Cockpit 3

INSTALLATION, CONFIGURATION AND CONTROL TOOL

User Manual
Phase Tools release 9.1
Doc. 48000000001 - ENG

SUMMARY

FIGURES.

1. INTRODUCTION

Phase tools is the setup and management tool for Phase Motion Control servo drives (including the Ax Drives Series and TW integrated motors). This manual contains the basic operations of configuration utility "Cockpit". For further details on servo drives features and capabilities, please refer to the corresponding reference manuals.

1.1   Installation requirements

- Operating system Windows XP, Windows 7 32bit/64bit or Windows 8 32bit/64bit
- PC with installed the Internet Explorer 8.0 browser or better
- PC with memory capacity 1GB (suggested) or better
- Minimum with 100MB free space on hard disk

1.2   Installation steps

- Visit Phase Motion Control at http://www.phase.eu in order to download the latest revision of Phase Tools setup
- Follow the setup wizard and click "next step", Phase Tools will be installed in the PC
- After the setup is finished, a new folder of "Phase Tools" will be added on Windows start menu, and the shortcuts of "Cockpit" and "LogicLab" (refer to LogicLab manual for further more information) will be created on the desktop

2. COCKPIT MAIN VIEW

Cockpit is the user interface to configure the digital Ax Drive Series and TW integrated motors. It is intuitive in the way that information is arranged and logically divided into groups for viewing and editing. It allows to create, analyze, modify and copy all parameters useful for applications and regulations of the device.

The configuration utility "Cockpit" operates as drive center of control. The main features are:

- Drive control and identification information;
- Multi-devices control and project management;
- Drive parameters reading and writing;
- Intuitive usage of graphic pages;
- Troubleshooting alarm information;
- Drive firmware updating;
- Initialization and remote control with Control Panel utility;
- Real time diagnostics interface with Softscope;
- Communication by serial interface RS232 with Modbus protocol, or IXXAT CAN interface with CANOpen protocol, or Ethercat protocol.

2.1   Main screen view

Double click the shortcut icon , Cockpit main screen is illustrated as Figure 1. It consists of those main panels: user interface window; catalog window; connection status window; output window; monitor window and status bar.
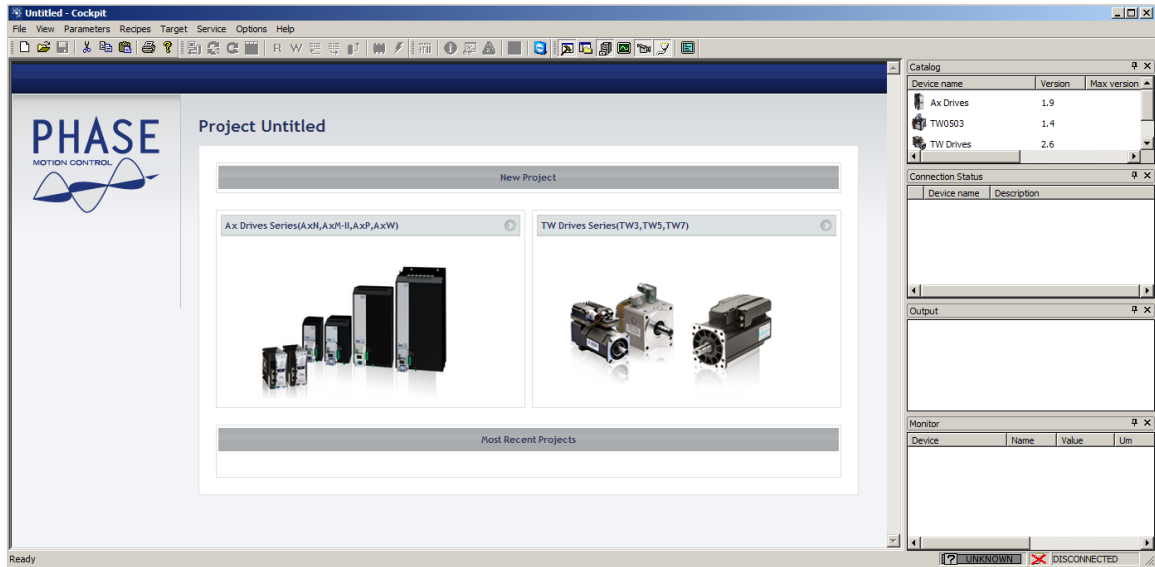
Figure 1 – Cockpit main screen

2.2   User interface window

User interface window shown in Figure 2 is the main part of the main screen. It is used to show pages and parameters grids opened by other controls/tools. It is intuitive and friendly provided that device identification and parameters information, and also let user make fully operations.
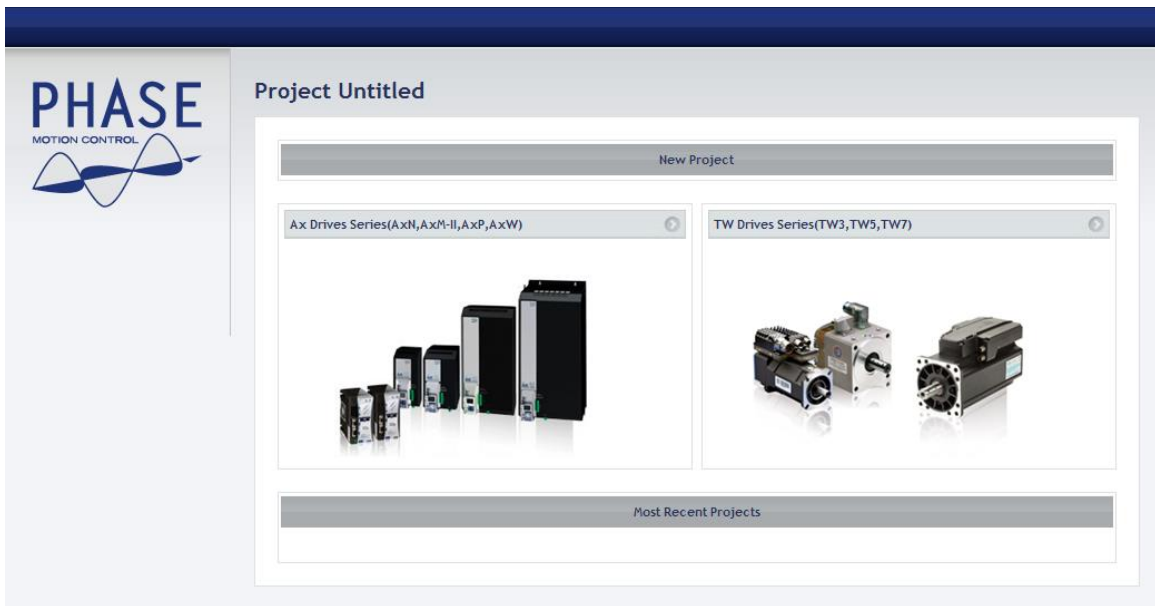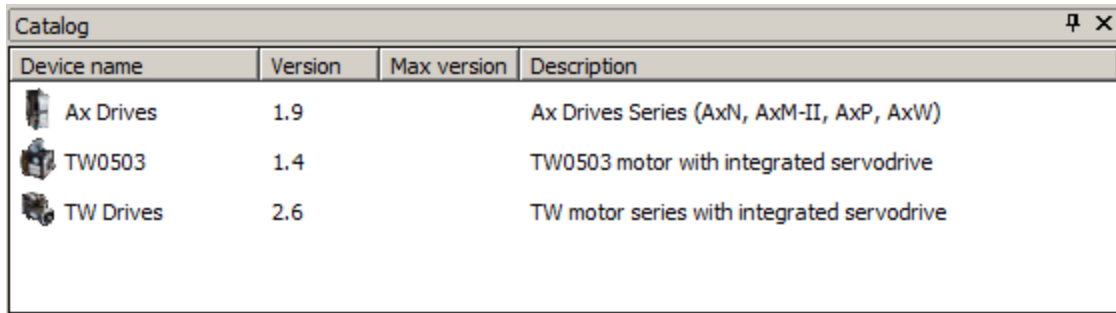
Figure 2 – User interface window

Next chapters will introduce all the features and operations on the user interface in details.

Other configurator windows can be docked on the main window. Refer to menu "View" to show/hide them.

2.3   Catalog window

The catalog window in Figure 3 shows all the catalog objects that can be added to the project depending on the supported product family. It is activated with the item "Catalog" from the menu "View", and also activated with the corresponding button on the toolbar.
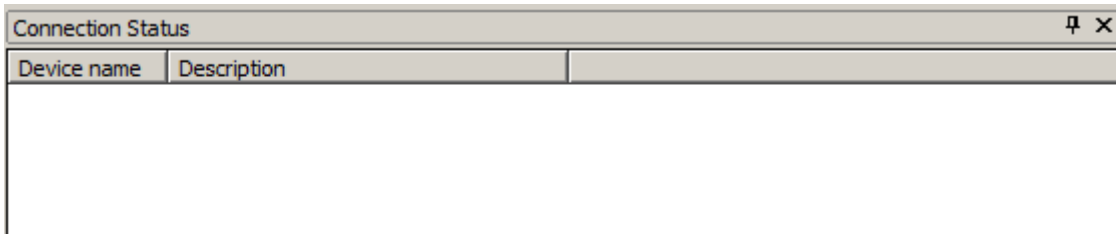
Figure 3 – Catalog window

It is possible to show all the available versions of the catalog items or only the latest version of each catalog item. Refer to menu "Options" to select/unselect "Show all versions in catalog" item.

2.4   Connection status window

The connection status window shown in Figure 4 is to monitor the detailed connection status of each device of the project. It is activated with the item "Connection Status" from the menu "View", and also activated with the corresponding button ![icon] on the toolbar.

Figure 4 – Connection status window

When a project is composed of more than one device, it is very useful to understand what are the devices in error when the connection status indicated in the status bar is "ERROR".

2.5   Output window

The output window shown in Figure 5 is the place where Cockpit report its output messages, and it is mainly used to report the common R/W operations results. The Reading/Writing parameters or polling operations can be logged to show their results on this window.

The output window is activated with the item "Output" from the menu "View", and also activated with the corresponding button ![icon] on the toolbar.
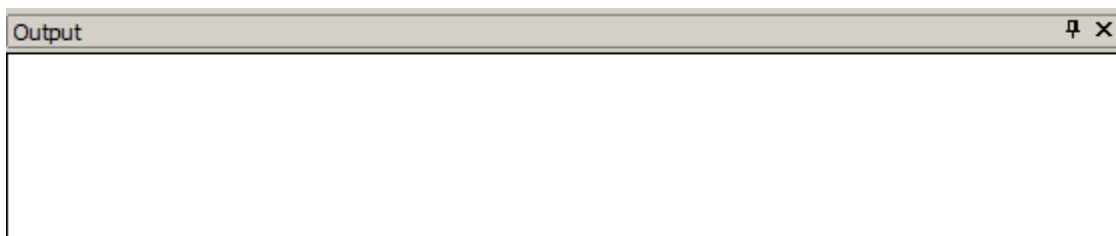
Figure 5 – Output window

It can be also used to log other configurator operations. Refer to menu "Options" to select "Logging options" item.

2.6   Monitor window

The monitor window in Figure 6 is one of the debugging tools supported by Cockpit. It allows viewing the actual values of parameters. The values in the monitor window are continuously refreshed with the values read from the device. Refer to Chapter 10.1 for more details.

The monitor window is activated with the item "Monitor" from the menu "View", and also activated with the corresponding button ▦ on the toolbar.
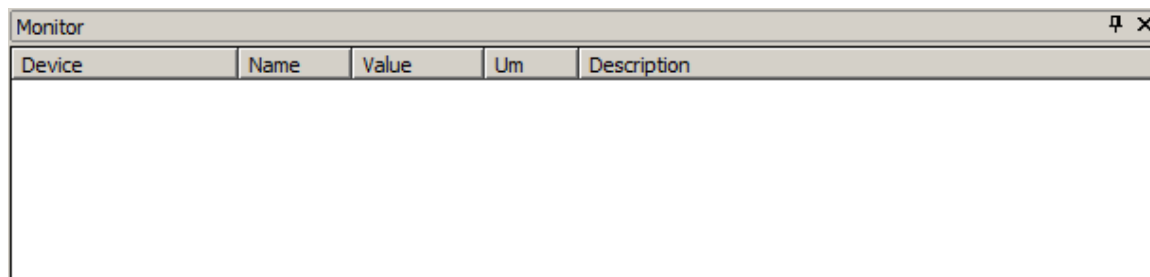
Figure 6 – Monitor window

2.7   Status bar

The status bar in Figure 7 displays the state of the device on the left border of Cockpit window and an animated control reporting the state of communication at the right border.

Figure 7 – Status bar

3. MANAGING PROJECTS

Cockpit is managed as projects and one project consists of one or more devices, which are including AxN, AxM-II family and TW integrated motors.

This chapter explains how to properly work with projects.

3.1   Create a new project

To start a new project, double click the shortcut of Cockpit icon where you install the Phase Tools. The same command is available that is to click "New" in the "File" menu of Cockpit main window and it is also activated with the corresponding button ▯ on the toolbar of an already opened project. The startup Cockpit window is shown in Figure 8.
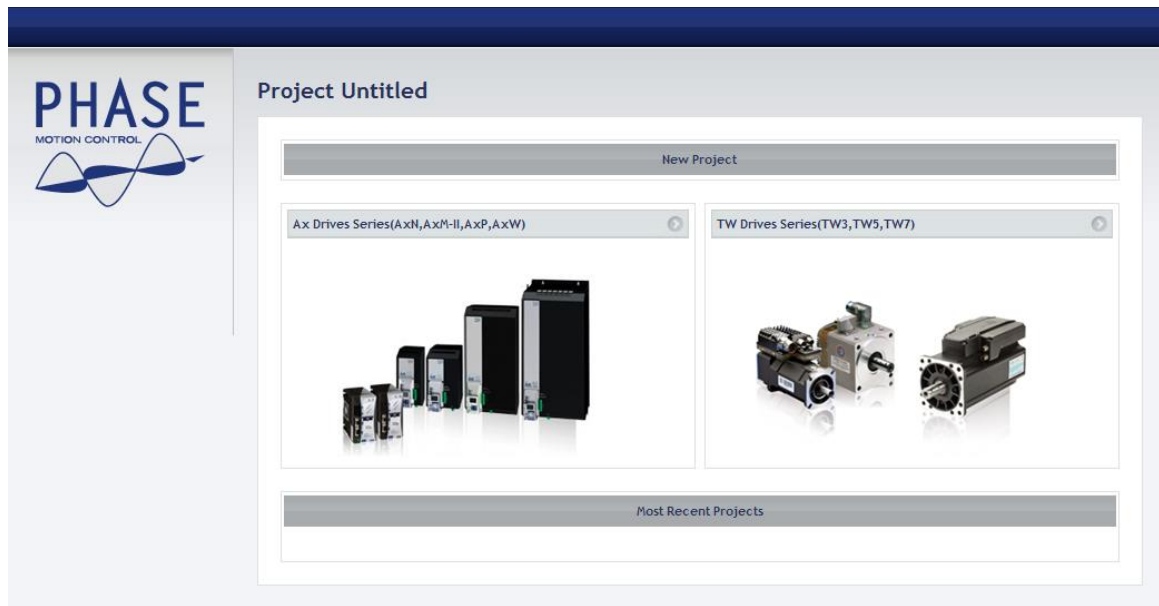
Figure 8 – Cockpit startup window

Cockpit supports two kinds of products series which are Ax drives series and TW series as you can see from the main page. First of all you have to select the target product series.

If your target is Ax drives series, click the "Ax Drives Series(AxM-II,AxP,AxW)" bar on the main page; If your target is Tw series, click the "TW Drives Series(TW3,TW5,TW7)" bar on the main page.

A project at least contains one device. Next section explains how to add a device to the project.

3.2   Add a device to the project

There are two different ways to add a device to the project:

- Adding a device from catalog
- Adding a device from automatic scanning

3.2.1   Add a device from catalog

The device catalog window shows all the necessary devices that could be added to the project. When the current project is new, having no one device, if your selected target is Ax drives series shown in Figure 9.
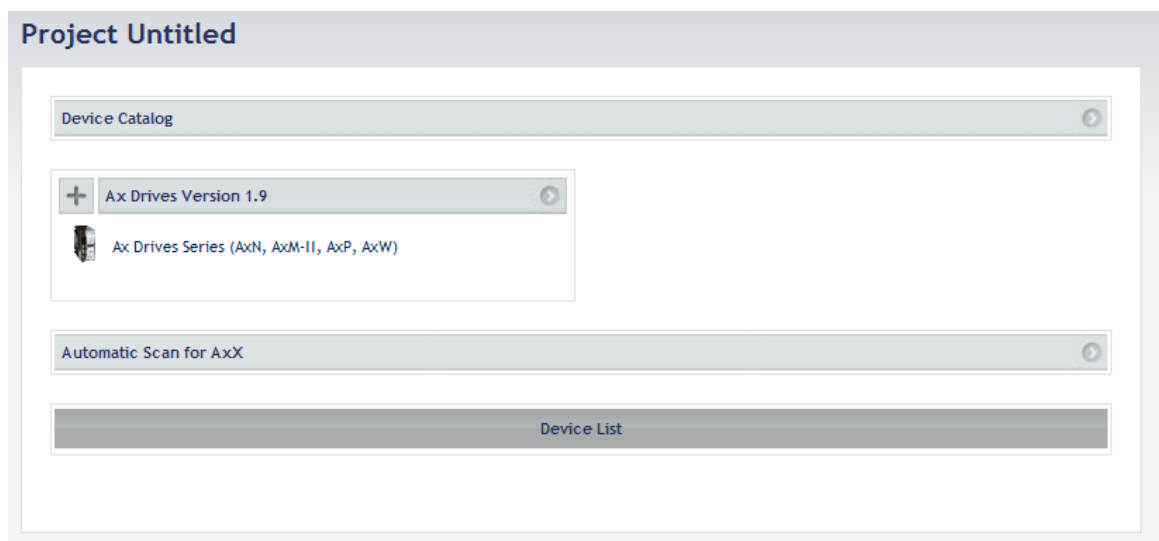
Figure 9 – Ax device catalog

Click the "Ax Drives Version 1.8" bar on Ax catalog to add a new device, the main page of the new device will be shown. You can also click the icon [+] to add a new device. The added device will be shown directly in the device list.

