

PLC-ANALYZER pro 6

PLC-Logic analysis in no time

User manual



The logic analyzer
for the PLC

PLC-ANALYZER pro 6 - User Manual

Copyright © 1993 - 2021 AUTEM GmbH. All rights reserved. No part of this user manual, including excerpts, may be reproduced, photocopied or electronically stored without the expressive written permission of AUTEM.

The software described in this manual is subject of a software license agreement and may only be used according to the terms of this agreement.

AUTEM GmbH
Dithmarscher Straße 29
26723 Emden
Germany

Telephone	+49 (0)4921 / 9610-0
Telefax	+49 (0)4921 / 9610-96
E-Mail	info@autem.de
Web	www.autem.de

AUTEM does not give any warranty for this manual as well as no express or tacit warranties on commercial quality and suitability for a particular use. AUTEM does not take over adhesion for errors contained in it or for damages that may occur as a result of using or applying this material.

The soft and hardware designations mentioned in this book are in most cases also registered trademarks and are subject to the legal regulations as such.

For references, suggestions and improvement suggestions we are always grateful. Please send these to AUTEM.

2nd Edition 2021

Table of Contents

Introduction	4
Overview	4
Fields of application	5
First steps	5
Installation	6
Hard- and software requirements	6
Installation	6
Uninstallation	7
Starting the program	8
Configuration	9
Signals	9
Projects and Signal files	10
Language	10
General settings	11
Working with PLC-ANALYZER pro 6	12
Create new project	12
Open project	12
Edit project	13
Signal source	13
Add signal source	13
Edit signal source	14
Pseudosignal	17
Trigger	19
Create new trigger	19
Edit Trigger	19
Trigger-Action	25
Create new Trigger-Action	25
Edit Trigger-Action	25
Send E-Mail	26
Send SMS	27
Show Message	28
Play acoustic signal	29
Activate / deactivate addresses	30
Activate / deactivate trigger	30
Execute command	31
Signal window	32
Create new signal window	32
Edit y(t) signal window	33
Edit text signal window	35
Edit HMI signal window	36
Edit video signal window	38
Destination	40
Create new destination	40
Edit Destination	41
Signal file	41
CSV-Text file	43
Video file	44
Insert variable	45
Start of recording	46
Project-Info	47
Signal display and analysis	48
Open signal file	48
Save signal file	48
Adjusting the signal display	49
Signal selection	49
Signal scaling	49
Change symbols and comments of signals	49
Moving signale	50
Insert note	50
Change signal color	50
Mark Section	50
Time information	51
Signal cursor	51
Time measurement	52

Search funktions	53
Search signal value	53
Search trigger	55
Search note	56
Search time	57
Compare signal files	58
Merge signal files	59
Event bar	60
Signal statistics	60
Print a signal file	62
Export a signal file	63
Export as Text (CSV)	63
Export as PDF	64
Export as Graphic	65
Export as HTML page	65
Import von Signaldaten	67
Signal file info	68
Signal source	69
Siemens SIMATIC S7	69
Installation	69
Installing additional hardware	69
Installing additional software	69
Configuration	70
Data acquisition	72
Supported PLC models and CPUs	72
Recordable PLC addresses	72
Number of recordable addresses	72
Time behaviour and particularities	73
Cycle-precise recording	74
Configuration of PLC-driver for cycle-precise acquisition	74
Input of addresses	74
Start acquisition	75
Particularities in signal display and analysis	76
Menu commands	77
File	77
Signal file	79
Search	80
View	80
Extras	81
Window	82
Help	82
Toolbar symbols	83
Help	84
Frequently Asked Questions - FAQ	84
Technical Support	86

Introduction

Welcome to the new PLC-ANALYZER pro 6, the software logical analyzer, that gives you valuable help in solving automation problems.

The SPS-ANALYZER pro 6 is very easy and intuitive to operate. The software supports you with notes and an extensive online help (F1).

Nevertheless, please read the manual to familiarize yourself with the various application possibilities of the SPS-ANALYZER pro 6.

This manual informs you about the installation, operation and application possibilities of the program.

