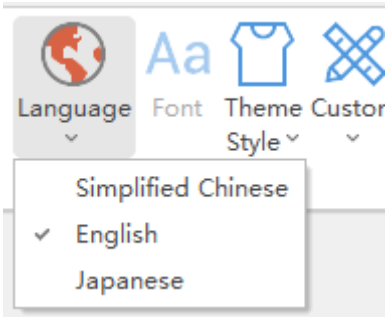


b. Open it, and it can be seen there are Chinese and English files. (zlang1: English, zlang2: Chinese). If you need add new others, you can copy one, then rename it, zlang 3, zlang 4, etc. (the file name must be zlang, and the No. must be one by one – zlang 1, zlang 2, zlang 3, zlang 4, zlang 5…).

c. After that, open the new added file. And do translation, that is, replace as needed language rmation.

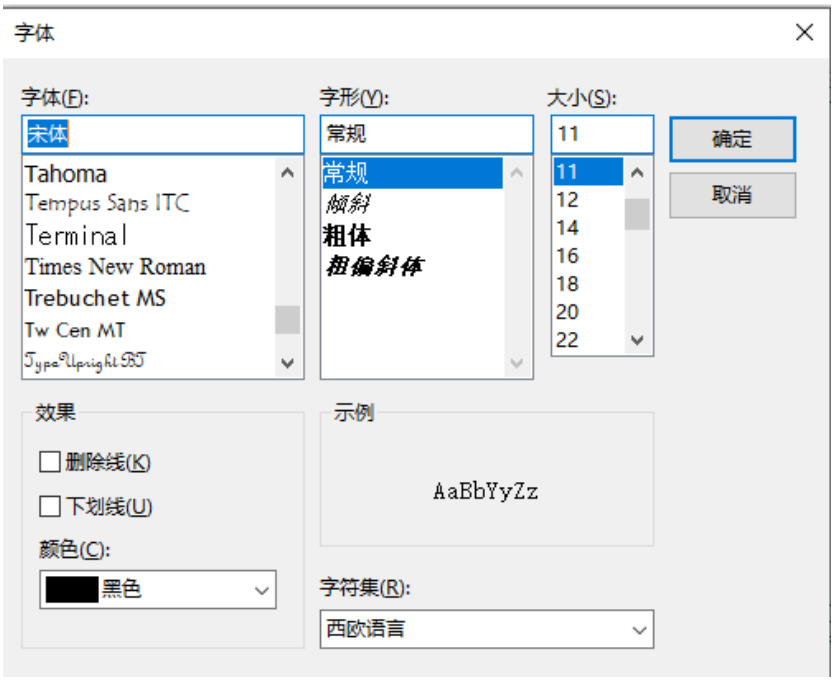


d. Save new language file, and restart RTSys.

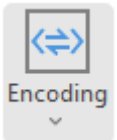


12.6. Set Fonts

In RTSys, you can modify Basic and PLC program file's font. Open it by "View"—"Font".



12.7. Encoding

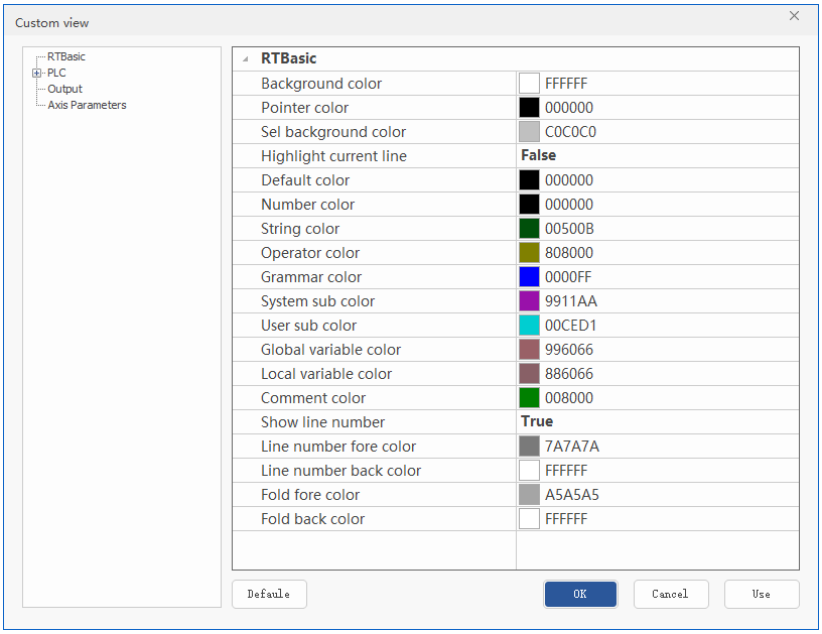


This is mainly to convert encoding formats of Basic program and C language program. Now, it supports ANSI format and UTF-8 format.

Note: controller only supports ANSI encoding format!