If your selected target is Tw series shown in Figure 10.
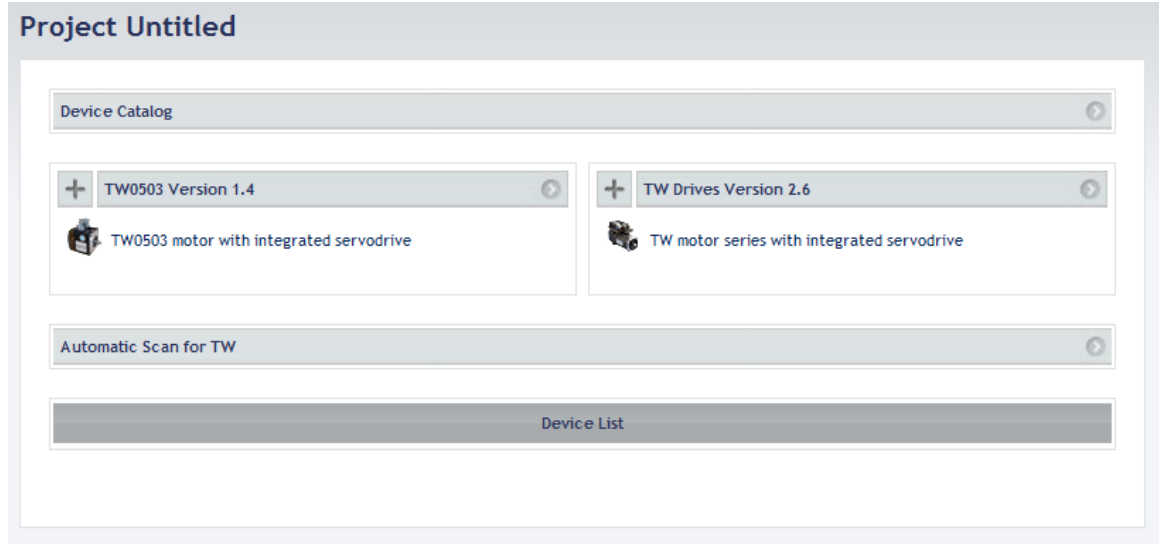
Figure 10 – Tw device catalog

Click the "TW Drives Version 2.6" bar on Tw catalog to add a new device, the main page of the new device will be shown. You can also click the icon [+] to add a new device. The added device will be shown directly in the device list.

When the current project is already existing one or more device shown in Figure 11, you can also add a device from catalog by clicking the corresponding bar or the icon [+] of the target device you want to add. It is the same way as above. The only difference is that, a dialog messagebox will pop up to require you confirming the operation. Click "Confirm", a new device will be added to the project too.
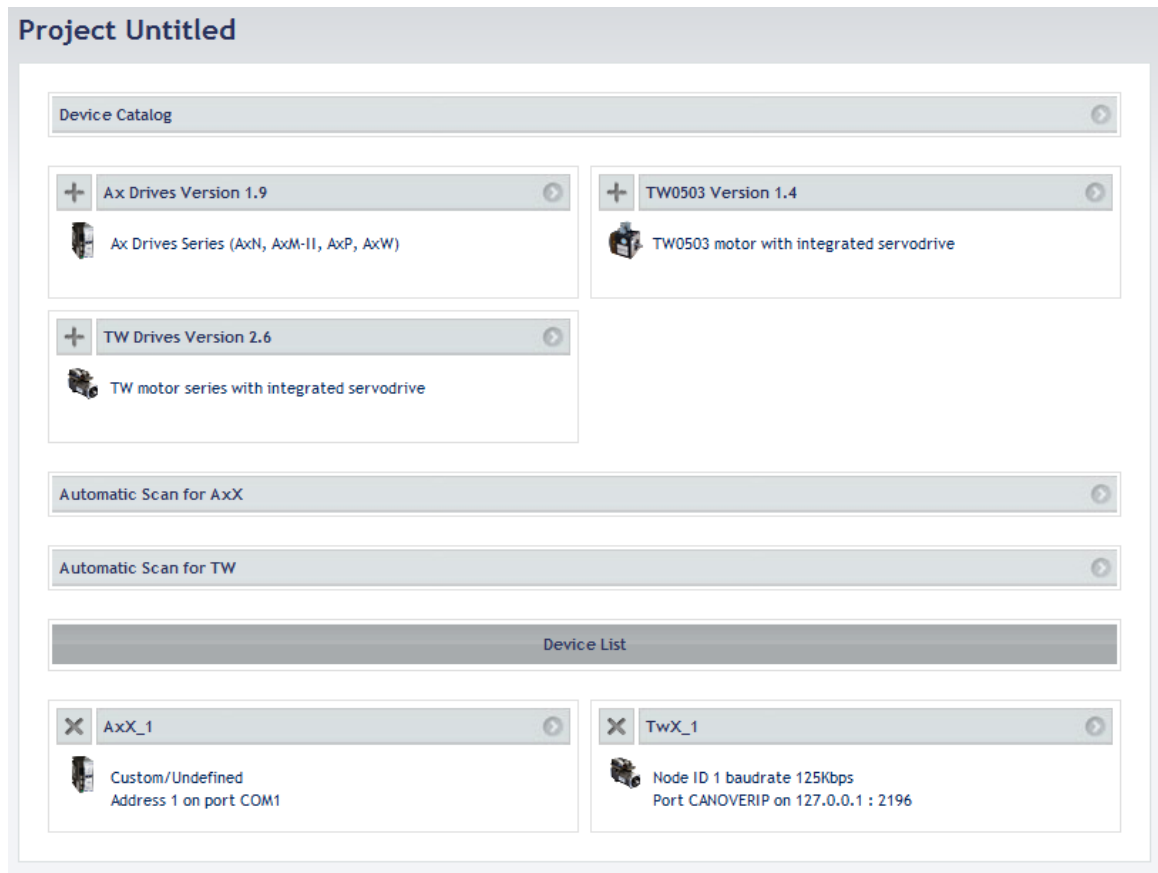
Figure 11 – Catalog list with existing devices

3.2.2   Add a device from automatic scanning

Cockpit provides the automatic scan function, which can find out all the necessary devices on the fieldbus. It is also convenient for users in which condition they do not have enough details for communication with devices, especially, when it is not clear that address id and baudrate of one Ax series drive, or node id and baudrate of one Tw series motor.

The following procedure shows you how to make automatic scanning for Ax drives:

- Click the "Automatic Scan for AxX" bar on the main page. Before starting scan, press the "Advanced" bar to check the communication parameters in Figure 12. Modbus is the default protocol for Ax drives; setup the correct COM port by which PC connects with the device; select the baudrate range **from a starting point to an ending; enter the address range you would like to scan. Press** "Advanced" again, the window retracts back and the settings are persisted. Refer to Chapter 6 for more details on communication parameters settings.
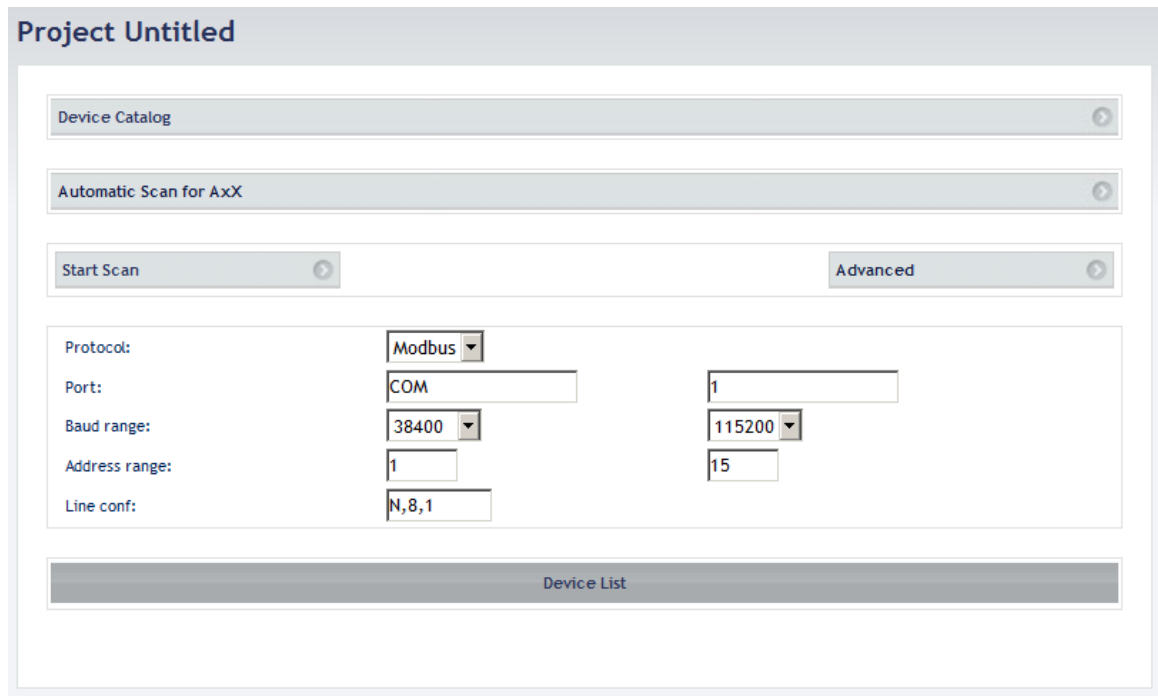
Figure 12 – Automatic scan for AxX

- Make sure the cables physical connections are ready. Then press the "Start Scan" bar, the automatic scan is running. It will scan every baudrate from baudrate range and each address id from address range in Figure 13.
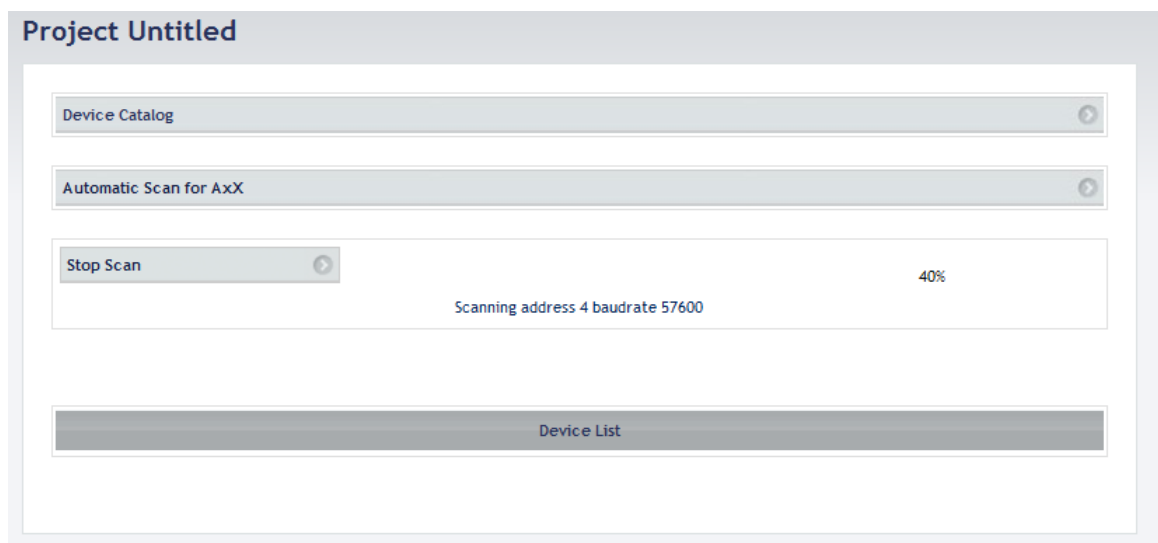
Figure 13 – Start scanning for AxX

- As soon as one device is found out, it will be listed under the "Automatic Scan" bar in Figure 14.
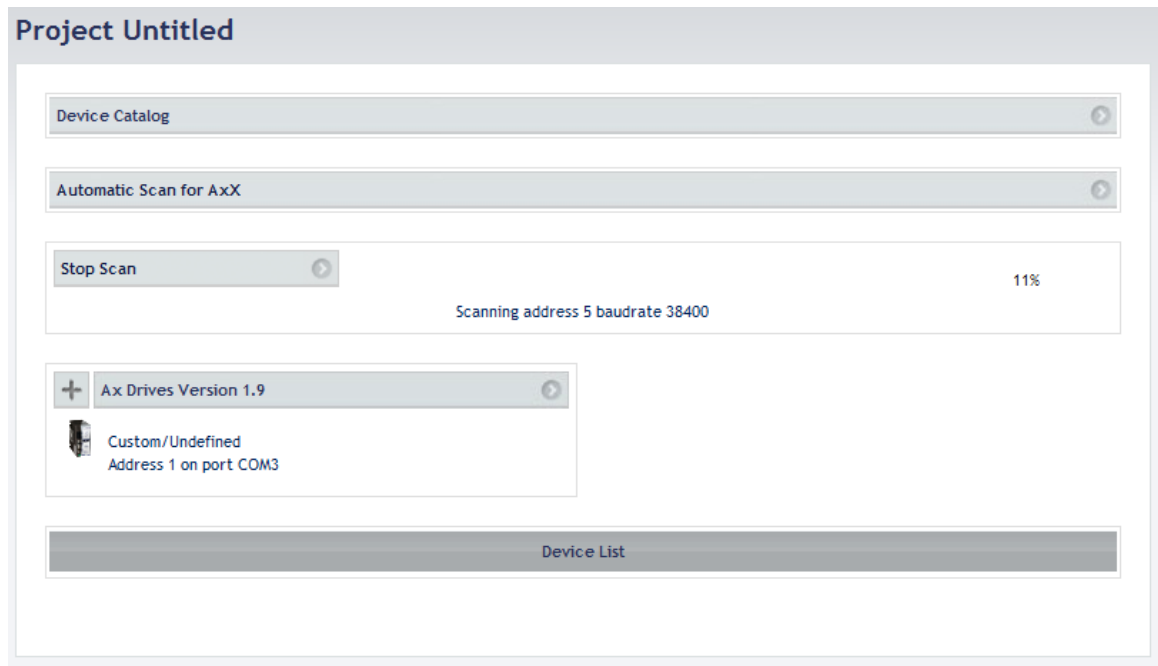
Figure 14 – Scanning result for AxX

- You can stop the scanning whenever you prefer by pressing the "Stop Scan" bar.

Cockpit applies two types of automatic scanning for Tw series:

- Single node scan
- Network scan

Usually when there is only one device of Tw series on the network, Single node scan is preferred to be used. Once it is found out one device, the single node scan procedure stops immediately. Network scan has to be used when there are more than one device on the network.

The following procedure shows you how to make automatic single node scanning for Tw series:

- Click the "Automatic Scan for TW" bar on the main window. Before starting scan, press the "Advanced" bar to check the communication parameters in Figure 15. CanOverIp is the default protocol for Tw series. Refer to Chapter 6 for more details on communication parameters settings.
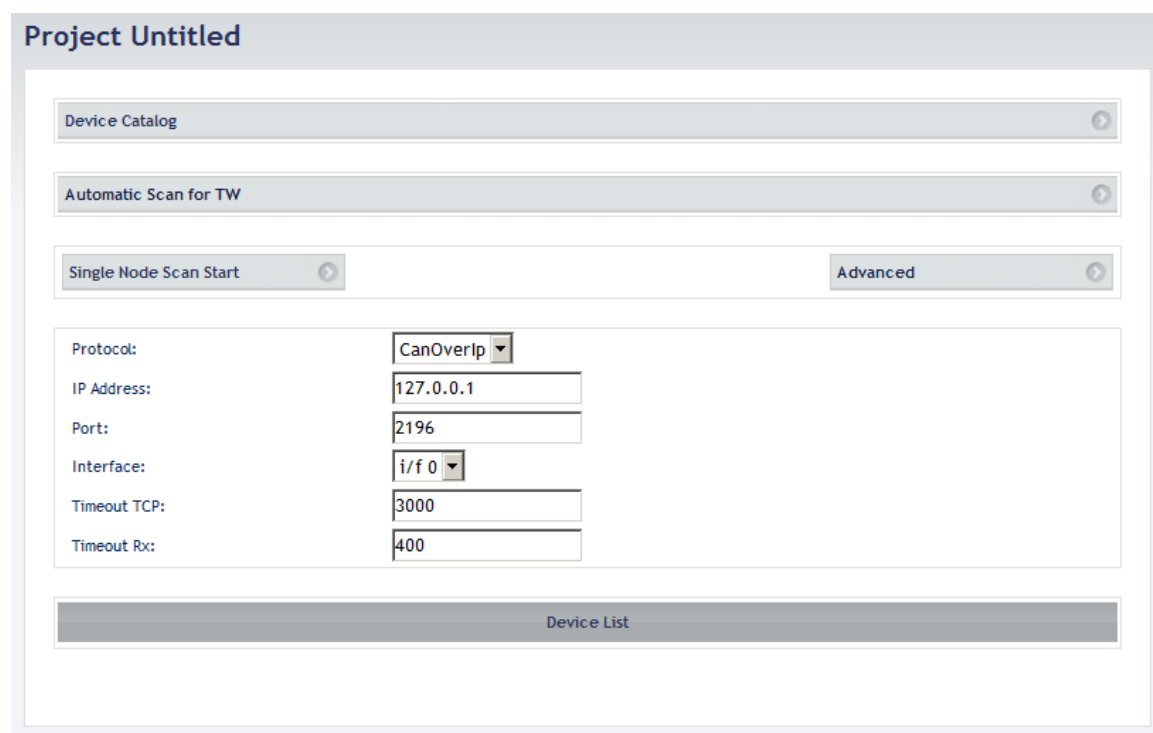
Figure 15 – Automatic single node scan for Tw

- Make sure the cables physical connections are ready. Then **press the** "Single Node Scan Start" **bar,** a dialog messagebox is appeared to warn you that the operation must be done only on single connected device per time. **Click** "Confirm", the automatic scan is running with each baudrate value from the baudrate range in Figure 16.
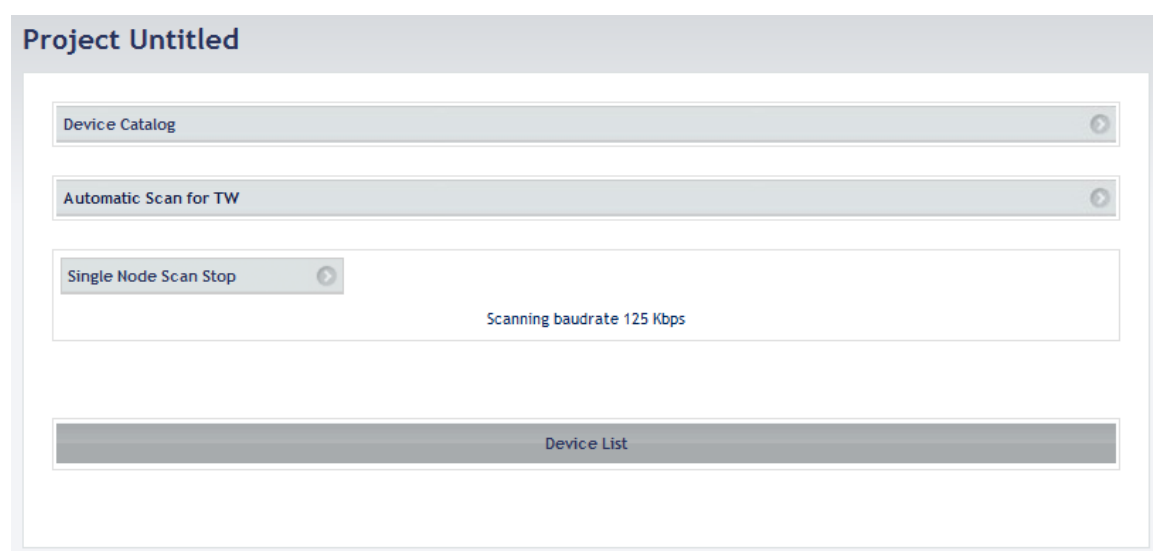
Figure 16 – Start single node scan