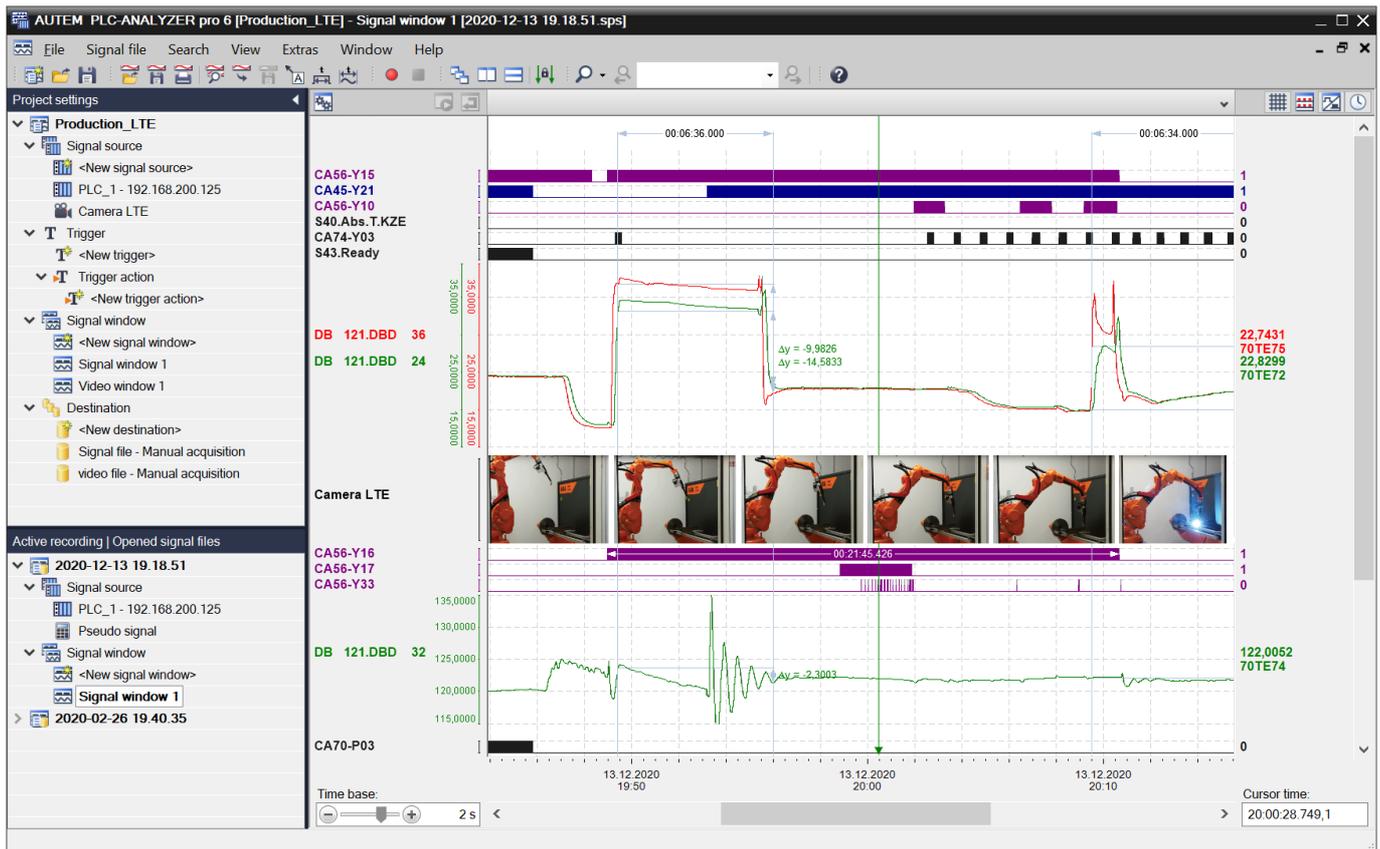


Fig. 1-1 PLC-ANALYZER pro 6

Overview

The program PLC-ANALYZER pro 6 is a software tool for logic analysis and registration of measured variables in PLC-controlled systems. It enables in a comfortable way the acquisition, representation and evaluation of PLC signals of all kinds. These include inputs, outputs, flags, timers, counters, data words and data blocks.

Equipped with the capabilities of a conventional logic analyzer, the PLC-ANALYZER pro 6 usually requires no software or hardware modifications to the PLC to acquire the measurement data. This is due to the use of the existing PLC protocol for data acquisition, whereby the typical couplings of programming devices - LAN connections or automation networks - are used by the PLC-ANALYZER pro 6. A programming device or PC connected to the PLC can thus be used directly for data acquisition without further modification.