12.8. Custom Window

RTSys supports customizing the display style of the programming interface of RTBasic, PLC LAD & IL. It provides a variety of setting items. You can customize the background color, cursor color, line number color, etc. You can also specifically set the function color, variable color, comment color, etc. The axis parameters are used to select the "axis parameters" view to display the parameter situation.



12.9. Reset Window Layout

Reset RTSys software windows to default position. Sometimes, you did modifications for the window position, if you want to use default window layout, use this function, click it, then restart RTSys.

Chapter XIII Common Errors

When errors come in operating project, RTSys will show error information, take it for reference to help debugging, we can also view errors by command “?*task”, double click error information to reach related code lines.

Error	Reasons & Solutions
Download button is gray	➢ You didn’t build the project or you didn’t open project .zpj file. Please create the project at first, then add the program file into project. ➢ Controller is not connected to RTSys. ➢ Some controllers only support downloading into ROM, that is, RAM is not supported.
WARN: no program set autorun.	The program file has no auto run task No.
Error:5002, Operate Failed!	There are multiple HMI tasks, please remove others, remain one.
Fail to Connect to Simulator	➢ Simulator is deleted by antivirus software. Therefore, please set simulator as “safe one” in antivirus software, then connect again. ➢ Simulator port No. is occupied (default is 502). Please check which one also is 502. You can find as follow methods: ➢ Win + R – enter cmd to open window. ➢ Enter command: netstat-ano, find 502 port, and remember corresponding PID (| findstr <port No.>) ➢ Keep inputting task|findstr “(PID value in last step)”, press Enter button, then see which one uses 502. ➢ In “task manager”, find corresponding PID program, and close it.
Controller ALM Led is ON.	➢ Enter “?*task” in output window to check error information again. ➢ Check axis state “AXISSTATUS”, whether it is 0. ➢ Check the information printed in “output”

	window.
Motor doesn't run when in "manual" motion.	➢ Check AXISSTATUS parameter, if it is not 0, please see corresponding position, especially position limit, soft position limit, alarm position. ➢ Check ATYPE, see corresponding axis, whether it is 0, whether configured as encoder. ➢ Check UNITS, if it is 0 or minimum value. ➢ Check motor wiring.
Motor only moves to one direction when in "manual" motion.	➢ Check AXISSTATUS parameter, if it is not 0, please see corresponding position, especially position limit, soft position limit, alarm position. ➢ Check if drive dual-pulse configuration is consistent with controller, controller pulse mode is configured by INVERT_STEP. ➢ Check motor wiring, especially directional cable.
Non-China operation system uses RTSys Chinese version, then appear "Garbled".	**1) <u>Situation 1: RTSys menu & window name can't show normally / are garbled.</u>** **Reason:** Different systems are with different encode formats, then sometimes Chinese can't be identified. **Solutions:** ➢ Win + R, then enter "cmd" to open interface. ➢ Enter the command "chcp", then you can get this system's code page No., for example, China land is 936 by default. ➢ After that, under RTSys installation path, open "Language" folder, then open "zlang2.dat" file, find code line "Lang = 936", change 936 as needed code No. (checked from cmd in step 2). **2) <u>Situation 2: RTSys menu bar is normal, but the project name, file name in projectview, and output window are abnormal.</u>** **Reason:** There is Chinese character of the project path. **Solution:** edit the storage path as English, there are below

	2 methods: ➢ **Method 1:** In Chinese system PC, modify the path as English, and open the project, make the project name as English also. Then, open it in other PC, it will be normal. ➢ **Method 2:** Step 1: open the project folder, use "notepad" to open .zpj file, and search filelist. Step 2: make all file's name as English, for example, if File 1 = 主程序.bas, change it to File 1 = main.bas Step 3: save it, and use other tools (like, notepad++) to modify the encode format as UTF-8, also please save it. **3) <u>Situation 3: file contents are garbled.</u>** **Reason:** Program file's encode format is not correct. ➢ **Solution 2:** Open it in the PC that can show normally, then click "View" – "Encode" in RTSys, set it as UTF-8, then send it to current PC. ➢ **Solution 2:** Use other tools to make the encode format as UTF-8, then open it by RTSys.
Open project in RTSys, connect to VPLC7XX controller, it reports an error "20020", or MotionRT software crashed when updating FPGA firmware.	Too large "Total Memary" configured in MotionRT7, which exceeds system memory. Please reduce Total Memary of MotionRT7.
Scope is abnormal.	➢ Imported .txt file that is not scope parameter. ➢ Imported scope file contents have been modified.
After opening xplcterm, it reports error of "open lcd failed"	➢ Send "?set_xplcterm" to see whether turn on the screen automatically, 1 means it has already opened, if 0, please restart the controller. ➢ Whether several xplcterm screens are opened, if it is xplc screen of RTSys itself, please open the