- Once the device is found out, the procedure will stop immediately, and the device will be shown below **the** "Automatic Scan" bar in Figure 17.
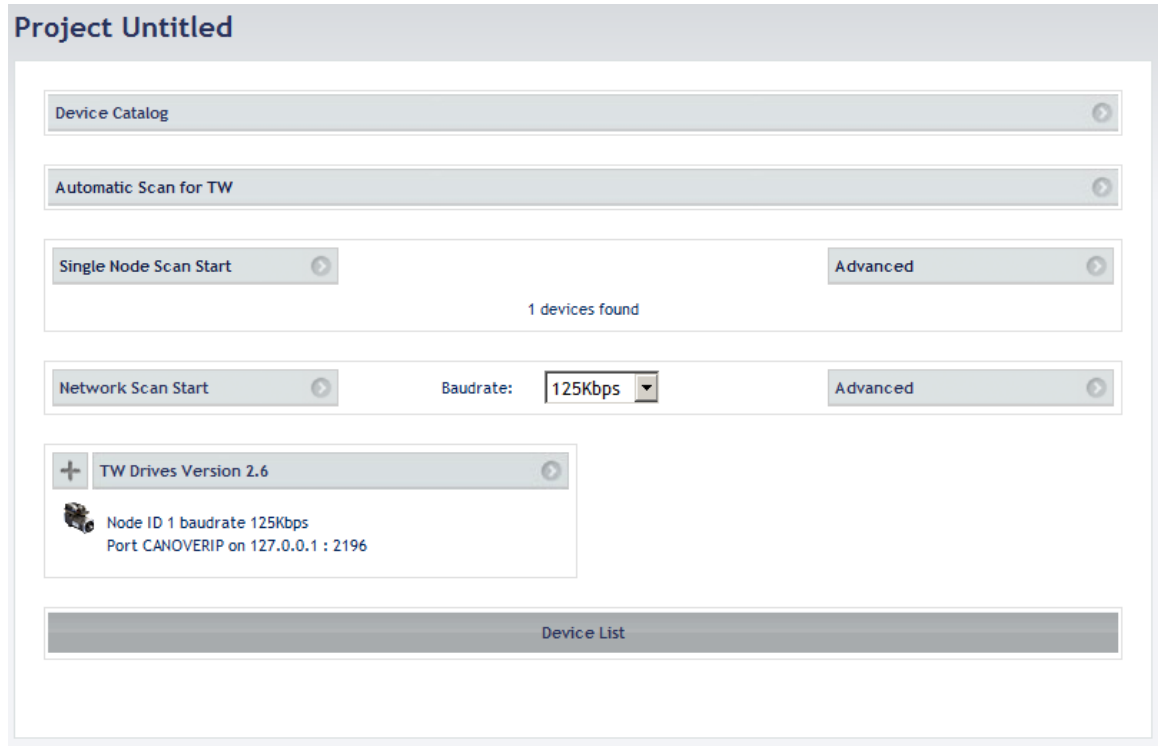
Figure 17 – Single node scan result

- You can stop the scanning whenever you prefer by pressing the "Single Node Scan Stop" bar.

The following procedure shows you how to make automatic network scanning for Tw series:

- Click the "Automatic Scan for TW" bar on the main window. For network scanning, select one scanning baudrate from the baudrate combo box for network scanning.
- Press the "Advanced" bar to check the communication setup parameters before starting scanning in Figure 18. CanOverIp is the default protocol for Tw series. Refer to Chapter 6 for more details on communication parameters settings;
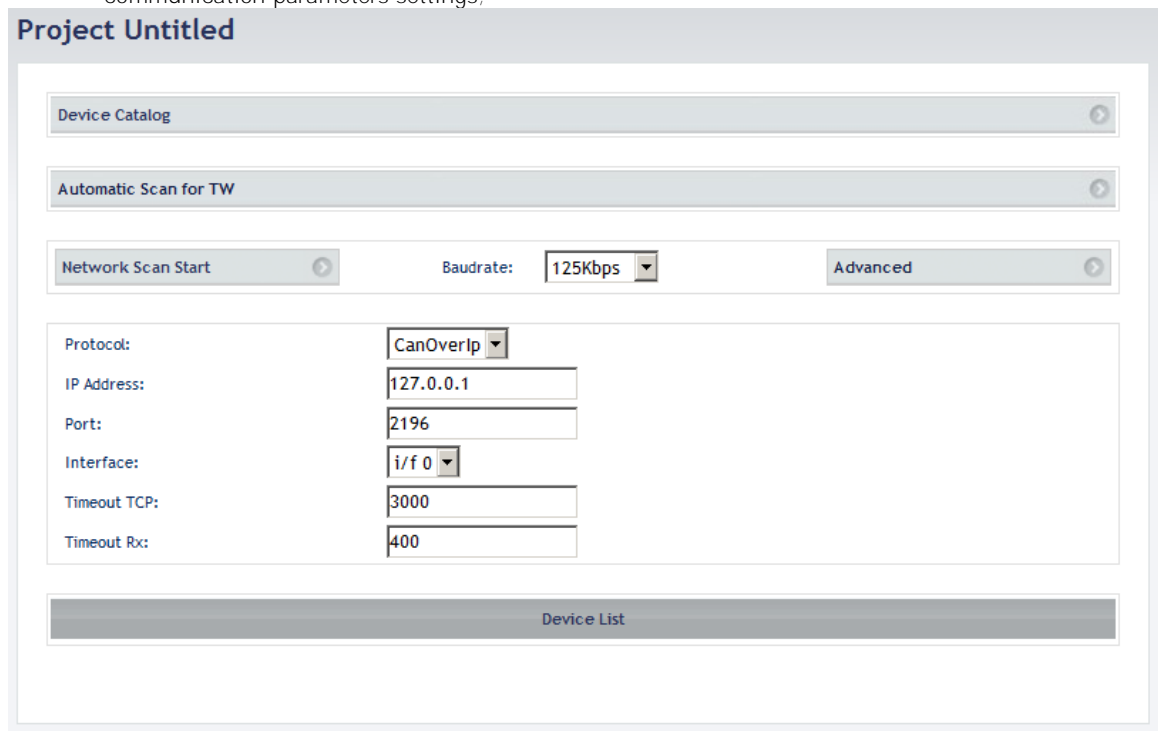
Figure 18 – Automatic network scan for Tw

- Make sure the cables physical connections are ready. Then **press the** "Network Scan Start" bar, the automatic scan is running with the selected baudrate and the node id range from 1 to 127 in Figure 19.
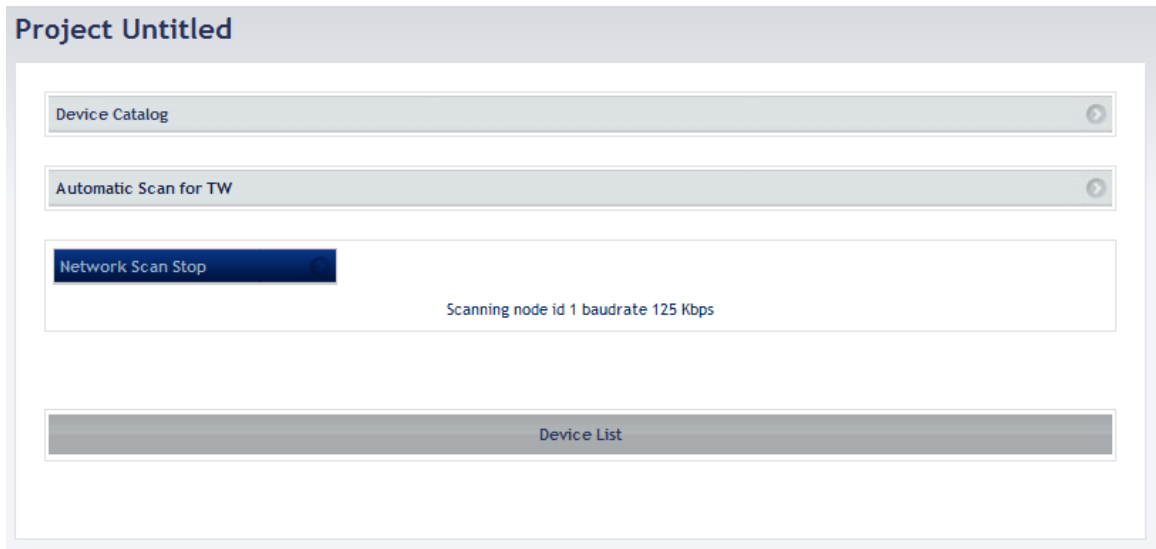
Figure 19 – Start network scan

- When devices are found, they will be listed below **the** "Automatic Scan" bar in Figure 20.
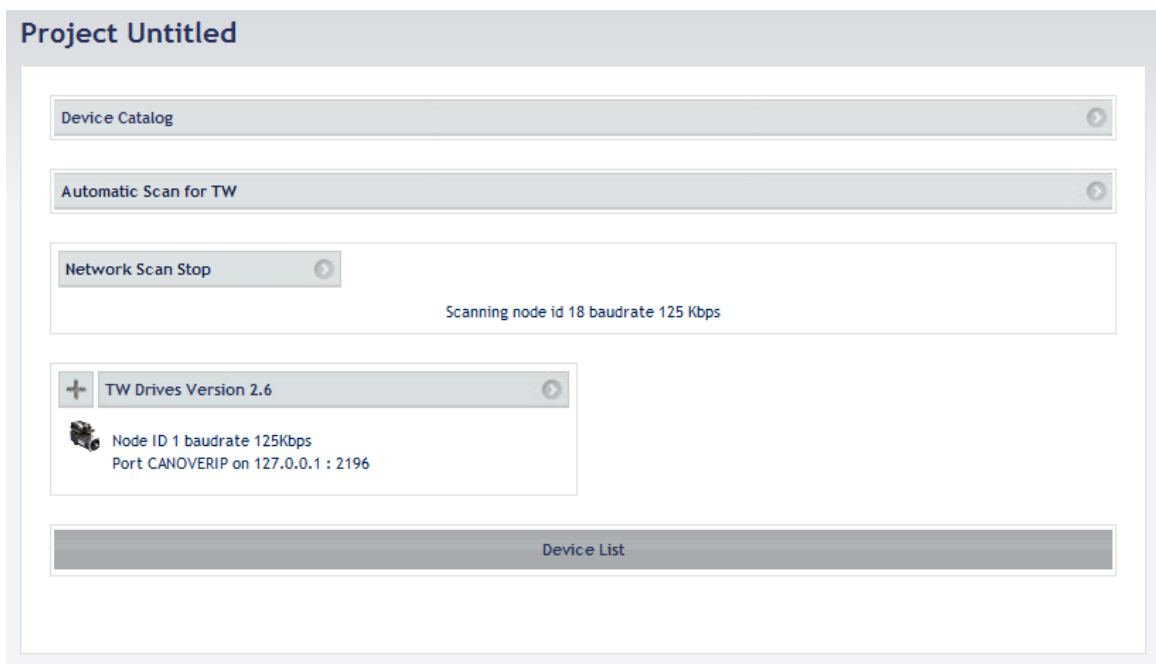
Figure 20 – Network scan result

- You can stop the scanning whenever you prefer by pressing **the** "Network Scan Stop" bar.

Note that such functions are useful when there is only one kind of products series. For example: only existing Ax drives or only Tw series on the same fieldbus. If there are both Ax drives and Tw series on the same fieldbus, such functions may not support.

The found devices are listed in the main window. You can press either the corresponding bar or the icon to add the target device into the current project. The difference is that, if you press the corresponding bar to add the device, a dialog messagebox will appear to inform that you can connect to the drive and update all parameters; if you press the icon, the target device will be added and shown directly under the device list.

Note that if the target device is already added on the project, you can not add it to the same project anymore. When you try to do such operation, a warning messagebox will be appeared.
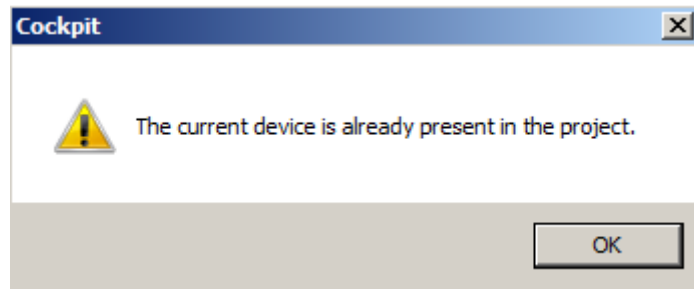
3.3   Delete a device from the project

The device list of the main page arrays all the devices in the current project.

To delete one device, click the corresponding icon on the selected device, a dialog messagebox is appeared to ask if you are sure you want to delete the selected device or not. Click "Confirm", the selected device will be removed from the project.

Note that, the last device in the project can not be removed.
When there is only one device in a project and you would like to remove it, the following messagebox is shown when you click the corresponding icon .
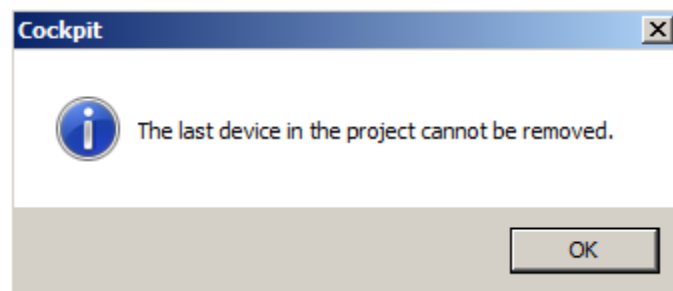
3.4   Open an existing project

To open an existing project, click "Open" in the "File" menu on the main window, or the toolbar icon .This causes a dialog box to appear which lets you load the directory containing the project and select the target project file.

You can also open an existing project from the item "Most Recent Projects" on the main window .

Note that the item "Most Recent Projects" only contains the projects files that is used recently. And if the directory of the project file is moved to another path, the project can not be opened correctly.

3.5   Save the project

3.5.1   Saving changes to the project

When you make any change to the project, you are required to save the project in order to keep the changes.

To save the project, you can select the corresponding item "Save" in the menu "File", or the toolbar icon .
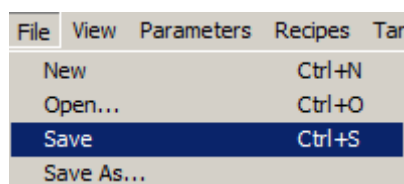
3.5.2   Saving to an alternative location

When you donot want to overwrite the current project file for some reasons, you may save the modified project to an alternative location by selecting "**save as ...**" from the menu "File".
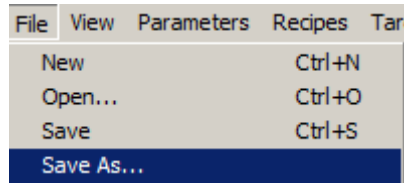
3.6   Close the project

You can terminate the working session explicitly closing the project or exiting Cockpit. In both cases, when there are changes not yet persisted to the project file, Cockpit will ask you to choose between saving and discarding them.

To close the project, select the item "Exit" from the "File" menu; or click the icon ☒ on the right top of Cockpit.
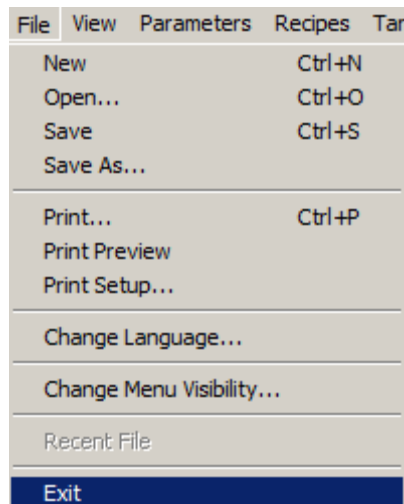
4. MANAGING DEVICES OF AXX FAMILY

AxX family is a programmable platform for brushless servomotors control. It is adequate to control most standard Phase Motion Control brushless motors. The regulation and control functions are managed by the firmware that is integrated into the drive memory. Refer to AxX reference manual for more details.

Cockpit is the dedicated software environment that is parameterizing and monitoring the AxX family operating conditions with a serial interface RS232. Moreover, the graphic interface makes it easy and versatile the drive configuration.

This chapter is to focus on how to manage devices of AxX family. When entering an AxX device, the main page is shown in Figure 21. It is the startup entrance of an AxX device, on which there are all the components of device configurations.
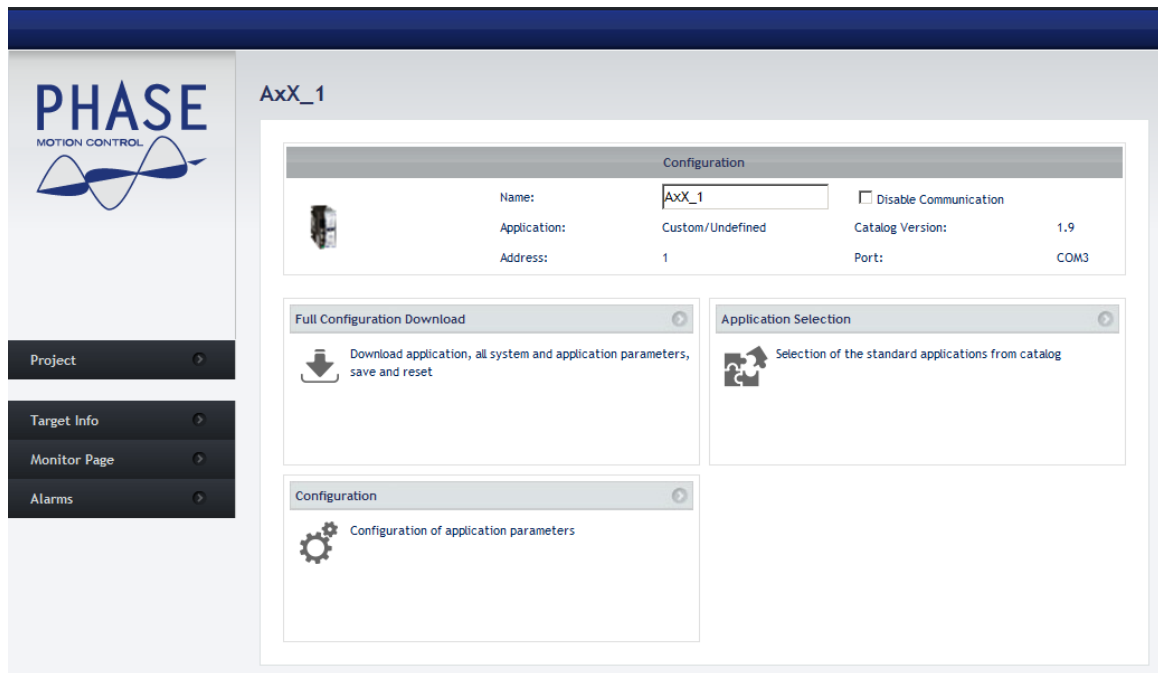
Figure 21 – AxX main page window

4.1   Device information

On the main page, the configuration table describes the basic device information.