With the PLC-ANALYZER pro 6, the user is also able to record internal PLC signals - e.g. flags, timers or variables from data blocks - which otherwise cannot be acquired from outside.

The special **AD_USB-Box**[®] measuring adapter can also be used to record external voltages and currents that are not directly available in the PLC.

The **video track module** also enables video recording synchronously with the PLC signal acquisition.

In this manual, the Ethernet TCP/IP - PROFINET driver for the Siemens SIMATIC S7 device family was chosen as

an example to illustrate the use of the program. The essential contents also apply to all other available PLC drivers, e.g. the PROFIBUS driver for SIMATIC S7 or the CoDeSys driver. Specific notes on the special features of the PLC drivers can be found in the corresponding driver addendum. It is essential that you read the respective driver addendum before using a PLC driver. Please also pay attention to the clearly marked warnings, which point out possible dangers when using the PLC-ANALYZER pro 6.



WARNING

Wherever faults occurring in the automation equipment cause major material damage or even personal injury, i.e. can be dangerous faults, additional external precautions must be taken or equipment must be created which guarantees or forces a safe operating state even in the event of a fault (e.g. independent limit switches, mechanical interlocks, etc.).

Fields of application

- Installation and maintenance
- Fault diagnosis in PLC systems
- Finding and isolating sporadic errors
- Analysis and optimization / cycle time reduction
- Predictive maintenance
- Condition monitoring
- Short- and long-term registration of measured variables
- Documentation and support for your QA system, TPM/OEE
- Development, training

First steps

A typical control expiration of the PLC-ANALYZER pro 6 looks as follows:

- Connect PC with the PLC programming unit interface
- Select signals to record
- Select signals to be displayed on the screen
- Start data recording
- End data recording
- Evaluate and analyse recorded signals in a graphical display (also possible during the running recording)
- Measurement of the recorded signals to determine time intervals

Installation

Hard- and software requirements

To operate PLC-ANALYZER pro the following system configuration is necessary:

- PC with 1.2 GHz
- Microsoft® Windows 7/8/10 (32/64 bit)
- 2 GB RAM
- 2 GB of available hard-disk space

Depending on the PLC driver, additional requirements may have to be met by the acquisition computer, e.g. the presence of a LAN adapter or additional device driver. Please read the chapter *Installation* in the corresponding driver addendum.

Installation

Install the PLC-ANALYZER pro 6 on your PC:

Download your personal installation file from <https://www.autem.de/get-plcapro6>. To do this, you will need the “download-activation code” printed on your Software License Certificate (“SLC”).

After the installation program starts, several dialog windows appear in which you are asked to make settings. Simply follow the instructions of the installation program.

After successful installation, the licenses have to be activated. You now need your licenses, which are usually contained on the supplied USB-LicenseKey Dongle. Plug the USB-LicenseKey Dongle into a free USB port on your computer and click the button *Activate licenses now*. The AUTEM LicenseManager will start immediately.



NOTE

It is possible to activate the licenses after the installation. Therefore, start the AUTEM LicenseManager in the program group PLC-ANALYZER pro 6.