	plug-in management, click “restart”.
When you installed new version, it tells you “no zmotion.dll, …”	RTSys is closed when installing, DLL is also closed, which causes system delay, deleting “zmotion.dll”. **Solution:** ➢ No need to re-install it, just copy one new zmotion.dll file from RTSys installation contents, which can be found through RTSys\dll32 standard, please choose it as needed.
When your system is not Chinese system, files can’t be found.	The project has Chinese path, which causes garbled text. **Solution:** ➢ Modify the file that includes Chinese name.
Remote library download failed in RTSys.	The controller firmware is too old. **Solution:** ➢ Please use the firmware that is after 2025 for remote library.
When connecting to simulator I RTSys, it doesn’t run after clicking simulator “config” (设置) button.	The simulator speed is > / = 80, which will cause simulator running not smooth. **Solution:** ➢ In RTSys register table, set suitable simulator speed: double click “SPEED”, set the value as <50 (hex system). The RTSys register table path: 计算机\HKEY_CURRENT_USER\SOFTWARE\ZSimulator\zsimu\SIMUSET
In RTSys – SCOPE, it reports the error off “no memory”.	No “write” permission. **Solutions:** ➢ Open RTSys installment folder, and open RTSys.exe property, then click “safe” – “edit”. ➢ In “group / user name”, find your name, and check below “write” permission (actually, these 4 operations should be allowed: read, read & exe, write, modify).

Appendix A: Menu List

Note: some menu names are with “Hyperlink”, you can click it to check details.

“File” Menu List:

Name	Image Mark	Description
New		Build new project file and select file type (different programming languages correspond to own file types)
Open…		Open existed project file (only can open .bas / .hmi / .plc file)
Close		Close currently opened project file
Close All		Close all opened project file
Save		Save current project file into current project path
Save As…		Save current project file into the other path
Save All		Save created project files into current project path
Make Lib		Compile created project file as .zlb file (it only supports Basic file and PLC file)
New Project		Build new project and save it to corresponding path
Open Project		Open created .zpj project
Close Project		Close currently opened project

“Common” Menu List

Name	Image Mark	Description
File		
New		Build new project file and select file type (different programming languages correspond to own file types)

Open…		Open existed project file (only can open .bas / .hmi / .plc file)
Save		Save current project file into current project path
Save As…		Save current project file into the other path
Save All		Save created project files into current project path
Controller		
Connect		Connect to controller / simulator
Disconnect		Disconnect to controller / simulator
Download RAM		Download project into controller / simulator's RAM, don't save when powered on.
Download ROM		Download project into controller / simulator's ROM, it will be saved when powered on.
Edit		
Read-Only		ON / OFF read-only mode, please note it is valid in basic file and plc file.
Go to Last		Jump to last position (the file you opened last time)
Go to Next		Jump to next position
Add Comment		Add comment in basic program file
Del Comment		Delete comment of selected lien in basic program file
Undo		Undo last operation
Do Undo again		Restore undo operation.
Common Tools		
Scope		Watch / debug the program that is running, and it can convert data to graphic, which can show real-time changes.
Register		Real-time watch each register values.
Debug		

Start/Stop Debug		Used to track project running.
Emergency Stop		Stop all axes' motions.

"Controller" List

Name	Image Mark	Description
Controller		
Connect		Connect to controller / simulator
Disconnect		Disconnect to controller / simulator
Download RAM		Download project into controller / simulator's RAM, don't save when powered on.
Download ROM		Download project into controller / simulator's ROM, it will be saved when powered on.
State the controller		Check controller state information: controller basic information, ZCan node information, slot node state, communication configuration, etc.
Firmware controller		Update controller firmware version.
System Time		Check controller current time, and support custom controller time or synchronize with PC time.
Modify IP address		Modify controller IP address, also can check current controller IP address.
Compare Project		Compare current PC project file with controller file, whether they are consistent.
Lock Controller		Lock the controller through password, when locked, host computer program can't be downloaded into controller.
Unlock Controller		Unlock the locked controller, enter the correct password to unlock it.
Reset the controller		Restart the controller, then it needs to connect to RTSys manually after powered on.

Connect		Connect to controller / simulator
Disconnect		Disconnect to controller / simulator
Download RAM		Download project into controller / simulator's RAM, don't save when powered on.
Download ROM		Download project into controller / simulator's ROM, it will be saved when powered on.
State the controller		Check controller state information: controller basic information, ZCan node information, slot node state, communication configuration, etc.
Firmware controller		Update controller firmware version.
	Project	
Compile All		Compile all files under the project, but don't download into controller.
Add to project		Add the file into current project, support adding program file, font file, image, etc.
Settings		Reserved
Make Zarfile		Generate specified ZAR encryption file, it can be encrypted by password or controller ID binding, the file suffix should be .zar.
Down Zarfile		Download ZAR encryption file into controller ROM.
Notes		Note the register of project file.
Indicator		ON / OFF ALM led of connected controller.