"Name":   the device name with the default value "AxX_1".
If there are more than one device of AxX family in one project, the number will increase by one. It is also possible to modify the device name as you prefer. Insert a new name on the text box and press "Enter", and it will be updated immediately.

"Disable Communication" option: this option is not selected by default. If selected, the communication for the device is not able to connect.

"Application": the PLC application information on the current device.
If the device has one PLC application, which is compatible with application catalog, the information including application name and version is shown correctly. Instead, default setting "Custom/Undefined" is displayed.

"Catalog Version": the version number of AxX device.

"Address": the address value of communication. Refer to Chapter 6 for more details.
"Port": the COM port setting of communication. Refer to Chapter 6 for more details.

4.2   Full configuration download

Full configuration download in Figure 22 is to download the linked plc application into the device, and write all the configuration values of the system and application parameters into the device. After that, the drive will be automatically save and reset for storing the configuration.
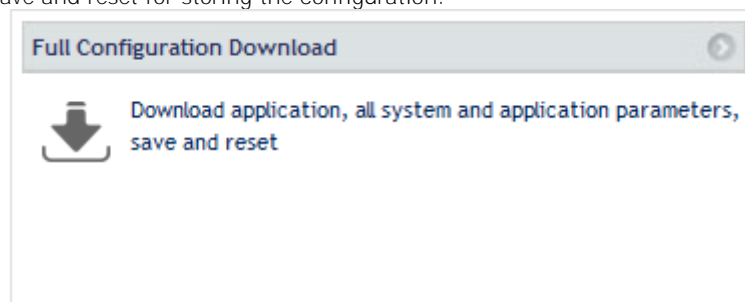
Figure 22 – Full configuration download

Note that it is the only function that can write correctly all the configuration parameters into the drive.

To make full configuration download, follow the procedure below:

- Click the "Full Configuration Download" bar on the main page, a messagebox is appeared to inform you that such procedure will overwrite any user application and parameters already in the drive with data in the actual parameter table.
- Press "Confirm" to continue. It is necessary to connect the target device before making such operation. Refer to Chapter 6 on how to communicate with one device.
- If the drive is not connected yet, a messagebox for online mode is required. Press "Confirm", the procedure will try to connect the device automatically. After connection is ok, the full configuration download starts to process with a processing bar.
- In the end the procedure executes successfully with a "Full configuration download terminated" message.

Note that, if any failure happens during the procedure, please back to the first step and start again.

4.3   Application selection

You can enter into application selection window in Figure 23 by clicking the "Application Selection" bar on the main page. It is listed all the available applications from catalog. Refer to AxX user manual for details on applications managements.
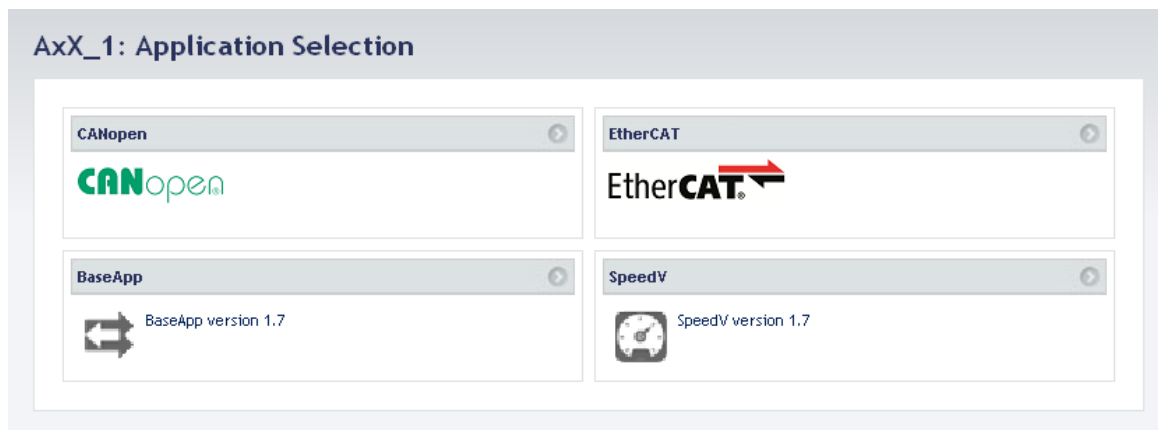
Figure 23 – Application selection window

To add one application into the device, just click the selected application bar.
Take AxX base plc application as example. If it is the device without any application embedded, click the "BaseApp" bar, it will automatically add the application into the device and back to the main page. In the meanwhile, the application information is updated on the configuration table.

If one application already exists in the device, after clicking the selected application bar, a dialogbox is appeared to warn you that actual application and related parameters will be deleted. Press "Confirm", the selected application will be added into device but the actual application and relative parameters will be deleted.

Note that offline mode is required when doing the operation.

Back to the main page by clicking the icon on the left top of Cockpit window.

4.4   Device configuration

You can enter into device configuration window in Figure 24 by clicking the "Configuration" bar on the main page.
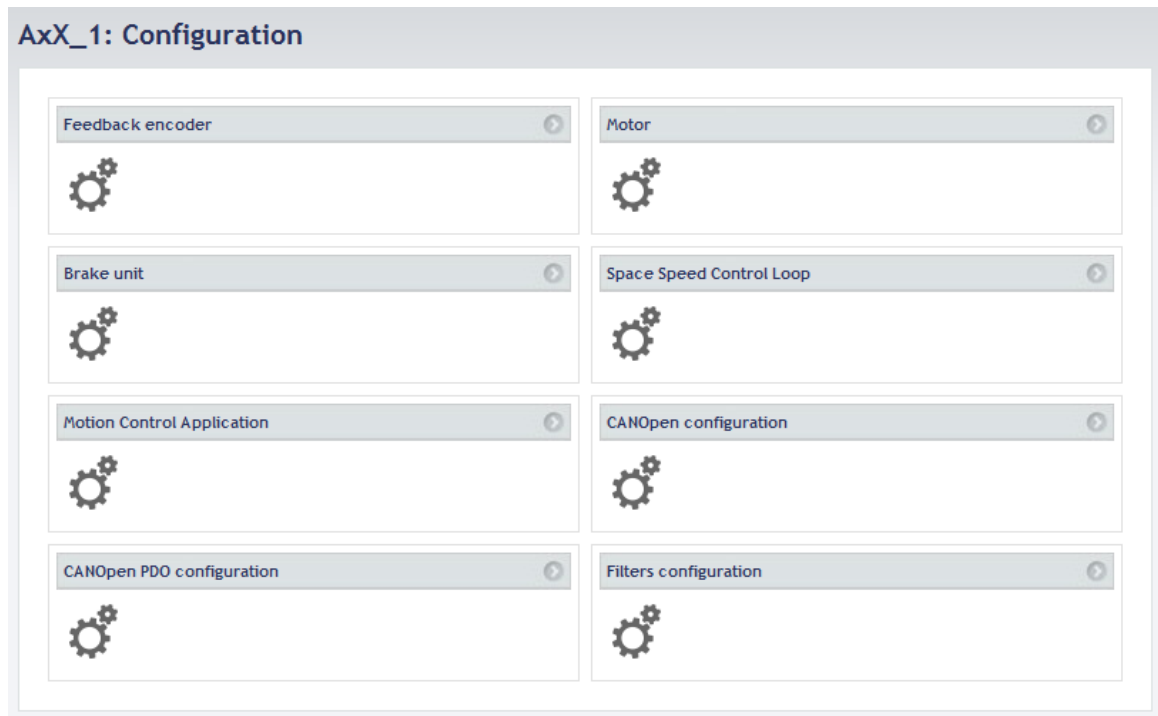
Figure 24 – Device configuration window

By default, it lists all the necessary system and application configuration links.
If the device has no application embedded or has one application without configuration pages, it will show only the system configuration pages.
If the device has one embedded application on which there are application configuration pages, it will show both system and application pages.
For advanced, the configuration pages could be also managed from LogicLab. Depending on different plc application, it is necessary to display only the configuration pages that user prefers. Refer to LogicLab manual for more details.

On the configuration window clicking the bar of the selected item, you can enter into parameters configuration in details. Please refer to AxX software manual for system parameters explanations in details in order to configure appropriate values.

On each parameters configuration page, you can back to the configuration window by clicking the "Done" bar

or the icon on the right top of the selected window.

Back to the main page by clicking the icon on the left top of Cockpit window.

4.5   Target information

You can enter into the target information window in Figure 25 by clicking the "Target Info" button on the left sidebar of the main page. The same command is available that is clicking "Target Info" on the menu "Service"

and it is also activated with the corresponding icon on the toolbar.

Figure 25 – Target information window

You can get the core information of AxX device on target information window.
The main components are System ID, System firmware, User Application, Controlboard ID, Powerboard ID, FPGA ID, Drive time. It is necessary to connect the device before viewing this window, please refer to Chapter 6 on how to connect one AxX device.

"System ID":   contains the device type, serial number and production date.
"System firmware": contains the firmware type and firmware version number.
"User Application": contains the PLC application name and version number.
"Controlboard ID": contain hardware controlboard type and version number.
"Powerboard ID": contain hardware power stage type and version number.
"FPGA ID":   contains hardware FPGA type and version number.
"Drive time":   contains the drive power-on time.

Press "refresh" button on the left sidebar of the target info window, you can get the latest data from a connected device.

Back to the main page by clicking the icon on the left top of Cockpit window.

4.6   Monitor page

You can enter into monitor page as Figure 26 by clicking the "Monitor Page" button on the left sidebar of the main page. You can also activate it with the corresponding icon on the toolbar.

Figure 26 – Monitor window

Monitor page provides you a friendly interface to verify the connections and settings between the motor and the drive.

Please setup drive with on-line mode before making operations on monitor page. Refer to Chapter 6 for communication settings. After the drive is connected, the monitor page refreshes every 400ms.

On "Power Stage" section, you can get the actual values of DC link voltage and heat sink temperature.
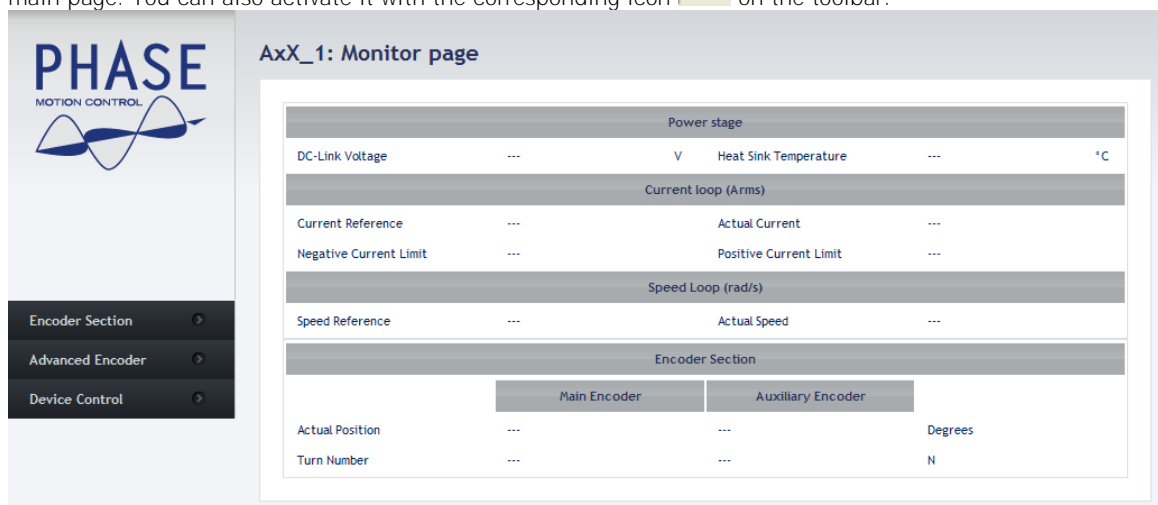On "Current Loop" section, it shows the current limits, the current reference value and the actual current when drive is in operation mode.
On "Speed Loop" section, it shows the speed reference and the actual speed of the connected device.

There are three buttons on the left sidebar of monitor page, "Encoder section", "Advanced encoder" and "Device control". By clicking those buttons, the relative section is shown in the window.

"Encoder section" in Figure 27 is shown by default. You can get the encoder reading values: position turn number and angle degree for both main encoder and auxiliary encoder.

Encoder Section			
	Main Encoder	**Auxiliary Encoder**	
Actual Position	---	---	Degrees
Turn Number	---	---	N

Figure 27 – Monitor window: Encoder section

On "Advanced encoder" section in Figure 28, it provides the encoder information with more details.

Advanced Encoder Readings			
	Main		**Auxiliary**
	Absolute	**Incremental**	
Virtual Position:	---	---	---
Revolutions:	---	---	---
Index Position:	---	---	---

Figure 28 – Monitor window: Advanced encoder

On "Device control" section in Figure 29, basic device control functions are applied, which are torque mode, profile velocity mode and profile position mode. You can control the drive with controlword and monitor the state of the drive by statusword. Refer to AxX software manual for further details on parameters explanations.

Status Word													
-	-	-	-	int limit active	-	warning	switch on dis	quick stop	voltage en	fault	operation en	switched on	rdy to switch on
⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙

Command Word								
halt	fault reset	-	-	-	en operation	quick stop	en voltage	switch on
⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙	⊙

Settings					
Target Position:	turns [ ]	angle [ ] °	Target Speed:	[ ] rad/s	
Mode of Operation:	[▼]		Current Mode:	---	

Figure 29 – Monitor window: Device control

Note that on this window auto refresh mode is enabled. In this case, the "Settings" values will be immediately written into the drive if you modify them.

Back to the main page by clicking the icon on the left top of Cockpit window.

4.7   Alarm information

The actual drive status is displayed both on the status bar and on the connection status window. When the drive is in alarm or warning condition, to view the alarms and warnings in details you can enter into alarm information window.

To enter into alarm information window in Figure 30, you can click the "Alarm" button on the left sidebar of the main page. You can also activate it with the corresponding icon on the toolbar. Moreover, the same command is also available by double clicking the device column on the connection status window.

Figure 30 – Alarm information window

All active alarms and warnings are listed on this window with refreshing time 500ms. Please refer to AxX reference manual for a detailed description of all Ax drive alarms and warnings. It is also necessary to connect the device before viewing the alarm information window. Refer to Chapter 6 for communication in details.

Back to the main page by clicking the icon on the left top of Cockpit window.

4.7.1   Alarm history information

To access the alarm history window in Figure 31, you can click the "Alarms history" button on the left sidebar of the alarm window.
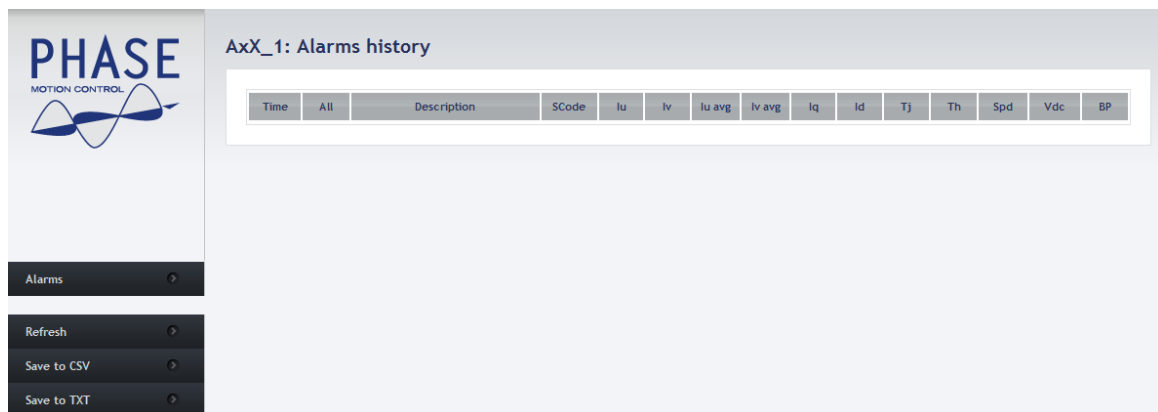
Figure 31 – Alarm history window

The latest 32 alarms occurred on the drive can be recorded on alarm history window and ordered by drive time decreasing. The latest alarm record is displayed at the first line of the table.
When an alarm is occurred, several variables and status values from the drive are recorded and reported on the alarm history window. Please refer to AxX reference manual for all Ax drive alarms in details.

Here are the explanation for each column on the alarms history table.

| "Time"	Power-on time of alarm.
"All"	All active alarm bits.
"Description"	Active alarm description.
"SCode"	Alarm subcode.
"Iu"	Instantaneous current phase U feedback.
"Iv"	Instantaneous current phase V feedback.
"Iu avg"	Average current phase U feedback.
"Iv avg"	Average current phase V feedback.
"Iq"	Instantaneous current quadrature feedback.
"Id"	Instantaneous current direct feedback.
"Tj"	Power module junction temperature.
"Th"	Heat sink temperature.
"Spd"	Average mechanical speed feedback.
"Vdc"	Instantaneous DC-bus voltage.
"BP"	Power dissipated by braking resistor.

You can save all the content on the alarm history window to a file with the extension name ".csv"/".txt" by clicking the "Save to CSV file"/"Save to TXT file" button on the left sidebar.

You can refresh the actual alarm history window by clicking the "Refresh" button on the left sidebar of alarm history window.

Note that, it is necessary to connect the device before viewing the alarm history window. Please refer to Chapter 6 for communication in details.

Back to the alarm window by clicking the "Alarms" button on the left sidebar of alarm history window.

Back to the main page by clicking the icon on the left top of Cockpit window.

4.8   Back to the project

The "Project" button on the left sidebar of the device main page is the entrance that allows you back to manage the whole project.

5.   PARAMETERS MANAGEMENT

All parameters sets for the system configuration and operation both on the AxX drive family and Tw family are resident in the memory. They are accessible from Cockpit configuration tool. The application parameters, which are located at defined memory addresses in the dedicated application, can be also managed by Cockpit.

Generally, parameters sets organized in hierarchical menus can be subdivided into three classes. which you can distinguish by colors on the parameter table.