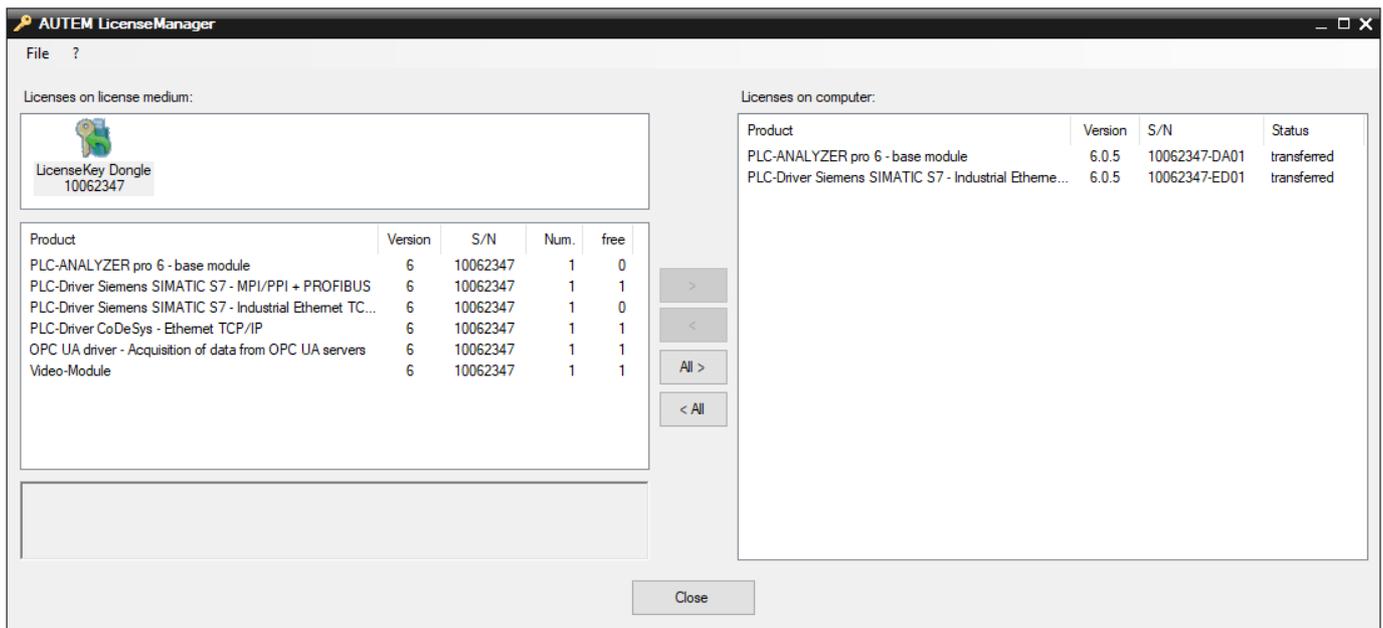


Fig. 2-1 AUTEM LicenseManager

The AUTEM LicenseManager now shows you on the left side all licenses available on the USB-LicenseKey Dongle. The column *free* shows the number of licenses available for transfer.

Select those licenses from the list *Licenses on this License Medium* you want to install on your PC. Press and hold the button <Ctrl>, if you want to select several licenses. Then press the button > to transfer the selected licenses to your PC.

The button *All>* transfers all licenses to your PC once.



Warning

Formatting or re-installation of your hard disk causes a loss of activated license(s). Insure that all activated licenses are re-transferred to License-Key Dongle before deleting your hard disk.

Uninstallation

If you no longer need a license on your computer, transfer it back to the USB-LicenseKey dongle.

Therefore insert the USB-LicenseKey Dongle into a free USB port of your PC and start the AUTEM LicenseManager of PLC-ANALYZER pro 6. Select those licenses from the list *Licenses on your PC* you want to uninstall. Press and hold the button <Ctrl>, if you want to select several licenses. Press button < to uninstall the licenses from your PC.

The button *All<* uninstalls all licenses at once.

In both cases the licenses will be transferred back to the USB-LicenseKey Dongle.



NOTE

The licenses can only be transferred back to that USB-LicenseKey Dongle, from which they were installed originally. The LicenseKey Dongle and the license(s) to be transferred back must have the identical serial number.

Starting the program



NOTE

Please make sure that time and date in your PC are set correctly. Correct the values if necessary.

Start PLC-ANALYZER pro simply by a double-click on the *PLC-ANALYZER pro 6* icon in the program group PLC-ANALYZER pro 6.