"Edit" List

Name	Image Mark	Description
	Edit	
Paste		Paste clipboard's content into project file.

Cut		Cut project file's selected program content / elements to clipboard temporarily.
Copy		Cut project file's selected program content to clipboard temporarily.
Delete		Deleted selected content of the file.
Add Comment		The whole line that is selected can be noted.
Del Comment		Delete notes of selected line.
Insert one Tab		Add one tab for the line where the cursor is.
Delete one Tab		Delete one tab for the line where the cursor is.
Go to Last Position		Jump to last position
Go to Next Position		Jump to next position
Undo		Undo last operation
Do Undo again		Restore undo operation.
Read-Only		ON / OFF read-only mode, please note it is valid in basic file and plc file.
Bookmark		
Toggle Bookmark		Set / delete the bookmark for selected line in file.
Last BookMark		Jump to last bookmark of the same project
Next BookMark		Jump to next bookmark of the same project
Edit BookMarks		Check file and line No. of the bookmark that was set, and it can edit the bookmark.
Find / Replace		
Find		Find needed content according to entered keyword (the range can be customized)
Replace		Replace content according to entered keyword (the range can be customized)

"View" List

Name	Image Mark	Description
Window		
Axis Parameter		ON / OFF "axis parameter" window, it can check commonly-used parameters in motion control.
Project		ON / OFF "project view" window, it can check how many files, file types, and auto run task No. in the current file. And support axis configuration, EtherCAT node configuration, etc.
Label		ON / OFF "label view" window, then it can check all SUB functions defined in basic file.
Hmi		ON / OFF "Hmi view" window, then it can check window information and component information included in Hmi file.
Output		ON / IFF "Find result" window, used to show results.
Find Result		ON / OFF "output" window, it can check content, print running result, online input command, etc.
Help		ON / OFF "help" window, used to show help documents.
Task		ON / OFF "Task" window, it will show when debugging. It can check each task's details.
Watch		ON / OFF "Watch" window, it will show when debugging. It can view variables, registers.
Show		
Language		Change RTSys showing language, there are Chinese and English, after choosing, please restart it.
Font		Set program file's font format, size.
Theme Style		Set RTSys software showing type (there are 4 styles).
Custom		Set window custom formats (there are 4 windows).
Reset		
Reset window		Reset software window layout, resume as default,

layout		please restart it.

“Tool” List

Name	Image Mark	Description
Scope		Watch / debug the program that is running, and it can convert data to graphic, which can show real-time changes.
Manual		Set axis parameters to operate the motor manually and directly.
In		Real-time watch IN state.
Op		Real-time watch OUT state
Register		Real-time watch each register values.
Image		Used to show and check the image in the latch channel, or change latch channel’s image.
AD/DA		Watch AD/DA values
PWM		Set / read PWM’s duty and frequency value
SDO		Write and read EtherCAT data dictionary into controller.
Troubleshooting		Watch controller state and show trouble shooting information.
Bus state diagnosis		Diagnose and show diagnosis information of EtherCAT and RTEX bus states.
Plug-in		Add custom small plug-in, there are “xplc screen” HMI simulation plug in by default.

“Debug” List

Name	Image Mark	Description

Start/Stop Debug		Start / stop debugging program and task.
Go		Run the program that has already opened “debug”.
Pause		Pause the program that is debugging.
Run to Cursor		Set the program run to which line.
Step Into		Jump to next command.
Step Over		Jump over next command.
Step Out		Jump out SUB subprogram.
Breakpoint		Add / delete breakpoint in Basic program.
Emergency Stop		Stop all axes’ motions.

“PLC” List

Name	Image Mark	Description
LD		Usual-on contact to connect with bus line.
LDI		Usual-off contact to connect with bus line.
LDP		Used to detect the rising edge of the normally open contact connected to the busbar. It connects a scan cycle only when the rising edge of the specified bit soft element (when it changes from OFF to ON)
LDF		Used to detect the falling edge of the normally open contact connected to the busbar. It connects a scan cycle only when the falling edge of the specified bit soft element (when it changes from ON to OFF)
STL		Starting command of program that uses stepper LAD command.
Compare		Compare two data, that is, compare operand S1 with operand S2 according to assigned condition. When the

		condition is met, contact is conducted, if not met, contact is off.
OUT		Command that drives soft element coil.
Function		Open PLC command input list, select the command.
LBL		Build PLC subfunction, which is used as entry of subfunction.
Horizontal line		Add LAD horizontal line.
Vertical line		Add LAD vertical line.
Horizontal line clear		Delete LAD horizontal line.
Vertical line clear		Delete LAD vertical line.
ToIL		Convert LAD to IL (instruction list)
ToLad		Convert IL to LAD (ladder of diagram)
Register usage list		Check registers usages and notes under current project.
Cross reference table		Check how to use types of registers and position for current project.
Add one run		Insert one row above the selected grid.
Insert column		Expand one column on the left side of selected grid.