- PAR parameters identified with Black color are read/written from Cockpit configurator and could be saved in the drive flash memory. In some cases Cockpit will inform some PAR system paramerers need a system reset after modifications.
- VAR parameters identified with Green color are read-only variables. They are used as diagnostic parameters in order to allow monitoring the drive variables.
- WKS parameters identified with Purple color are used to control drive behavior. They can be read/written from Cockpit configurator but restored to zero after drive reset.

Usually parameters sets are aligned with logic blocks and some principal sections collect more commonly used parameters. The default values set by the manufacturer are appropriately used in most cases. It could be necessary to tune them for particular applications in order to optimize the drive performances. Note that the user must be aware that the changes introduced assume a very deep knowledge of the drive firmware and function control blocks implementation. Please refer to reference manuals for system parameters descriptions in details.

This chapter focuses on how to manage (read/write) parameters with the device.

5.1   Node tree

There is the node tree window in Figure 32 on Cockpit working session, which is hidden by default. To active it, you can click the item "Tree" from the menu "View" or the corresponding icon [icon] on the toolbar.
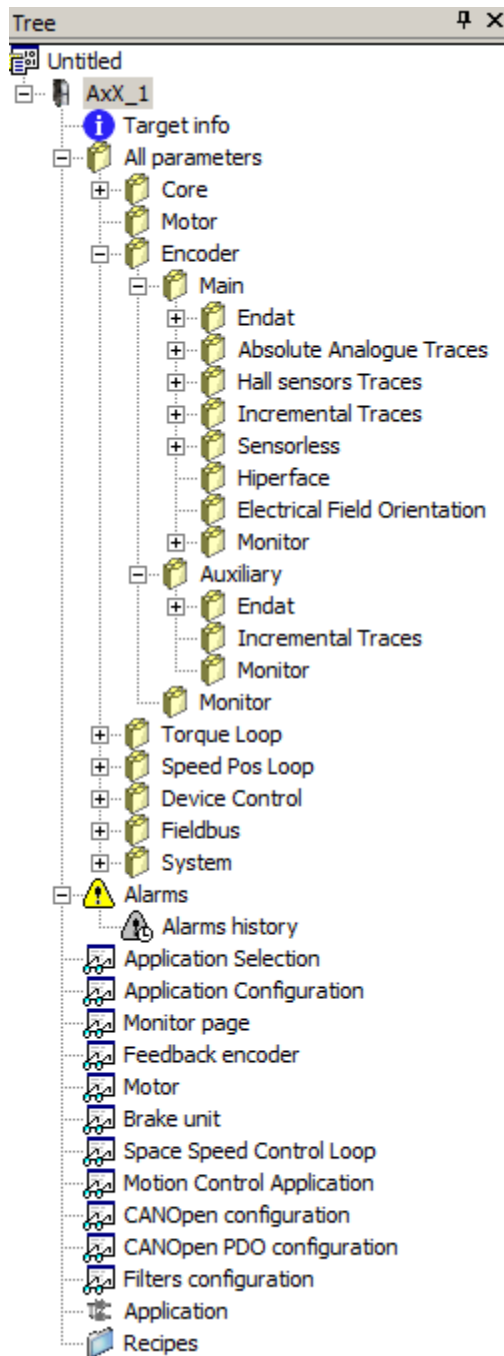

<div align="center">Figure 32 – Node tree</div>

Parameter files appear in the node tree on th left side of Cockpit window. It contains subfolders with parameter groups and different tools necessary to work with the selected device.

There are some advanced functions applied from the AxX node tree, refer to Chapter 9 in details.

To expend a folder to view its subfolder, click on the symbol [+] next to it.

To collapse a folder, click on the symbol [–] next to it.

5.2   Parameters table

You can use the parameters table to configure the device instead of the HTML pages. To open it, you can click the selected menu from the node tree.

Generally the parameters are structured in hierarchic subgroups collecting all parameters related to the different catalog devices. Take Motor parameters table as example, click the menu ![Motor] on the node tree, it is shown as Figure 33.

IPA	Name	Type	Value	Um	Default	Description
27803	parMotorData.Resistance	REAL	3.100	Ohm	3.100	Motor Stator Resistance
27804	parMotorData.Inductance	REAL	14.330	mH	14.330	Motor Stator Inductance
27805	parMotorData.KT	REAL	1.01	Nm/Arms	1.01	Motor KT
27806	parMotorData.CurrentNomin	REAL	3.82	Arms	3.82	Motor Current Nominal Zero Speed
27807	parMotorData.CurrentNomin	REAL	2.20	Arms	2.20	Motor Current Nominal
27808	parMotorData.CurrentPeak	REAL	7.00	Arms	7.00	Motor Peak Current
27809	parMotorData.SpeedNomina	REAL	523.00	rad/s	523.00	Motor Nominal Speed
27810	parMotorData.ThermalConst	UINT	2000	sec	2000	Motor ThermalConstant
27811	parMotorData.StatorInertia	REAL	0.10	mKg*m^2	0.10	Motor Stator Inertia
27814	parMotorData.PoleNumbers	UINT	8	--	8	Electrical poles count
27817	parMotorData.MaximumTem	REAL	0.00	°C	0.00	Motor Maximum Temp
27818	parMotorData.DirectInductan	REAL	0.000	mH	0.000	Motor Synchronous or Direct Inductance

Figure 33 – Parameters table

Each parameter record defines the following fileds:

"IPA"   indecates the parameter index as specified by Modbus protocol.
"Name"   is a mnemonic name used to identify the parameter.
"Type"   indicates the parameter data type.
"Value"   contains the actual value retained by Cockpit.
"Um"   refers the measure unit of the value.
"Default"   contains the default value retained by Cockpit.
"Description"   contains a brief description of the parameter.

For further information on all system parameters in details please refer to related reference manuals.

5.3   Read/Write one parameter

The displayed parameter value is related to the last read from the device or the modified value put in the field by user. When a value is different from the device value, because of editing or because it has not been read yet, it is shown in red color. It is shown in red until a Write of the parameter on the device is successfully done or a Read of the parameter is restored its value from the device. The value shown in the grid will be updated after a Read/Write operation. If the value in the grid is different from the default value, the default value is shown in blue instead of black.

Note that in next sections the drive is reqired in connected mode, refer to Chapter 6 for drive connection.

Assumed that drive is in connected mode, after selecting one parameter from the parameter table or the page, you have several ways to read and get the value stored in the drive:

- Click the item "Read selected" on the "Parameter" menu;

- Or click the corresponding icon ![R] on the toolbar;
- Or press the shortcut by CTRL+ "R".

Assumed that drive is in connected mode, after selecting one writable parameter from the parameter table or the page, click the Value field and put the modified value on it, it will be visualized in red to indicate that it is not written yet. Then you have several ways to write and send the value to the drive:

- Click the item "Write selected" on the "Parameter" menu;

- Or click the corresponding icon ![W] on the toolbar;
- Or press the shortcut by CTRL+ "W".

After Read/Write operation is done successfully, the parameter value is displayed in black.

Note that, on the parameters table, those above operations are also available when selecting more than one parameter by CTRL. Especially, you can use those above operations to read/write all parameters on the parameter grid with the "select all" command by clicking the "Select all" item on the "Parameter" menu, the corresponding icon ⊞ , or the shortcut CTRL + "A".

Note that, on the parameters grid, it is also possible to write the default values of the selected parameters on the current menu by clicking the "Write default values" item on the "Parameters" menu.

5.4   Read/Write parameters

Assumed that drive is in connected mode, on the parameters table or the page, you have several ways to read and get the values stored in the drive:

- Click the item "Page read" on the "Parameter" menu;

- Or click the corresponding icon ⊟ on the toolbar;
- Or press the shortcut by CTRL+Shift + "E";
- It is only available on the parameters table by clicking the item "Refresh page" on the "Parameter" menu or the corresponding icon ⟳ on the toolbar.

Assumed that drive is in connected mode, after parameters modifications are done, you have several ways to write and send the values to the drive:

- Click the item "Page write" on the "Parameter" menu;

- Or click the corresponding icon ⊟ on the toolbar;
- Or press the shortcut by CTRL+Shift + "G".

After Read/Write operation is done successfully, the parameter values are displayed in black.

5.5   Read/Write all parameters

Assumed that drive is in connected mode, it is advisable to fully read all drive parameters at the first startup to verify the consistence with the drive. you have several ways to read and get all the values stored in the drive:

- Click the item "Read all" on the "Parameter" menu;

- Or click the corresponding icon ⬆ on the toolbar;
- Or press the shortcut by CTRL+Shift + "R";

To write all the visuable parameters, the following ways can be done:

- Click the item "Write all" on the "Parameter" menu;
- Or press the shortcut by CTRL+Shift + "W";

Note that, such commands can not write all the device parameters. Full configuration downlaod is suggestable to make all the parameters written into the device.

5.6   Auto refresh mode

Auto refresh mode is usable to speed up reading and writing parameters. When the device is in auto refresh mode, the parameters values on the actaul parameters table or the page will update with the drive continuously. When the user changes the value in the editbox on the parameters table or the page, it will be wriiten to the drive instantly.
To active drive in auto refresh mode, the command is:

- Click the item "Auto refresh mode" on the "Parameters" menu;

- Or click the corresponding icon ⟳ on the toolbar;

5.7   Save parameters

When parameters have been optimized for the desired operation, it can be saved for subsequently usage. After making "Write" operation, the modified data is written directly to the drive flash memory, but not keeping permanently. It is possible to lose the data at system drive reset. Flash saving is necessary to store the parameters values permanently into the drive.

To save into drive flash memory, the command is:

- ▪ Click the item "Save parameters" on the "Parameters" menu;

- ▪ Or click the corresponding icon [icon] on the toolbar;
- ▪ Or press the shortcut by CTRL+Alt + "S".

Note that, some writable parameters are not active immediately after making "Write" operation, the parameters save and reset operation are required. After that, the modified parameters values will be fully active and ready to use. Refer to Chapter 6.4 "Drive reset" for reset operation in details.

6. COMMUNICATION

Cockpit provides the communication interface to establish a connection with the target device in order to monitor and debug the device. This chapter focuses on the communication operation required to connect to a target device.

6.1   Setup the communication

In order to establish the communication connection with the target device, it is therefore necessary the physical cables are ready and the network is properly configured, according to both AxX and Tw user manuals specifications.

To setup the communication, click "Communication settings" in the "Target" menu on the Cockpit main window. The devices link configuration window is appeared as Figure 34.
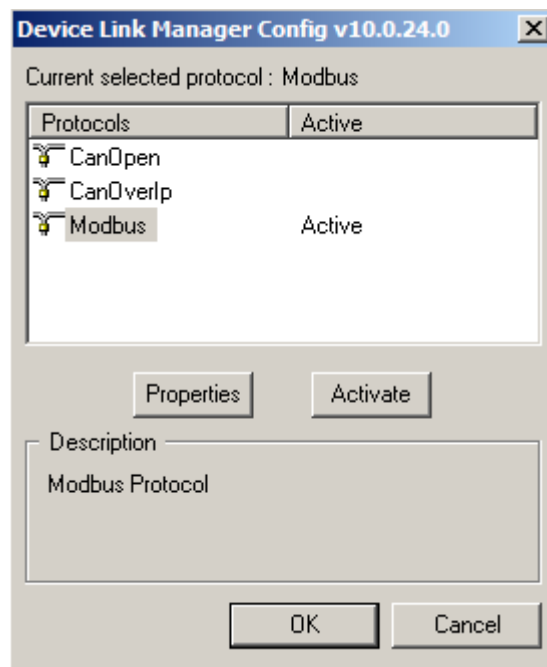

Figure 34 – Devices link configuration

You can select the appropriate protocol from the listed communication protocols. If a protocol you expect to appear is missing in the list, please check the setup executable program you have run on your PC.

6.1.1   Communication with AxX family

The communication with AxX family is accomplished by Modbus protocol with RS232/422 serial line. You can active the Modbus protocol and press the "Properties" key, the Modbus configuration window will show as Figure 35 and allow to setup the communication parameters for AxX family.
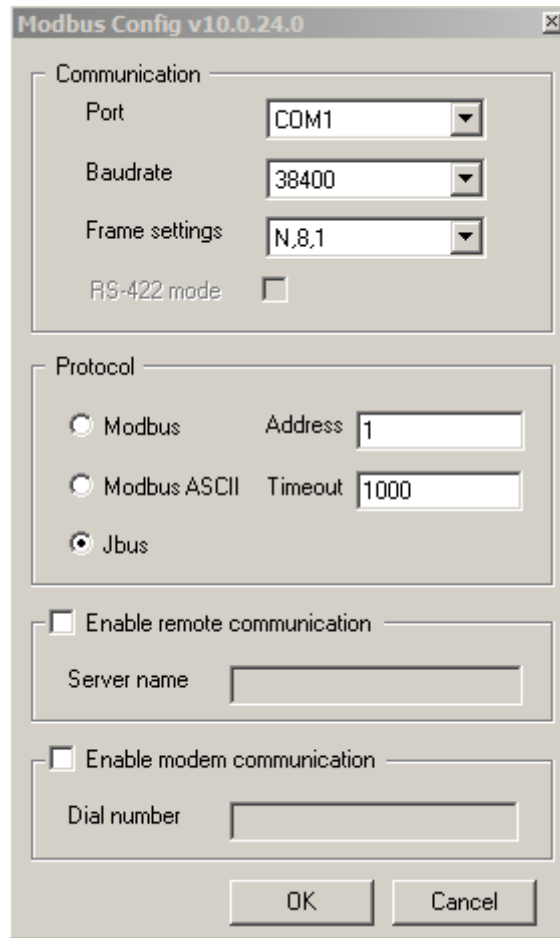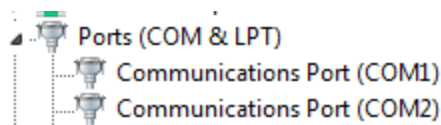

Figure 35 – Modbus configuration

Port: the port number on the PC connected with a serial cable to the RS232/422 drive port. The default value is COM1. You can check it from the PC device manager.

Baudrate: drive serial communication setting is 38400 by default.
Frame settings: serial port configuration setting is no parity, 8 data bits and 1 stop bit (N,8,1) by default.

The protocol default settings are: Jbus, address 1 and timeout 1000. Note that the setting address number must be identical to the address given to the drive.

Refer to AxX software manual for Modbus protocol and communication parameters specifications in details.

6.1.2   Communication with Tw family

The communication with Tw family is accomplished by CanOverIp protocol. You can active the CanOverIp protocol and press the "Properties" key, the CanOpenTcp configuration window will show as Figure 36 and allow to setup the communication parameters for Tw family.
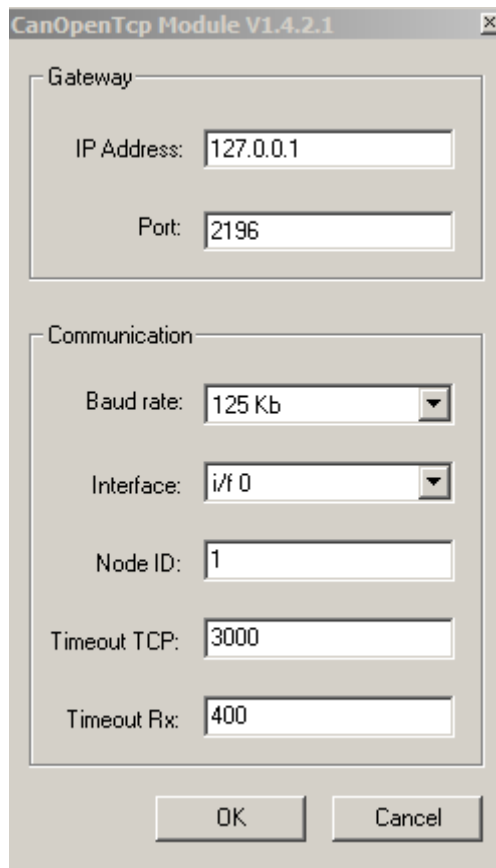
Figure 36 – CanOpenTcp configuration

The gateway default settings are IP address 127.0.0.1 and port 2196. It is suggested do not modify them.

The communication settings by default are baudrate 125Kb, Interface i/f 0, Node Id 1, TCP timeout 3000 and Rx timeout 400. It is necessary to choose and setup the baudrate and node id with appropriate values of the target device.

Refer to Tw software manual for CanOpen protocol and communication parameters specifications in details.

6.2   Enable the communication

After applying the changes you made to the communication settings by pressing "OK", now you can establish the connection. To enable the communication, the command is:

- Click the item "Connected mode" on the "Parameters" menu;
- Or click the corresponding icon on the toolbar;
- Or press the shortcut by CTRL+ "T".

You can disable and re-enable the communication using the same command. When the connection is active, the related menu "Connected mode" is selected and the corresponding icon appears as pressed.

Configurator allows the user to specify for each device its own communication settings to let the configurator communicate with each of the physical target devices of the project. It is possible to connect to physical targets once configured communication settings for each device in the project.

When the connection is active, the configurator starts communicating with targets using specified communication settings. Polling of a parameter is used to understand if the target is actually connected. If any of the target devices cannot be reached, the connection still remains active and continues polling devices but connection status indicated ERROR. When all the targets answers correctly to the polling requests connection status indicates CONNECTED.

The state of communication is shown in the right border of the status bar. If you have not yet attempted to connect to the target, the state of communication is set to ✖ DISCONNECTED .

When you try to connect to the target device, the state of communication becomes one of the following:

- ▪   🖳 CONNECTED : the communication has been established;
- ▪   ❌ ERROR :   the communication can not be established. You should check both the physical link and the communication settings.

6.3   Save the communication settings

When you connect to target devices using the communications settings, you may expect to save them for the future usage, especially the COM port setting which you usually use the same port for all devices on the same PC.

Saving the project file can obtain to save the communication settings in the meanwhile. Refer to Chapter 3.5 "Save the project" for details.

6.4   Drive reset

When establishing a connection with the target device, you can monitor and debug the status on the target, drive reset operation is to clear the actual status on the target and run at startup. When drive is in alarms/warnings status, it is necessary to make reset operation.

On Cockpit window, you have the following ways to reset drive:

- ▪   Click the item "Drive reset" on the "Target" menu;
- ▪   Or click the corresponding icon ⚡ on the toolbar;
- ▪   Or press the shortcut by CTRL + Alt + "R".

You can also refer to hardware manual for hardware reset operation.

7. CONTROL PANEL

The integrated control panel, which is one of the useful functions that Cockpit embodies for AxX family, allows to take direct "software" control of all I/Os of the device using the communication line. It allows monitoring and controlling the I/O interface of the device. When acting as a monitor, it is shown the actual status of the physical I/O; when controlling is enabled, the physical I/O are virtually disconnected and not used, the control panel will simulate the I/O interface to work with the drive.

This chapter focuses on the control panel operation to monitor and control the I/O interface of the device.

7.1   Control panel

One AxX device links to one relative control panel. To active it, first you have to focus on one target device of the project, then you have the following ways to active:

- ▪   Click the item "Control panel" on the "Target" menu;
- ▪   Or click the corresponding icon �🎚 on the toolbar.