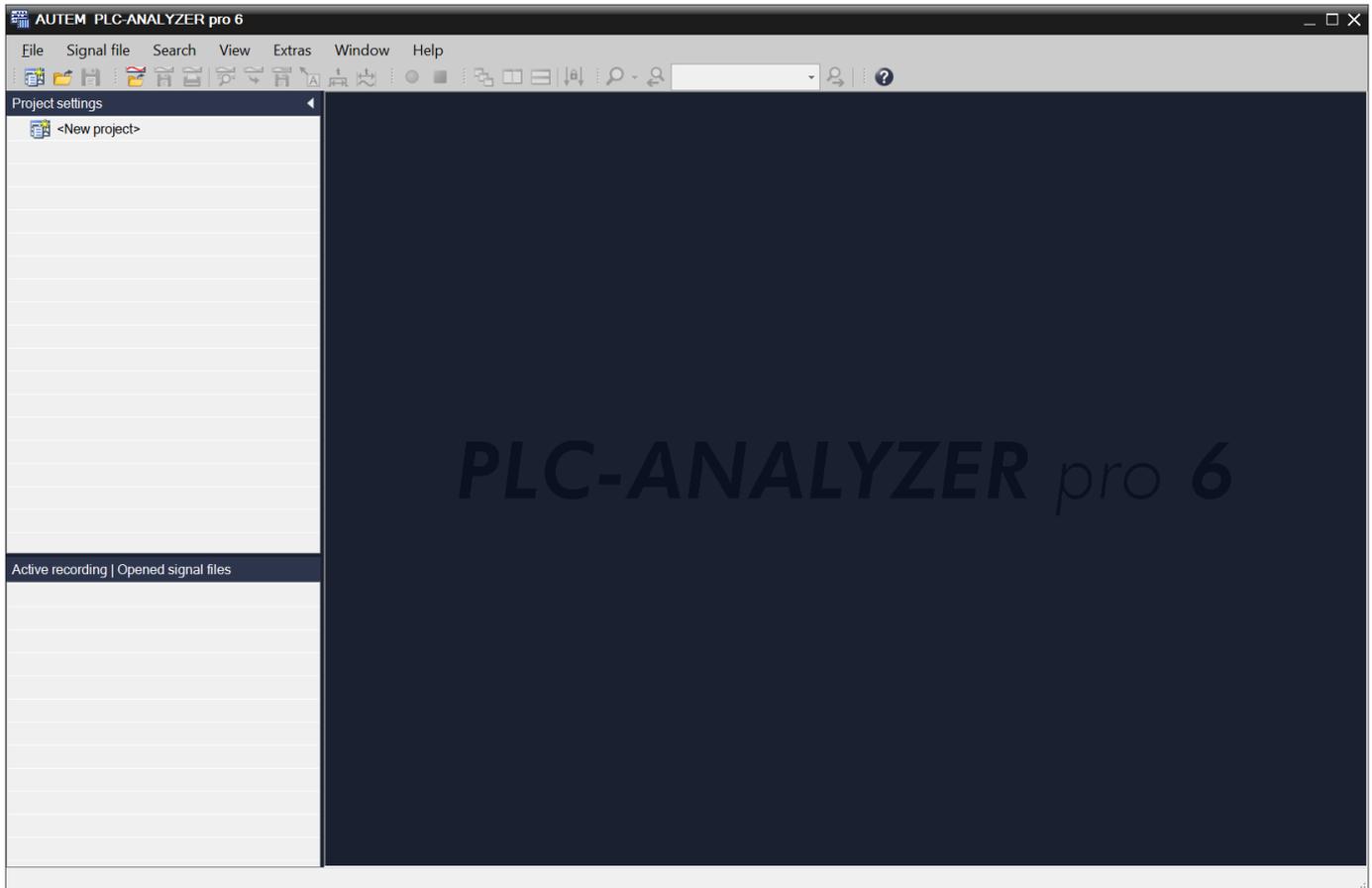


Fig. 2-2 The initial screen

After successful activation of the licenses, the above screen should appear (*fig. 2-2*). Otherwise, a message window will appear asking you to activate the corresponding licenses.

After you have successfully completed the installation of the PLC-ANALYZER pro 6, you should make some important basic settings in the program. You will find a detailed description on the settings in the [next chapter](#).

Configuration

Before you can start the signal data acquisition, you must configure the PLC ANALYZER pro 6. The general settings are made in the *Extras* menu item. Configure the program according to your requirements under *Settings*. During the installation, the program has already been preconfigured according to your specifications.



Fig. 3-1 Option menu

The settings are saved when exiting the PLC-ANALYZER pro 6. The next time the program is started, the settings are used again.

Signals

Under *Signals* you set the signal display of signal information.