"HMI" List

Name	Image Mark	Description
New Window		New build one Hmi window.
Import Window		Import existed Hmi window (only can be .hmi form)
Background preset		Preset global window background and element form.

Show thumbnails	/	Show as window thumbnail.
Show details	/	Show as window & element details.
Control Class		Open/hide “control class” window, save all HMI files, which can be called directly from control class.
Text Lib		Set texts with multiple languages at once and save them to call in element.
Picture Lib		Add the picture to picture library, and support calling. There are system picture library and user picture library, pictures are used only for Hmi.
KeyTrans		Bind functions of physical keys and virtual keys.
Arrangement	/	Sort multiple elements, there are many options.
Modifying multiple Addresses		Modify multiple register addresses.
Hmi settings		Preset Hmi system, including starting window, resolution, etc.
Property		ON / OFF “property” window, and it can check / set HMI element / window property information.
Quick picture lib		ON / OFF “shortcut picture lib” window, HMI pictures can be checked, and can be used or deleted quickly.
Layer		Show / hide elements of top, middle, bottom parts.
Grid	/	Show / delete the grid.
Control name	/	Show / hide element name in Hmi window.
Language		Switch the language in text library
State		Switch element state.

Appendix B: RTSys Shortcut Keys

Operation	Shortcut Key
PLC Shortcut Keys	
LAD Add "–"	F8
LAD Delete "–"	Ctrl + F8
LAD Add "|"	F9
LAD Delete "|"	Ctrl + F9
LAD Add one Row	Shift + Insert
LAD Delete one Row	Shift + Del
Convert to IL	Ctrol + I
Convert to LAD	Ctrol + L
Controller Connection / Disconnection Shortcut Keys	
Connect to Controller	Ctrl + Alt + C
Connect to Simulator	Ctrl + Alt + S
Disconnect	Ctrl + Alt + D
Editing Shortcut Keys	
Go to Define Position	F12
Go to Last Position	Ctrl & -
Go to Next Position	Ctrl & +
Set / Cancel Bookmark	Ctrl + F2
Edit Bookmark	Ctrl + M
Last Bookmark	Shift + F2
Next Bookmark	F2
Find	Ctrl + F / Shift + F4
Replace	Ctrl + H
Debugging Shortcut Keys	
Add / Delete Breakpoint	F9
Start / Stop Debugging	Ctrl + F5
Run Debugging	F5
Run to	Ctrl + F10
Step Into	F11
Step Jump	F10
Step Out	Shift + F11

Cut, Copy, Paste, Delete, Select All, Redo, Resume, Open, Save, etc.	General shortcut keys
Help	F1
Interface Operation Shortcut Keys	
"Connect to Controller" Window	Enter button means connect
"Command & Output" Window	"↑", and "↓" can check history, Enter button to send command.
HMI Thumbnail	Del – delete, "↑" – last one, "↓" – next one.
Label Interface	"Enter" means jump
Scope Interface	↑, ↓, ←, → -- viewed waveforms
HMI Drawing	ESC to cancel drawing
HMI Interface	↑, ↓, ←, →: move one pixel for selected control, + Shift can quick to move (2 pixels). + Ctrl can align controls.
LAD Interface	↑, ↓, ←, →: switch selected grid, + Shift can add / delete selected area, Home button can return to the first column, End button can back to the last column, PgUp button can jump to first row, PgDn button can jump to last row, Enter button means enter information.

Appendix C: How to Configure EtherCAT

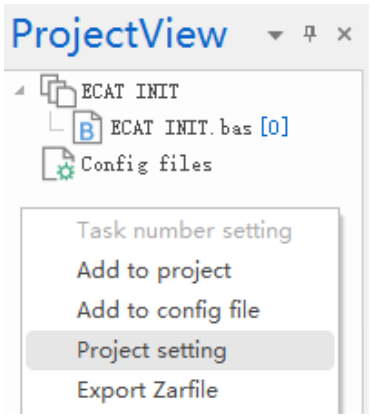
For each window and menu, please refer to "Project View".

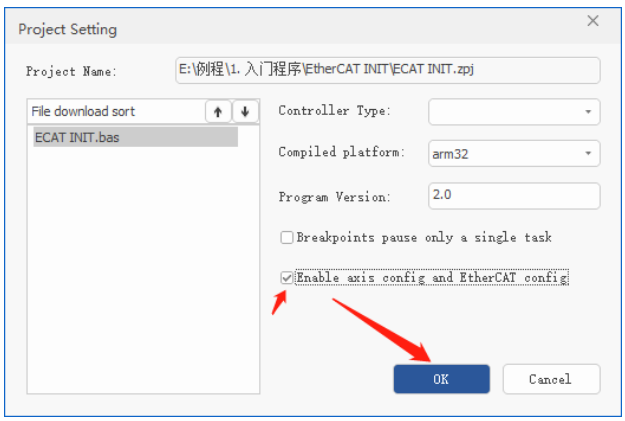
Note: EtherCAT can be configured when it has project or without project. If there is no project, after configured, save and export configured parameters. If there is project, you can save it in this project directly.