On AxX family the control panel consists of eight digital inputs, eight digital outputs, four analog inputs and four analog outputs by software. Instead, there are only physically eight hardware digital inputs, four hardware digital outputs, four hardware single ended analog inputs and two hardware differential analog outputs.

Digital I/O is represented by means of buttons and leds, analog I/O by means of sliders and progress bars and the values are also available in the form of edit boxes shown as Figure 37 and Figure 38.
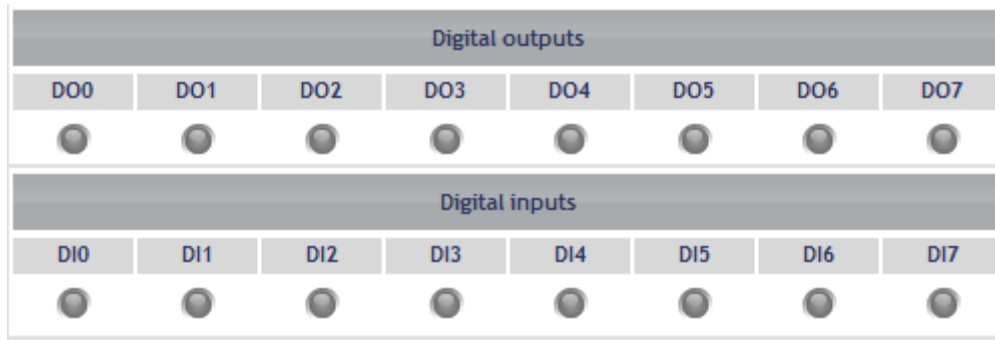
Digital Inputs/Outputs

Figure 37 – Digital Inputs/Outputs

- Grey icon ⚪ means digital input/output is OFF
- Red icon 🔴 means digital input/output is ON
- The name of digital input/output is configurable from LogicLab plc application, by default it is named as "DIx"/"DOx".
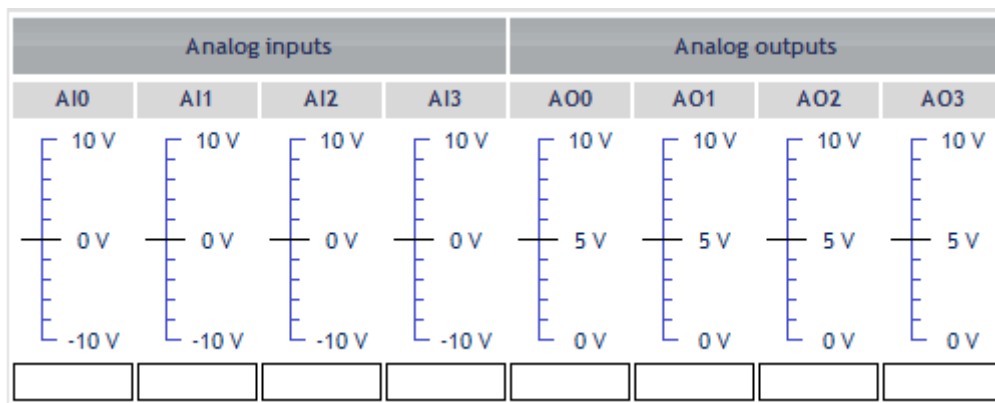
Analog Inputs/Outputs

Figure 38 – Analog Inputs/Outputs

- User can feed the value both on the textbox and on the scalebox for analog channels
- The name of analog input/output is configurable from LogicLab plc application, by default it is named as "AIx"/"AOx".

After activating the control panel, there are two switchable panels:

- Standard panel
- Enhanced panel

The following sections will introduce those two panels in details.

7.2   Standard panel

When the device is not connected, at the control panel startup it is shown as the standard panel as Figure 39. It is also shown when the connected device is detected that the device firmware is not supported with the enhanced panel.
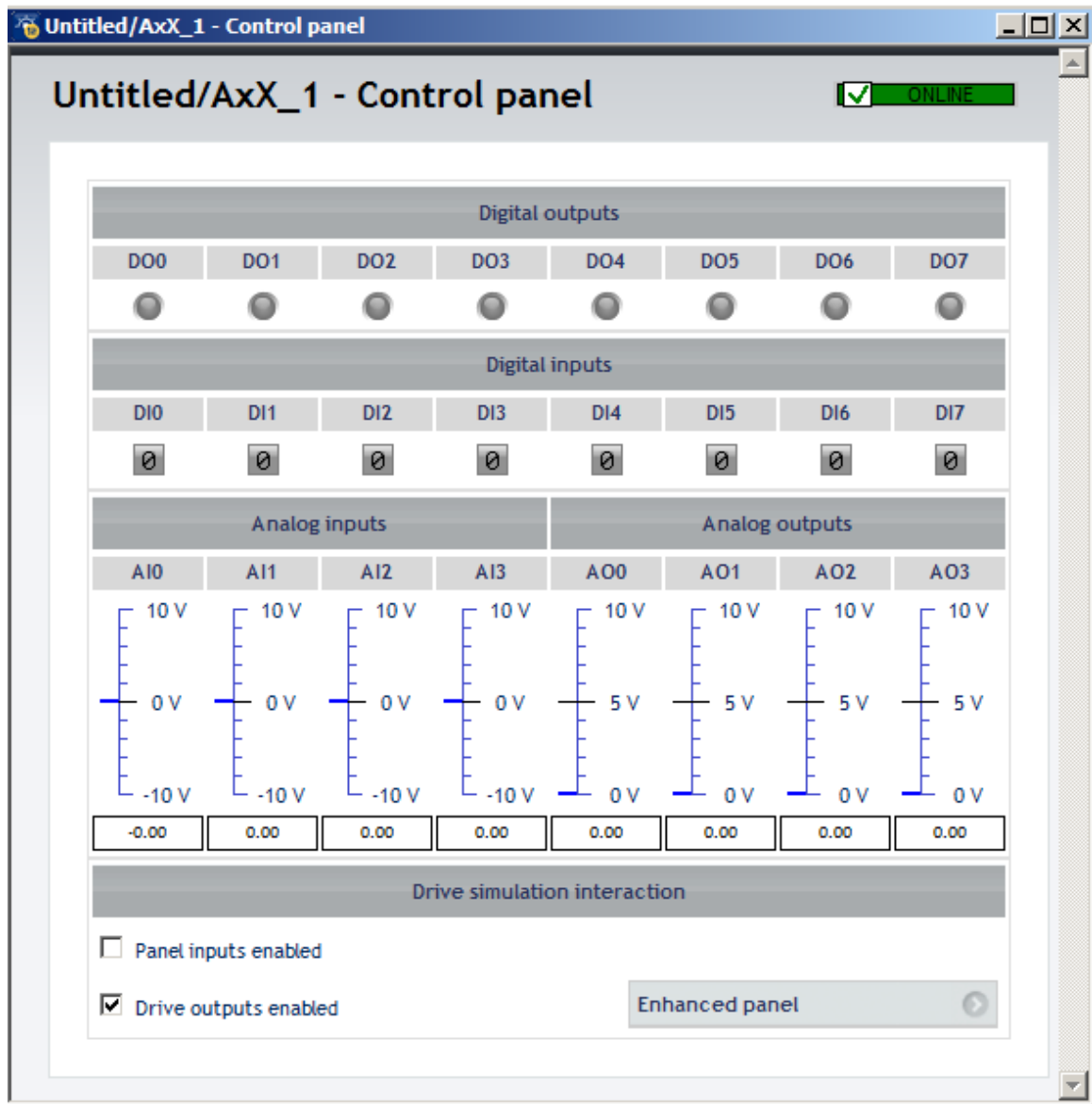
Figure 39 – Standard control panel

- When the first time activated, the "Panel Inputs Enabled" option is disabled and the control panel acts as a monitor of the actual status of physical I/Os
- When the "Panel Inputs Enabled" option is enabled, the drive physical digital and analog inputs are virtually disconnected.
- User may take the control of those inputs from control panel window only when the "Panel Inputs Enabled" option is enabled.
- The "Drive outputs enabled" option is activated by default. If it is disabled, the drive physical outputs cannot be monitored from control panel window.

7.3   Enhanced panel

It is shown as the enhanced panel in Figure 40 when the device is connected and it is detected that the firmware is also supported with the enhanced panel.
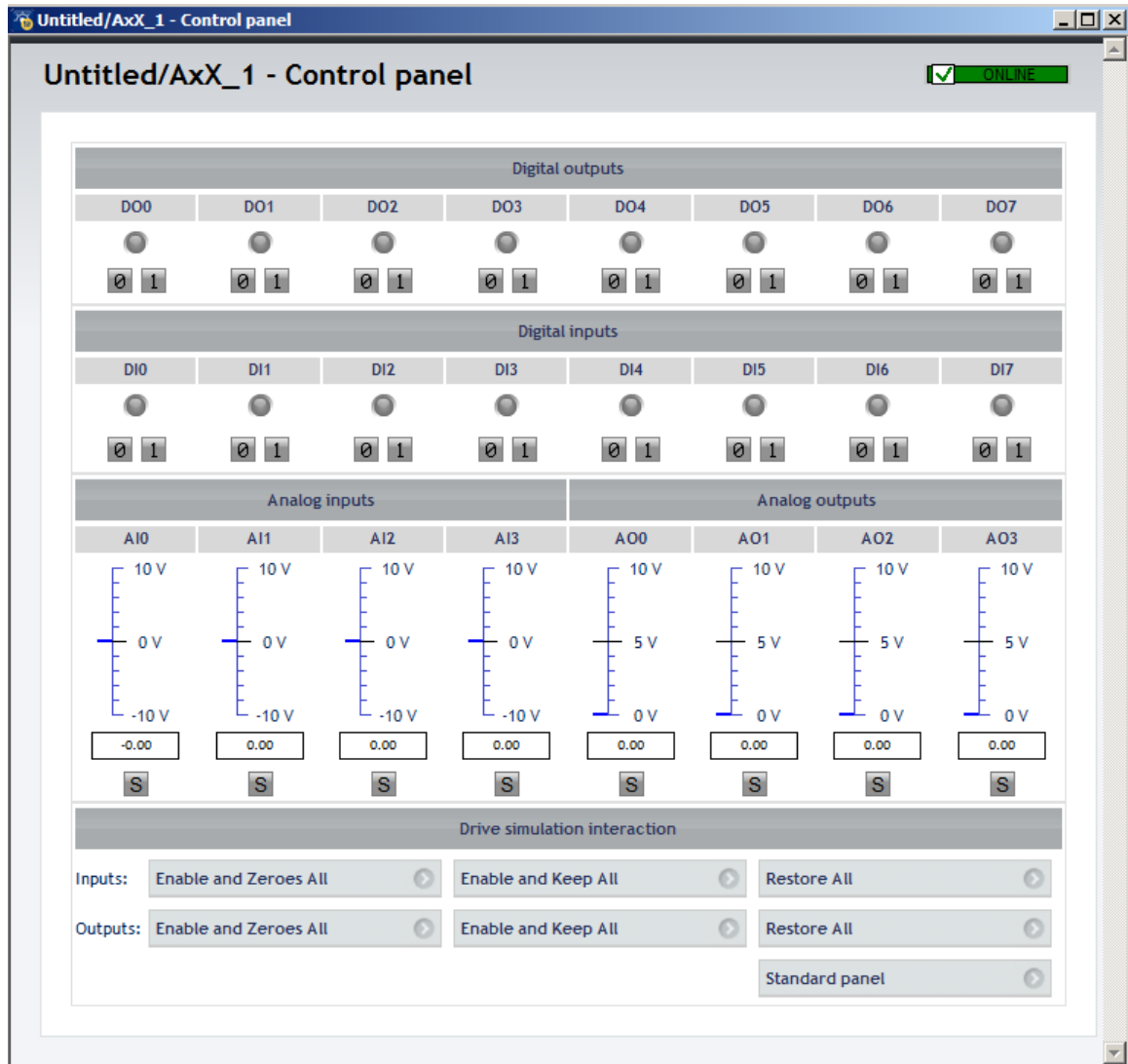
Figure 40 – Enhanced control panel

- Every digital input/output is independently controllable on the enhanced panel. By default, it is monitored the physical state on the "DIx"/"DOx"; Press icon [0] to icon [0], it is feed OFF state to the "DIx"/"DOx"; press icon [1] to icon [1], it is feed ON state to the "DIx"/"DOx".
- Every analog input/output is also independently controllable on the enhanced panel. By default, it is monitored the physical value on the "AIx"/"AOx"; press icon [S] to icon [S], both the textbox and the scalebox of the "AIx"/"AOx" is configurable, and the "AIx"/"AOx" is enabled and can be simulated from control panel window.
- "Enable and Zeroes all" function is to enable simulation on all Inputs/Outputs and zeroes all values.
- "Enable and keep all" function is to enable simulation on all Inputs/Outputs while keeping actual values.
- "Restore all" is to disable simulation on all Inputs/Outputs and restore IO port driven inputs/outputs.
- "Standard panel" is to switch back to the standard panel.

8. FIRMWARE UPGRADING

Firmware upgrading is released embodied in the Phase Tools setup at regular period on the Phase Motion Control web site, which could include new functions and generic enhancements. It could be done completely via Cockpit, that applies extension tools for both AxX family and Tw family to upgrade firmware independently.

To open the firmware upgrading window, you have the following ways:

- Click the item "Load firmware" on the "Service" menu;
- Or you can run as a standalone application by selecting the "Utilities" folder from the programs on Phase tools setup menu.
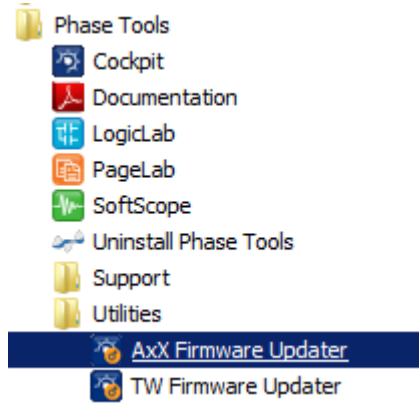
Note that the operation of firmware upgrading has to be performed by technical people trained in the Phase Motion Control.

8.1   AxX firmware updater

The firmware upgrading for AxX family is shown in Figure 41 on the following window.

Figure 41 – AxX firmware updater

- Communication status should be connected before starting the procedure. If it is not connected, press the icon [>] to enter into communication setting window, refer to Chapter 6 to connect the device in details.
- With the "Browse" button select the target firmware file to be downloaded, e.g.SysApp-1-8-312-xxxx-xx.sre, the file information will be shown on the "Download" frame.
- There is a list of available files, which support different kinds of devices depending on different hardware settings. On the "system firmware" frame and "FPGA ID" frame, it is shown the connected device information. The "xxxx-xxx" part of the file name is defined related to the "FPGA ID". You can select the correct .SRE file corresponding to the FPGA ID type and build.
- The baudrate could be selectable with different value from the baudrate combo box.
- Press "Load" button to start the upgrading process and show the processing on the "Operation" column on the "Download" frame.

8.2   Tw firmware updater

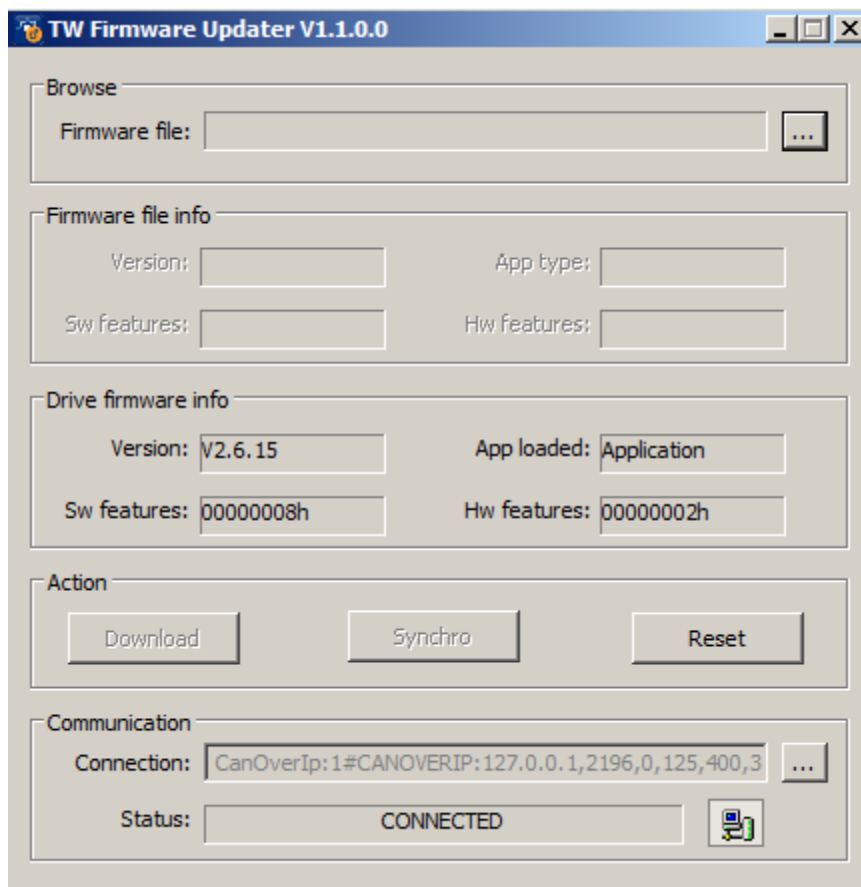The firmware upgrading for Tw family is shown in Figure 42 on the following window.

Figure 42 – Tw firmware updater

- Communication status should be connected before starting the procedure. If it is not connected, press the icon [...] to enter into communication setting window, refer to Chapter 7 to connect the device in details.
- With the "Browse" button select the target firmware file to be downloaded.
- The file information is shown on the "Firmware file info" frame. The connected device information is shown on the "Drive firmware info" frame.
- Press "Download" button to start the upgrading process.

Note that, if it is not possible to connect the device, synchronize it by pressing "Synchro" button, power cycling the device. When "synchronization executed" is appeared in the "operation" window, the device will be connected.

Note that, if any troubles occur during the firmware upgrading, it is necessary to start from the beginning the load of the selective firmware file. If the download is successfully completed, the device will reset and send a bootup message on the output window. In addition, you can reset the device manually by pressing the "Reset" button.

9. ADVANCED FUNCTIONS

Cockpit provides some useful functions to simplify and optimize users operations. The following advanced functions are introduced in details.