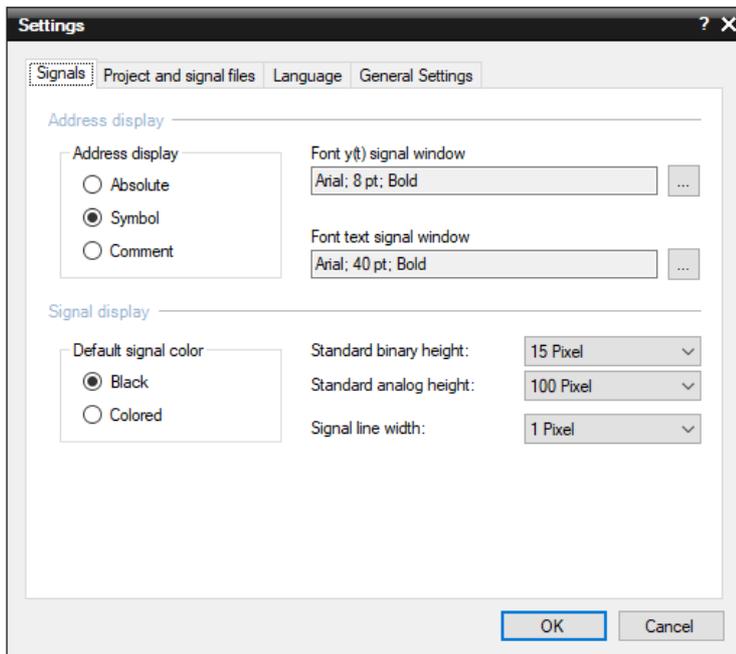


Fig. 3-2 Setting signal options

Under *Address display*, select the display for the address indication. Besides displaying as absolute address (e.g. M 34.5) it is possible to display also as symbolic identifier or as. The desired *font* can also be set.

Under *Standard Signal Colour* the signal display can be set. By choosing the option *colored* each address type (input, output, flag, etc.) will be displayed in a different colour. By choosing the option *black* all signals to be displayed are shown in black. The signal colour can be changed at any time.

The *Standard height of binary signals*, the *Standard height of analog signals* as well as the *Signal line width* can also be set.

Projects and Signal files

Set the project and signal file options here.

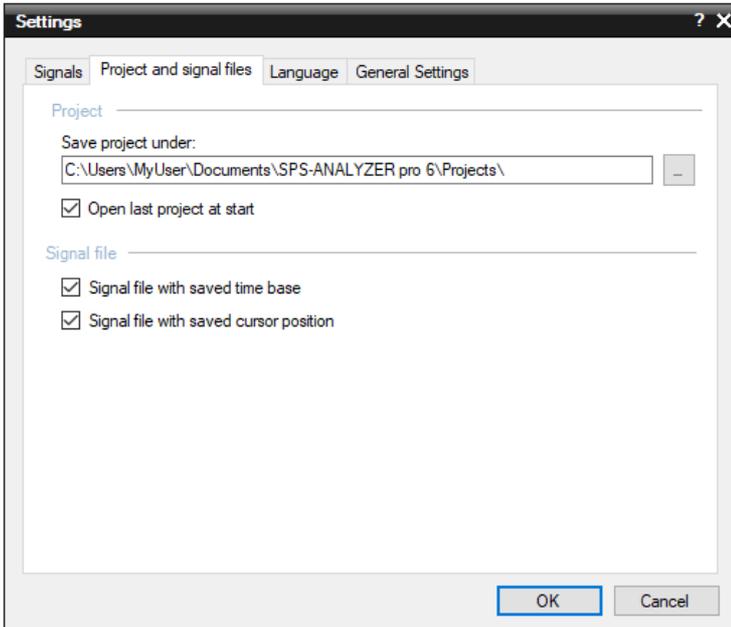


Fig. 3-3 Choose project- and signal file options

Under *Project*, set the default directory path for the projects. The PLC-ANALYZER pro 6 works with this directory unless you explicitly specify otherwise. Click on the button to the right of the text field to change the directory path. Also specify whether the last loaded project should be opened at start-up.

Under *Signal file*, select whether the last saved *cursor* position and *time* base should be set automatically when opening a signal file.

Language

In this window you can set the dialog language of the PLC-ANALYZER pro 6. Then press the *OK* button. The language is switched during runtime of the PLC ANALYZER pro.