With Real Drive Device

(1) Connect RTSys to controller.

(2) Enable axis configuration and EtherCAT configuration functions (right-click empty place of Project View, choose "project setting", and check it, click OK).





(3) Add drive device's .xml / .zml file (if controller and RTSys has already known your drive model, of if you don't need to configure drive parameter, please ignore this operation).

Generally, when the device is only shown "Drive n" after scanning, the drive is not known by controller and RTSys, which means you need to add corresponding file (.xml / .zml) according to below method (1).

➢ Two methods:

- Open RTSys software file path – open EtherCAT folder – put needed .xml / .zml file into here – in RTSys project view, right click controller – update xml/zml list.
- In RTSys project view, right click "config file" – select "add to project" – select needed .zml file (this method only add zml format file).

--difference between two methods--

Zml file added by method 2 will be downloaded into controller, for method 1, it will not download but can be identified.

(4) Scan the device. In RTSys Project View, right-click EtherCAT node / EtherCAT – 0, then select "scan device" (if there are several slot No., please select corresponding slot No. to scan, for example, EtherCAT-0, EtherCAT-1, etc.). If no device is scanned, and report an error "Online command warn, Slot return error:3205", please refer to following "solve device unscanned".

(5) Configure bus period, axis mapping relation, digital IO, analog mapping relation. In RTSys project view, double click controller, then manually configure parameters.

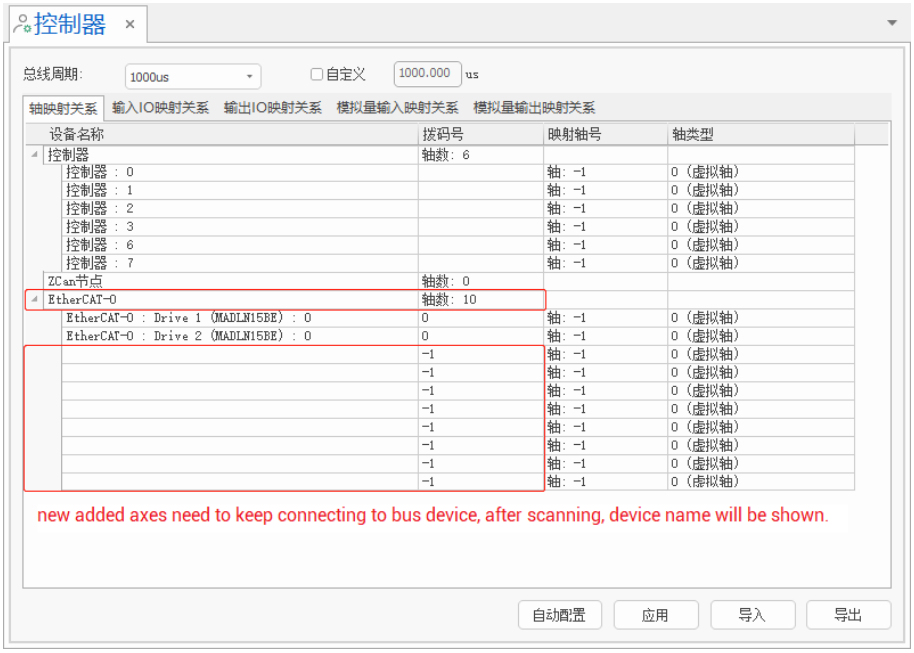
--how to operate "axis mapping relation window", other IO, AIO are same methods--

Tips:

- When the controller supports several axis types, you need to allocate and map axis resources. For example, after scanning, it reads 6 controller pulse axes, and the number of axes expanded by ZCAN is 0, EtherCAT-0 (bus axes) has 2, then please allocate them, set each axis' axis No. (can't be same) and each axis' axis type (ATYPE).
- When "Controller" window has been opened, if you need to add new devices, please re-double-click "project view" – "controller" to update, updating again.