9.1   File comparison

When you may expect to compare the differences between two devices, click the item "File comparison" on the menu "Service", select the target project file, which contains the device you would like to compare with, then press "Open", the File comparison window in Figure 43 is popped up in front of the current working session.
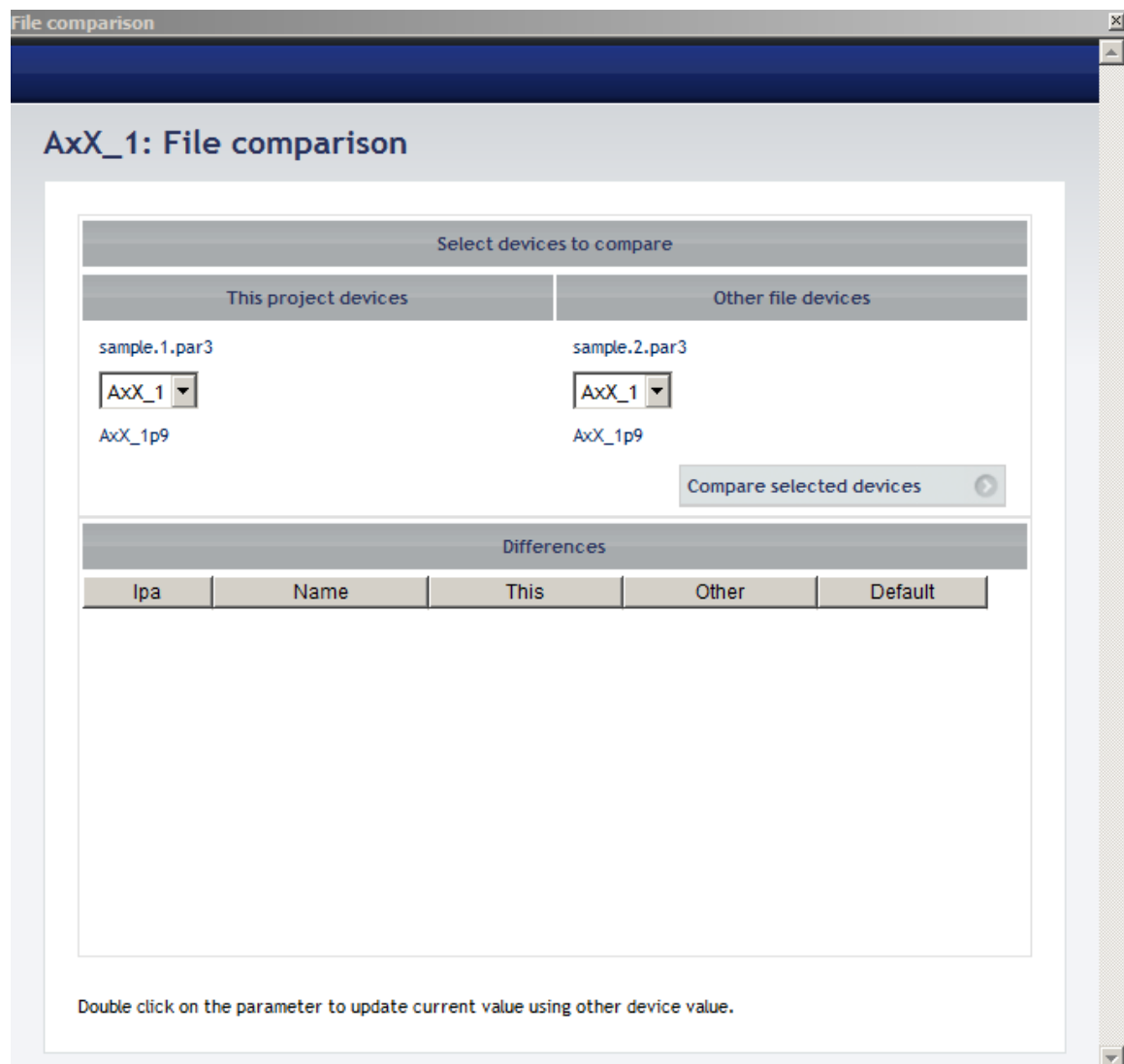

Figure 43 – File comparison

On the "Select devices to compare" table, the left part is displayed the current project information: the file name of the project; the device selection list, on which you may select the target device in the project to

compare; and the catalog version of the selected device. And the right part is displayed the compared project information.
Press the "Compare selected devices" bar, the differences of configuration parameters between two selected devices are listed in the "Differences" table below.

By double clicking the selected parameter in the "Differences" table, you may update the value in the current project device with the one in the compared project, and simultaneously the selected parameter will be removed from the table.

9.2   Export parameters to EDS/XML

The Tw drives are slave systems using a subset of the standard CANopen protocol to provide communication with a CANopen master system. As well as Tw drives, the Ax drives can also be working as CANopen slave systems. Further more, some types of Tw family and Ax family support EtherCAT protocol, refer to Hardware manual for different drives types in detail, and they can work as EtherCAT slave systems.

The usage of devices in communication networks requires the configuration information of the devices parameters and communciation fancilities. Both CANopen and EtherCAT define standardized methods to access the devices parameters via object dictionary. The standardized file format EDS is described for CANopen devices and the standardized file format XML is for EtherCAT devices.

Cockpit provides the entries for exporting EDS/XML files.
By clicking the item "Export parameters to EDS" on the menu "Service", a pop-up window is appeared to let you define a file name and choose a destination dictionary for the EDS file, then press "Confirm", an EDS file with standardized format including user specific application configuration information are exported.
The same way as above, by clicking the item "Export parameters to XML" on the menu "Service", you can export an XML file with standardized format, which includes user specific application configuration information.

9.3   Catalog upgrade

As Phase Tools including firmware releases are periodically upgrading on Phase Motion Control web site, the existing projects configured with previous catalog version are necessary to upgrade as well in order to append new functions and generic enhancements.

In the project both system device and application catalog vresions can be upgraded. Note that it operates on the Node tree. Refer to Chapter 5.1 for activating the Node tree. To upgrade system device catalog, you may right click the device root node in the Node tree. To upgrade application catalog, you may right click the application node in the Node tree. Selecting the "Change" item on the popup menu, The Device catalog window in Figure 44 is shown with the latest catalog version for the selected node in the Node tree.
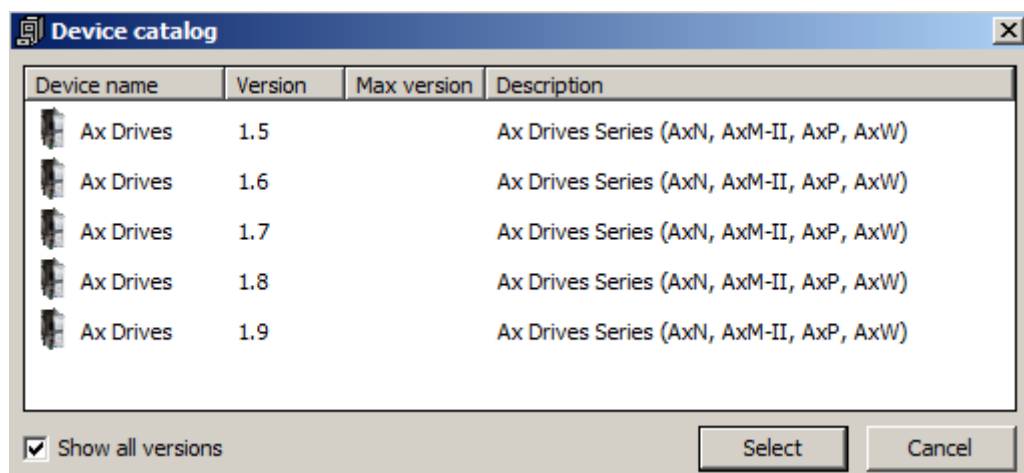

Figure 44 – Device catalog window

If enabled "Show all version" in the Device catalog window, all the catalog version for the selected node will be displayed.
Select the target catalog version by clicking the item in the Device catalog window, then press the "Select" button, the current catalog version will change to the selected one. In the end the message for the procedure result is reported in the Output window.
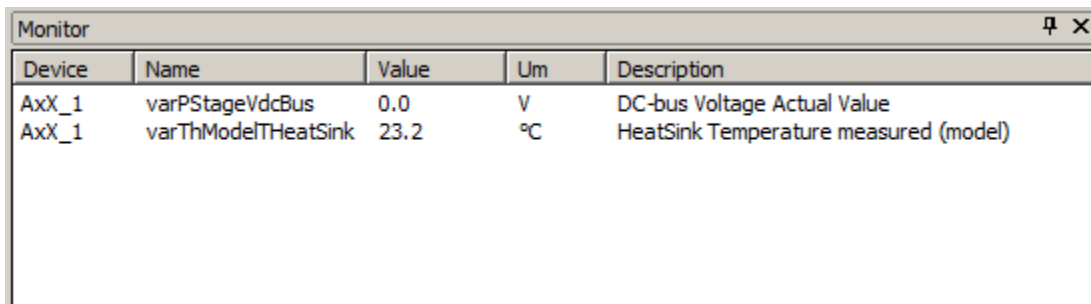
10. DEBUGGING TOOLS

Cockpit provides two debugging tools, which help users to check whether the device behaves as expected or not. Both these two debugging tools basically allow users to watch the value of selected variables while the devices were running. These two debugging tools are:

- Monitor window
- SoftScope

This chapter shows you how to debug devices using these tools.

10.1 Monitor window

The monitor window in Figure 43 allows you to watch the current values of a set of parameters chosen by user, which come from one project device and even from different project devices.

Figure 45 – Monitor window

Each parameter added in the monitor window contains the following fields:

Device   the device belonging to
Name   the name of the parameter
Value   the value of the parameter that is cyclically refreshed
Um   the measure unit of the parameter value
Description   the parameter description

Being an asynchronous tool, the monitor window does not guarantee synchronization of values. Therefore, when reading the values of the parameters in the monitor window, please note that they may refer to different execution cycles of the corresponding tasks.

10.1.1 Open/Close the monitor window

As introduced on Chapter 2.6, to open the monitor window, you can select the item "Monitor" from the menu "View", and also you can active the corresponding button on the toolbar.

To close the monitor window, you can deselect the item "Monitor" from the menu "View", or click on the corresponding button again. Alternatively, you can click the icon in the top right corner of the monitor window.

Please note that, in both cases, closing the monitor window means simply hiding it, not resetting it. Actually, if you close the monitor window and reopen it again, you will see that it still contains all the parameters you added to it.

10.1.2 Add/Remove parameters

In connection status, you can add parameters to the monitor window. Selecting the corresponding parameter from the parameter table, dragging it and dropping directly to the monitor window, the parameter is added and queued in the monitoring list. You can also select more than one parameters per time using the "Ctrl" key from the parameter table, then drag and drop them to the monitor window.

To remove one parameter from the Monitor window, you can select it in the Monitor window and press the "Delete" key.

10.1.3 Parameters refresh

The parameters in the monitor window are cyclically read from the device memory. The interval between two consecutive readings of the same parameter depends on the number of the parameters to be read by configurator in an entire cycle of readings.

However, this action is carried out asynchronously, that is, it may happen that a higher priority task modifies the values of some parameters while they are being read. Thus, at the end of a refreshment process, the values displayed in the monitor window may refer to different execution states of the project.

When the target device is disconnected, the Value field in the monitor window will get frozen with the last updated value until communication is restored.

10.2   SoftScope

SoftScope is a powerful debugging tool, which can be working either as the asynchronous debugger or as the synchronous debugger. Being an asynchronous tool, SoftScope allows you to plot the evolution of the values of a set of parameters. In addition, being a synchronous tool, it allows you to select a set of parameters, to have them sampled synchronously which occurs when the processor reaches the position where you place the trigger and to have their curve displayed in a proper window.

The following sections will introduce you how to operate the SoftScope in details.

10.2.1 Open SoftScope

When you are working on one selected device, you can easily open SoftScope from Cockpit window by clicking

the item "Open Softscope" on the "Target" menu or clicking the corresponding icon ▨ on the toolbar.

You can also run SoftScope as a standalone application by selecting the item "SoftScope" from the programs on Phase tools setup menu. From the Start Menu to run SoftScope, the initial interface is appeared in Figure 44.

Figure 46 – SoftScope initial interface

Click "**New acquisition …**", you can create a new SoftScope acquisition by selecting one target device from the pop-up catalog window.

Click "**Open acquisition …**", you can open an existing SoftScope acquisition by loading the correct directory.

Some SoftScope acquisitions you have used recently are listed in the "Most recent acquisitions". You can directly open them if they are not moved to another path manually.

10.2.2 SoftScope interface

Opening SoftScope, the interface is shown in Figure 45 for accessing the debugging. It consists of several parts as shown below.
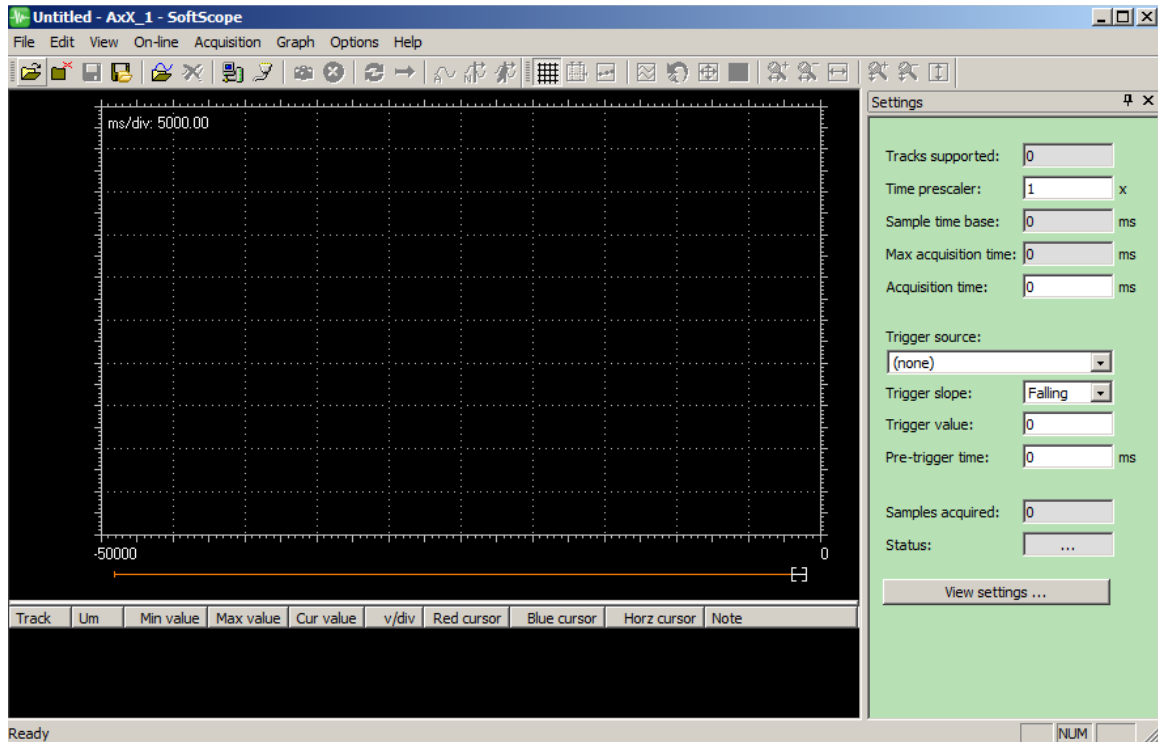
Figure 47 – SoftScope interface

The Menu bar collects all the operation commands for SoftScope, which can be used to control the acquisition process and the way data are displayed. See next sections for relative commands in details.

The Main toolbar   See Chapter 10.2.3 for details.
The Graph toolbar   See Chapter 10.2.4 for details.
The Chart area   See Chapter 10.2.5 for details.
The Tracks list window   See Chapter 10.2.6 for details.
The Settings bar   See Chapter 10.2.7 for details.

The Output bar prints the communication messages and acquisition status to give user more information. Active it from **the item** "Output bar" **in the menu** "View".

10.2.3 Main toolbar

The Main toolbar in Figure 46 contains the main controls of acquisition, which allow you to better supervise the working of this debugging tool, so as to get more information for the target device under scope. Those controls are also made accessible through the Menu bar.
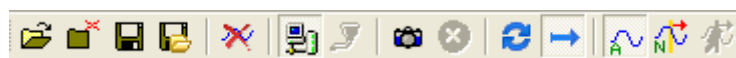
Figure 48 – SoftScope main toolbar

: the same commands are: the item "**Open …**" on the menu "File"; or the shortcut CTRL + "O". By this command you can open an existing acquisition file while the actual acquisition will be closed automatically. It is the same operation as "**Open acquisition …**" in Figure 44.

: the same commands are: the item "New" on the menu "File"; or the shortcut CTRL + "N". By this command you can open the initial interface as Figure 44 to create a new acquisition.

: the same commands are: the item "Save" on the menu "File"; or the shortcut CTRL + "S". Save any changes to the actual acquisition.

: the same command is the item "**Save as …**" on the menu "File". Save the actual acquisition to an alternative location instead of overwriting in the same directory.

: the same command is the item "Remove selected track" on the menu "Edit". Remove one selected parameter track from the Tracks list window.

: the same command is the item "Connect" on the menu "On-line". Connect/Disconnect with the device. SoftScope must be working in the connected mode. If it is disconnected, the control commands for data acquisition are disabled automatically.

: the same command is the item "**Set up communication …**" on the menu "On-line". It is enabled only when the communication is disconnected with the device. Refer to Chapter 6 for setting up communication in details.

: the same command is the item "Run acquisition" on the menu "Acquisition". Start to run acquisition. You can also start the acquisition by pressing the shortcut "F5".

: the same command is the item "Stop acquisition" on the menu "Acquisition". Stop running acquisition. It is enabled only when the acquisition is running. You can also stop the acquisition by pressing the shortcut Shift + "F5".

: the same command is the item "Continuous capture" on the menu "Acquisition". If the command is enabled, the acquisition will be continuously running for the next sampling after the previous acquisition is finished. The acquisition could stop until you press the command .

: the same command is the item "Single sequence capture" on the menu "Acquisition". If the command is enabled, the acquisition will stop when the actual acquisition is done. The Single sequence capture is enabled by default.

: the same command is the item "Auto trigger mode" on the menu "Acquisition". Enabling Auto trigger mode, the acquisition starts running immediately when you press the command regardless of the trigger condition. The Auto trigger mode is enabled by default.

: the same command is the item "Normal trigger mode" on the menu "Acquisition". Enabling Normal trigger mode, the acquisition starts running until the trigger condition becomes true after you press the command .

: the same command is the item "Force trigger" on the menu "Acquisition". Force the trigger condition to be true and start running acquisition. It is enabled only when the acquisition is in Pre-trigger mode, which is the trigger condition is still false but you have pressed the command .

10.2.4 Graph toolbar

The Graph toolbar in Figure 47 acts in a well-defined way on the behaviour of SoftScope interface. It deals with visualization controls of acquisition data and allows you to specify in detail when SoftScope is supposed to sample the parameters added to the Tracks list window. Those controls are also made accessible through the menu "Graph" on the Menu bar.

Figure 49 – SoftScope graph toolbar

: the same command is the item "Toggle grid" on the menu "Graph". The command is to show/hide the grid on the Chart area. With the grid it is more visualized to get the value of each sample.