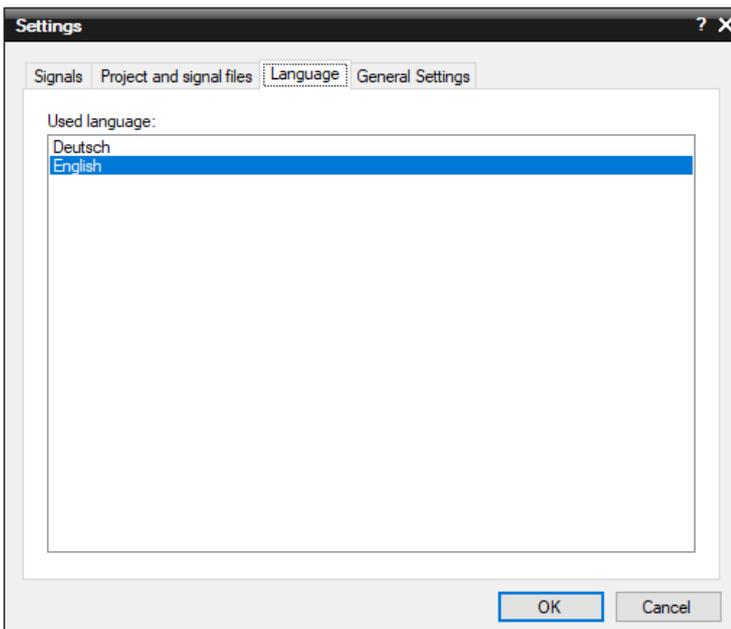


Fig. 3-4 Choose language

General settings

Set the general settings of the PLC-ANALYZER pro here.

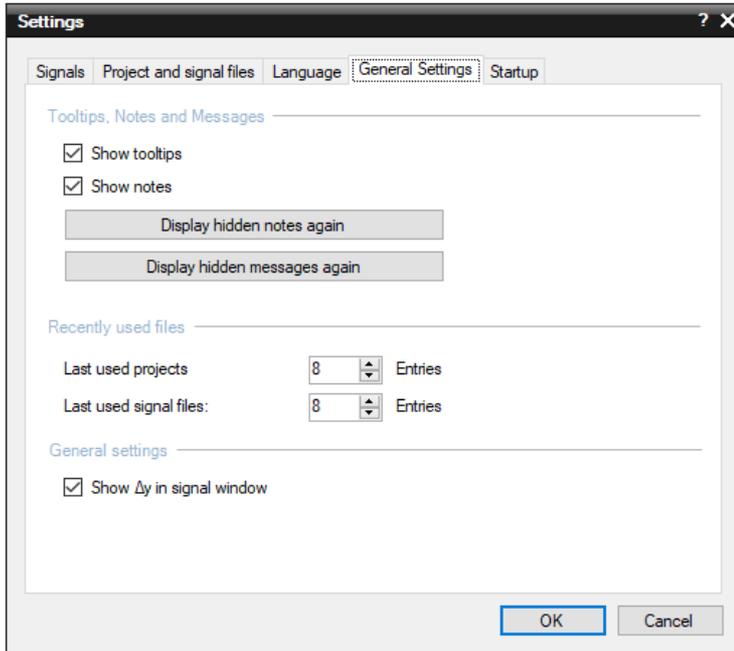


Fig. 3-5 General settings

Activate *display tool tips* to get tips on individual menu items and elements as soon as you hold the mouse pointer over the element for a short time. By activating *show notes*, notes are displayed for each useful operating instruction while working with the PLC-ANALYZER pro.

Notes, also some messages and warnings can be deactivated by clicking the '*Do not show hint any more*' check box. You can reactivate these messages by choosing '*Display no longer displayed notes*' or '*Display no longer displayed messages again*'.

Under *List of recently used files* you can specify the number of entries for recently used projects and signal files in the menu.

Activate *Show Δy in signal window* to display the deltas of the signal values when a time difference is displayed.

Working with PLC-ANALYZER pro 6

This chapter describes the procedure for recording data from a PLC and evaluating them. With the PLC-ANALYZER pro 6, each data acquisition is controlled via a so-called project. All settings required for data acquisition are stored in a project.

Create new project

Select the menu item [New project...](#) in menu [File](#) or [<New project>](#) in the project tree to create a new project. In the window *New project*, enter a *Name* for the new project and adjust the *path* if necessary. You also assign a *description* to the project here. The *description* can also be changed later under [Project-Info](#).

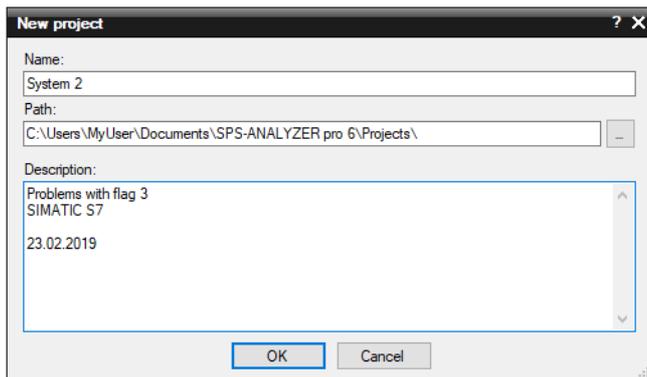


Fig. 4-1 Definition of a new project

Press *OK* to create the project with the specified name. Then you can go on with [processing of the project](#).