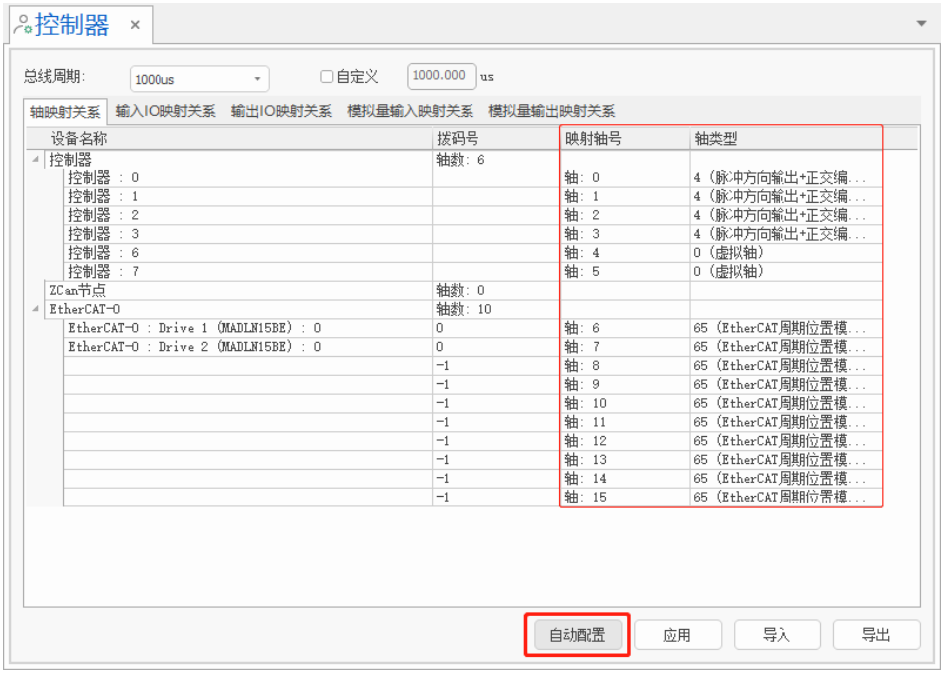
Operations:

a. Assign axis numbers at first. In EtherCAT-0 – DIP Code, modify the total axes as 10 (here, connect to controller that supports 16 axes, so it can add axes).

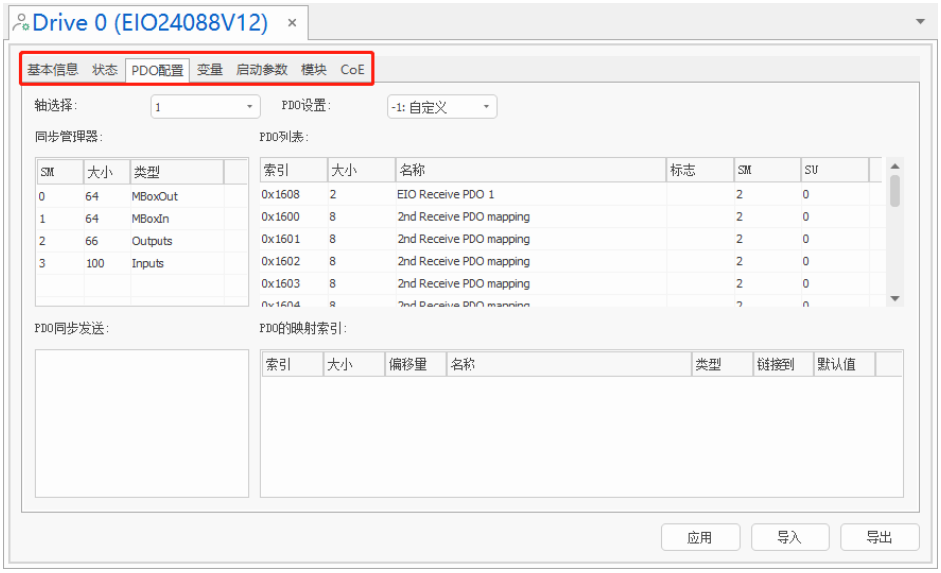


b. Assign each axis' axis No. and axis types. Please set them in corresponding position, if you don't want to manually set, you also can click "auto config" to assign and set.

Note: if connected Zmotion EIO, ZIO expansion modules, expanded axes only can be configured as pulse axis type.

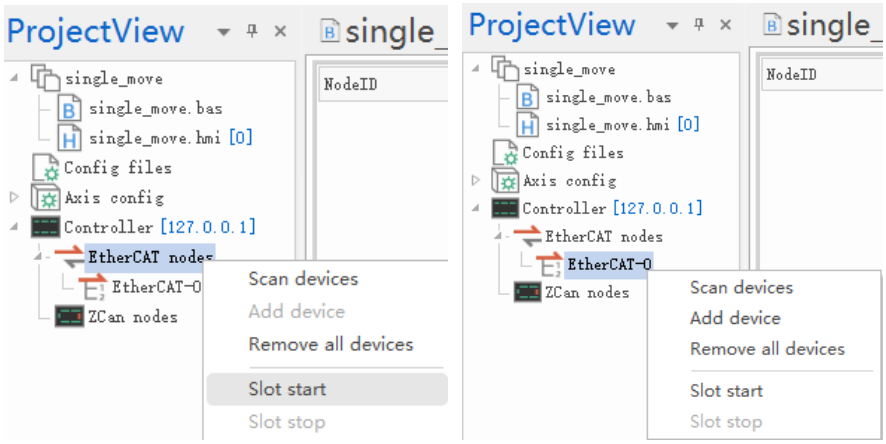


(6) Configure / check each EtherCAT bus device's PDO, ON parameter, etc. In RTSys project view, double click Drive n, then manually set each parameters (Drive n: scanned single device No., for example, "Drive 0", "Drive 1")



(7) Configure axis basic parameters and switch signals. In RTSys project view, single click "axis config" small arrow to open axis list, and double click needed axis No., next, you can do axis parameters configuration.