: the same command is the item "Show measure bars" on the menu "Graph". The command is to show/hide simultaneously the two vertical cursors (the red cursor and the blue cursor) and the horizontal cursor (the green cursor) on the Chart area.

: the same command is the item "Show acquisitions" on the menu "Graph". The command is to show/hide the exact sample point in which the parameters are triggered at each acquisition.

: the same command is the item "Vertical split" on the menu "Graph". The command is to split the vertical axis into as many segments as the dragged-in parameters in order that the diagram of each parameter is drawn in a separate band.

: the same command is the item "Undo zoom settings" on the menu "Graph". The command is to return to the previous graph setting.

: the same command is the item "Show all values" on the menu "Graph". The command is used to fill all the values sampled in the current record set for all dragged-in parameters to the Chart area.

: the same command is the item "Show all values for selected track" on the menu "Graph". The command is used to fill all the values sampled in the current record set for the selected parameter to the Chart area.

: the same command is the item "Horizontal zoom in" on the menu "Graph". The operation enlarges the curves in horizontal axis of the Chart area so that more details may be viewed more clearly.

: the same command is the item "Horizontal zoom out" on the menu "Graph". The operation shrinks the curves in horizontal axis of the Chart area so that the entire curves may be viewed.

: the same command is the item "Horizontal show all" on the menu "Graph". The operation is used to horizontally display the record set samples. The first sample is placed on the left margin and the last sample is placed on the right margin of the Chart area.

: the same command is the item "Vertical zoom in" on the menu "Graph". It is the same function as the command on the vertical axis instead of horizontal axis. It enlarges the selected curve in vertical axis of the Chart area.

: the same command is the item "Vertical zoom out" on the menu "Graph". It is the same function as the command on the vertical axis instead of horizontal axis. It shrinks the selected curve in vertical axis of the Chart area.

⬍ :   the same command is the item "Vertical show all" on the menu "Graph". It is the same function as

the command ↔ on the vertical axis instead of horizontal axis. Max value sample is placed near the top margin and min value sample is placed near the bottom margin.

Note that all the commands in the Graph toolbar are disabled if no parameter has been added to the Tracks list window.

10.2.5 Chart area

The Chart area in Figure 48, the main part to display the acquisition, contains the following items to better analyse the acquisition.
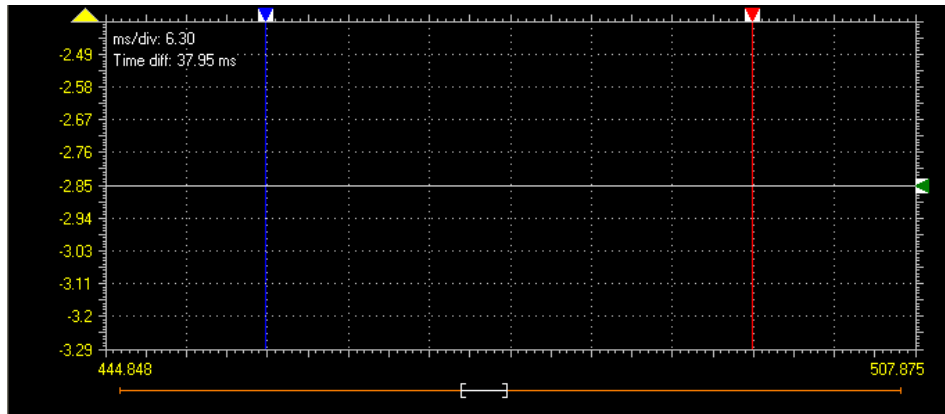
Figure 50 – SoftScope chart area

- Plot: the plot area containing the actual curves of the dragged-in parameters.
- Samples: acquisition samples in the current record set for the dragged-in parameters.
- Blue cursor: one of the two vertical measure bars, which can be exploited to take some measures on the Chart area.
- Red cursor: one of the two vertical measure bars. The same function as the Blue cursor.
- Horizontal cursor: the horizontal measure bar. The same function as the Blue cursor.
- Zero cursor: identifying the zero position of the selected track in vertical axis. It allows you to slide the curve up and down along the vertical axis by dragging it.
- Scroll bar: it appears on the bottom of the Chart area, allowing user to slide the curve back and forth along the horizontal axis to display the samples.

10.2.6 Tracks list window

The lower section of the SoftScope interface is the Tracks list window in Figure 49, which consists of a table for all dragged-in parameters.

Figure 51 – SoftScope tracks list window

Each dragged-in parameter has the following fields:

Track	the name of the parameter
Um	the measure unit of the parameter value
Min value	the minimum value in the record set
Max value	the maximum value in the record set
Cur value	the current value of the parameter
v/div	the scale value between two engineering units in vertical axis
Red cursor	the value of the parameter in the Chart area with the Red cursor
Blue cursor	the value of the parameter in the Chart area with the Blue cursor
Horz cursor	the value of the parameter in the Chart area with the horizontal cursor
Note	the parameter description

10.2.7 Settings bar

The Settings bar in Figure 50 allows user to setup acquisition sampling and visualization properties for parameters in the Tracks list window.
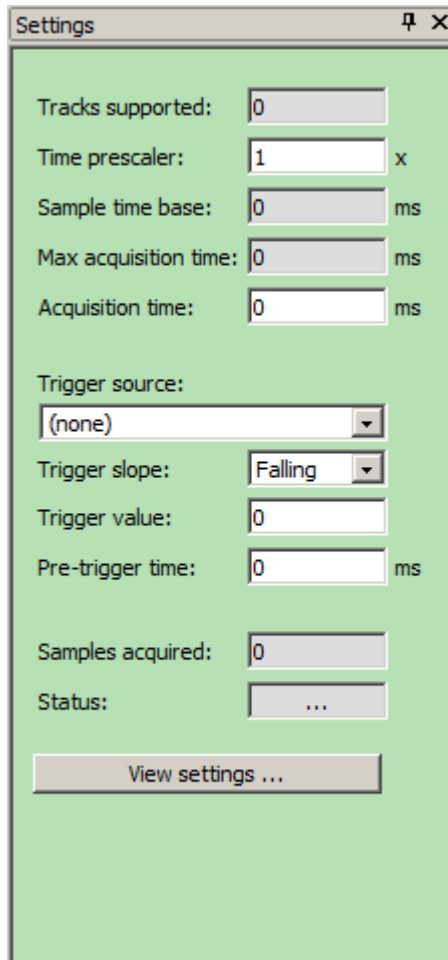

Figure 52 – SoftScope settings bar

Tracks supported shows the maximum number of parameters, which could be added into the Track list window depending on different target devices.

Time prescaler is an edit box to amplify or reduce the base sample time with an integral number.

Sample time base displays the base sample time depending on the value of Time prescaler. Once the base sample time is modified, the maximum acquisition time is changed dependently.

Max acquisition time shows the maximum time during acquisition.

Acquisition time is an edit box to apply the required acquisition time. It must be less than or equal to the Max acquisition time.

Those are the settings for trigger condition:
Trigger source   a combo box to choose one parameter as a trigger.
Trigger slope   a combo box to select a Rising/Falling slope.
Trigger value   to apply the value of the parameter when triggering.
Pre-trigger time   to apply the acquisition start time before triggered.

Note that, the trigger source can be only selected from the parameters dragged in the Tracks list window and the trigger value has the same measure unit as in the Tracks list window.

Samples acquired shows the number of samples in the current record set.

Status is an image box to show the status of acquisition processing. The acquisition status can assume one of the following values:

Idle :	No acquisition is running and it is ready to start acquisition.
Pre-trig :	In normal trigger mode, no samples are taken as the trigger condition is false.
Triggered :	In normal trigger mode, the trigger condition becomes true and it starts acquisition.
Acquiring :	The samples are acquiring.
Stopped :	The current acquisition process is stopped.

Note that, if the communication is disconnected, three dots are filled in the Status box instead of above values.

Generally, SoftScope applies default scales to the axes; however, you can customize the appearance of the acquisition data with different scales by clicking the button "View **settings ...**" on the Settings bar. It is appeared in Figure 51.
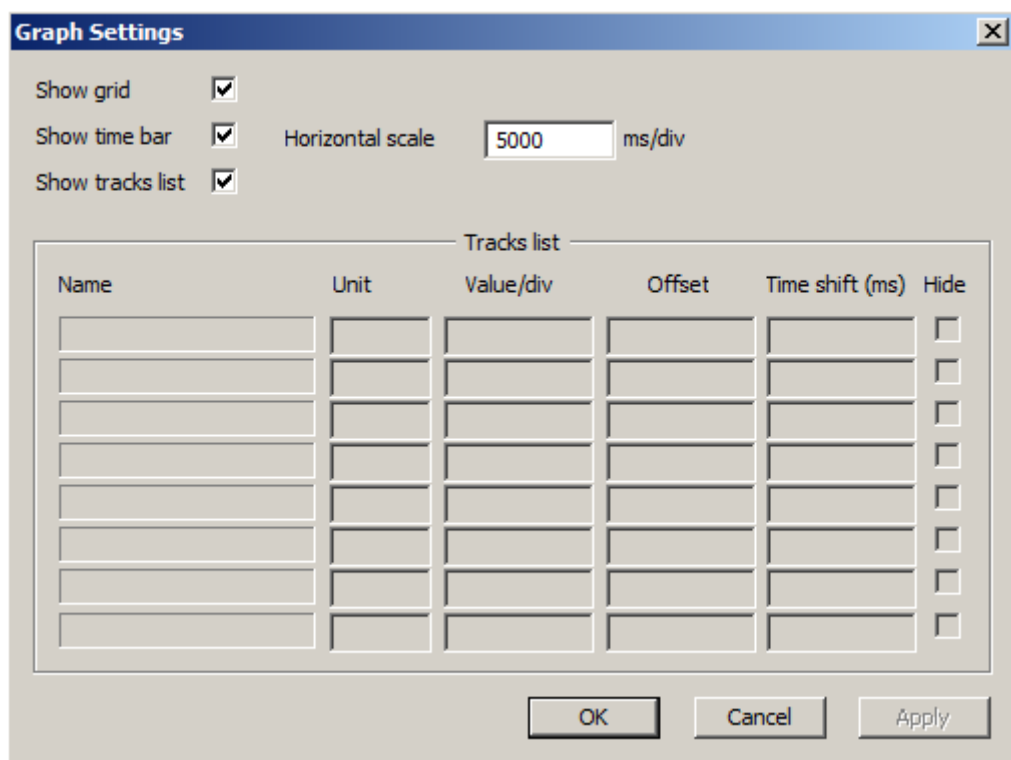

Figure 53 – SoftScope graph settigns window

Show grid is a selective option to show/hide the grid on the Chart area. The same as the command on the Graph toolbar.

Show time bar is a selective option to show/hide the time scroll bar at the bottom of the Chart area.

Show tracks list is a selective option to show/hide the Tracks list window below the Chart area.

Horizontal scale is to set how many milliseconds per unit of horizontal axis. The unit is the space between two vertical lines of the background grid.

Tracks list table allows you to adjust some graphic properties of the plot for each parameter. There are the following fields:

Unit   the measure unit of the parameter value
Value/div the value per unit of vertical axis for the selected parameter.
Offset   the difference between the value crossing the horizontal axis and zero for the selected parameter.
Hide   the flag to show/hide the selected parameter on the Chart area

Note that, you have to press the "Apply" button to make you changes effective; or press the "OK" button to apply your changes and close the Graph Settings window.

10.2.8 Add/Remove parameters

To add parameters to SoftScope, the same operation as on the Monitor window is that, selecting the corresponding parameter from the parameter table, dragging it and dropping to the Tracks list window or the Chart area on the SoftScope interface.

On the Tracks list window the parameter is added and queued, in the meanwhile, on the Chart area a new track identifying the parameter with the same color in the Tracks list window is also added. Once the first parameter is dropped, the control commands on the Main toolbar and the Graph toolbar are automatically enabled to allow user setup sampling and visualizing.

You can do the same procedure to apply all the parameters you want to investigate. You can also select more than one parameters per time using the "Ctrl" key from the parameter table, then drag and drop them to SoftScope.

Different parameters in the Tracks list window and Chart area are distinguished by different colors. There is the maximum number of tracks supported by different product families. Especially, the max number for Ax series is 8; and the max number for Tw series is 4. The read-only box "Tracks supported" on the "Settings bar" is displayed the correct value depending on different selected device.

Note that, if the communication is disabled, you cannot add any parameter to SoftScope. The icons and on the Main toolbar are communication relative commands. Refer to Chapter 6 for communication setup in details.

Note that, you can add to Softscope all the parameters in the parameters table, but also all the variables and parameters defined in the LogicLab. Please refer to LogicLab manual on how to operate in details.

To remove one parameter from SoftScope, you can select it by clicking in the Tracks list window, then press the "Delete" key, or click the item "Remove selected track" on the "Edit" menu. You can also remove it by clicking the corresponding icon on the Main toolbar.

10.2.9 Data acquisition

SoftScope applies two acquisition modes generally:

- Auto trigger mode
- Normal trigger mode

Note that the communication must be enabled in order to make acquisition either on Auto trigger mode or on Normal trigger mode. Refer to Chapter 10.2.3 for communication commands.

Auto trigger mode is an acquisition mode without any trigger source. After dragging and dropping parameters into SoftScope, the Auto trigger mode starts acquiring data immediately when you press the "Run acquisition" button . In the meanwhile the status of acquisition in the Settings bar is changed to Acquiring . Figure 52 is an example for acquisition in Auto trigger mode.
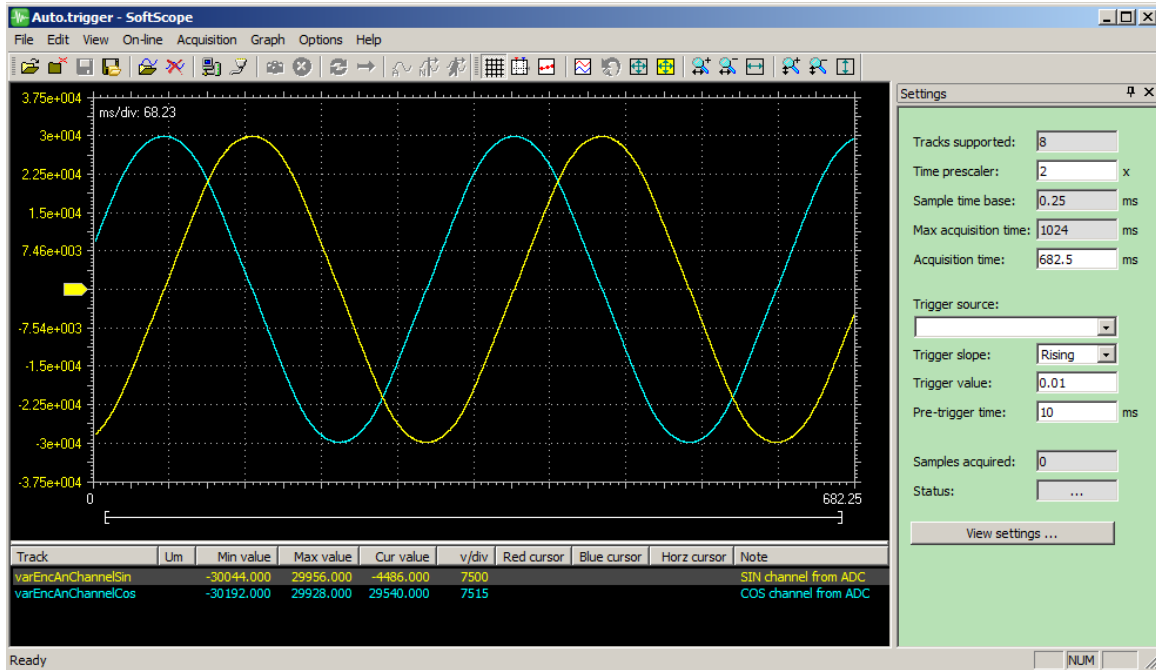
Figure 54 – Auto trigger mode

Normal trigger mode is used to make acquisition with one trigger source. After dragging and dropping parameters into SoftScope, you have to setup one trigger source condition on the Settings bar refer to Chapter 10.2.7, and then press the "Run acquisition" button to start the Normal trigger mode. If the trigger condition is currently false, the data acquisition does not start even though you press the "Run acquisition" button. At this moment the acquisition status in the Settings bar is Pre-trig . Once the trigger condition becomes true and the acquisition status is changed to Triggered , data acquisition starts and now the status is Acquiring . Figure 53 is an example for acquisition in Normal trigger mode.


Figure 55 – Normal trigger mode

Especially in Normal trigger mode, when the trigger condition is false, you can press the button to force the trigger and start acquisition immediately.

Both in Auto trigger mode and in Normal trigger mode, during acquiring, you can press the button whenever you prefer to stop acquisition.

When acquisition is completed, SoftScope plots the curves of dragged-in parameters to the axes as default scales in order that all data fit in the Chart area. To better analyse the acquisition data, you may expect to adjust the scales and viewings.

You may zoom in and out on horizontal axis or vertical axis by corresponding buttons , , and . Refer to Chapter 10.2.4.

You may quickly adapt the scales of horizontal axis, vertical axis or both to display all the samples by corresponding buttons , , and .

You may expect to split the respective tracks by the button when you have sampled two or more parameters.

You may expect to highlight the single values detected during acquisition by the button .

You may expect to show the measure bars by the button .
If you want to measure a time interval between two events, you just have to move one bar to the point that corresponds to the first event and the other to the point that corresponding to the second one. The time interval between the two bars is shown in the top left corner of the Chart area.
You can also use the measure bar to read the value of all the parameters in the Chart area at a particular moment. You just have to move the bar to the point, which corresponds to the instant you want to observe in the Chart area. Then in the Tracks list window you can read the values of all the parameters at that particular moment. Refer to Chapter 10.2.6.

You may expect to apply a different scale of the horizontal axis, which is common to all the tracks, or specify a distinct scale for vertical axis of each dragged-in parameter by opening the Graph settings shown in Figure 51. Refer to Chapter 10.2.7.

Especially you may expect to load other tracks from other existing acquisition files to the current in order to better analyse and investigate data. To do this, you can click the item "**Load offline tracks …**" on the menu "File" to load a destination directory, the tracks will be added in the current acquisition file. Please note that the number of tracks loaded is limited by the max number of supported in the current device.

10.2.10 Save and Close SoftScope

At the end of acquisition, you can save the acquisition samples to a file in order to be further analysed in the future.

To save the acquisition, you can select the corresponding icon on the Main toolbar. Refer to Chapter 10.2.3 for "Save" commands.

You may also save the acquisition to an alternative location by clicking the corresponding icon on the Main toolbar. Refer to Chapter 10.2.3 for "**Save as …**" commands.

After saving the acquisition, you can terminate SoftScope explicitly closing the current working session or exiting it. To do that, you can select the item "Exit" from the "File" menu; or click the icon on the right top of SoftScope.