(8) Open the bus. **When you need to open multiple slot No. devices**, please find EtherCAT node, and right click, select slot start. **When you need to open all devices on the same one slot No.**, please find EtehrCAT-0, and right click, select slot start (if there are several slot No., please select corresponding slot No. to scan, for example, EtherCAT-0, EtherCAT-1, etc.).



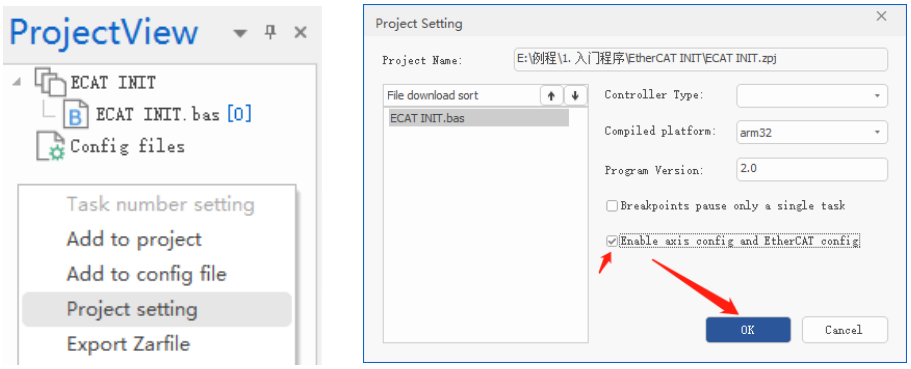
(9) Generate and apply (use) bus initialization file "Startup.bas" file (not "must" step). After generated, download it into controller, which is used when running offline. In RTSys project view, double click axis confg or controller or Drive.n, there is one button of "use" at the right bottom. (if you want to add content into this initialization file, find the file, right click it to set).

Without Real Drive Device

Note: without real device, no way to open the bus (slot start)!

(1) Connect RTSys to controller.

(2) Enable axis configuration and EtherCAT configuration functions (right-click empty place of Project View, choose "project setting", and check it, click OK).



(3) Manually add device. Right click EtherCAT-0 – add device, then select needed one and set numbers, next click add.

(4) Add drive device's .xml / .zml file (if controller and RTSys has already known your drive model, of if you don't need to configure drive parameter, please ignore this operation).

Generally, when the device is only shown "Drive n" after scanning, the drive is not known by

controller and RTSys, which means you need to add corresponding file: open RTSys software file path – open EtherCAT folder – put needed .xml / .zml file into here – in RTSys project view, right click controller – update xml/zml list.

(5) Configure bus period, axis mapping relation, digital IO, analog mapping relation. In RTSys project view, double click controller, then manually configure parameters. Please see without real drive one.

(6) Configure / check each EtherCAT bus device's PDO, ON parameter, etc. In RTSys project view, double click Drive n, then manually set each parameters (Drive n: scanned single device No., for example, "Drive 0", "Drive 1")

(7) Configure axis basic parameters and switch signals. In RTSys project view, single click "axis config" small arrow to open axis list, and double click needed axis No., next, you can do axis parameters configuration.

(8) Generate and apply (use) bus initialization file "Startup.bas" file (not "must" step). After generated, download it into controller, which is used when running offline. In RTSys project view, double click axis confg or controller or Drive.n, there is one button of "use" at the right bottom. (if you want to add content into this initialization file, find the file, right click it to set).

How to Solve "Device Not Scanned"

After scanning, if there appears "Online command warn, Slot return error:3205", which means controller can't verify the drive, then **you need download configured .zml file and program into controller together.**

(1) Add device manually. Right click EtherCAT-0 – add device, then select needed one and set numbers, next click add (if there are several slot No., please select corresponding slot No. to scan, for example, EtherCAT-0, EtherCAT-1, etc.).

(2) Configure / check each EtherCAT bus device's PDO, ON parameter, etc. In RTSys project view, double click Drive n, then manually set each parameters (Drive n: scanned single device No., for example, "Drive 0", "Drive 1")

(3) After configured parameters, export this device's configured parameters and generate .xml file (RTSys project view, double click Drive.n, click export at the right bottom).

(4) Add generated .zml file to the project, and download into controller together (RTSys project view, right click config file, and select add to project). After that, please do axis configuration according to "with real drive" content above