Open project

Open a project in the menu *File* either via the history list (contains the most recently loaded projects) or via *Open project...* Alternatively, click the icon  in the toolbar.

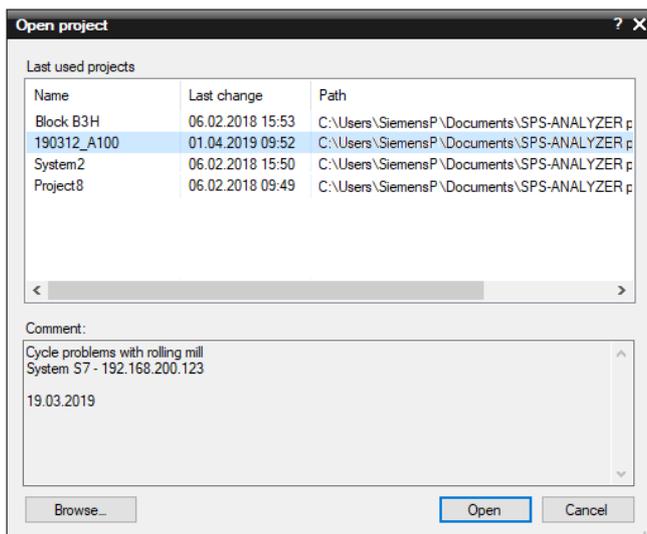


Fig. 4-2 Open project

From the *recent projects* list, select the project you want to open. Click *Browse* if you want to open a project that is not in the list.

Under *Comment* you can see the description assigned to the project.

Selecting the *Open* button opens the project and displays it in the project tree.

Edit project

All objects of the current project are displayed in a tree structure on the left-hand side of the PLC-ANALYZER pro 6. The entire editing of the project is carried out via these *project settings*.

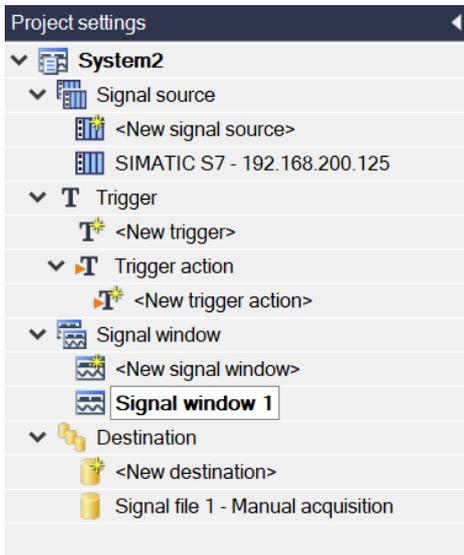


Fig. 4-3 Project settings

Add Object:

Add by double-clicking a [<New signal source>](#), a [<New trigger>](#), a [<New signal window>](#) or a [<New destination>](#).

Paste a previously copied object by clicking the right mouse button on *Paste*.

Edit object:

Double-clicking on an existing object opens the object's editing window. Here you make all settings for the selected object.

Delete object:

Right-click on the object and choose menu item *Delete*.

Copy object:

Right-click on the object and select the menu item *Copy*. This copied object can be pasted into another project or as a copy into the existing project.

Rename object:

Right-click on the object and select the *Rename* menu item. Alternatively, click on the selected project again and give it a new name.

Signal source

Signal sources are the project's data providers. Signal sources can be PLC drivers, [pseudo-signals](#) or special drivers (video track, OPC-UA, AD_USB-Box®). They provide data that can be used by the other objects (such as [signal window](#) or [trigger](#)) of the project.

Add signal source



Fig. 4-4 New signal source

In the project settings, double-click *<New signal source>* in order to insert a new source into the project.

You can also add previously [copied signal sources](#). To do this, click on *Signal source* with the right mouse button and choose *Insert*.

In window *Add signal source*, select the desired signal source. The signal source can be a PLC driver, a [pseudo signal](#) or special driver (video track, OPC-UA, AD_USB-Box®).

With *Add*, the selected signal source is transferred to the project and the [configuration window](#) of the signal source is opened.

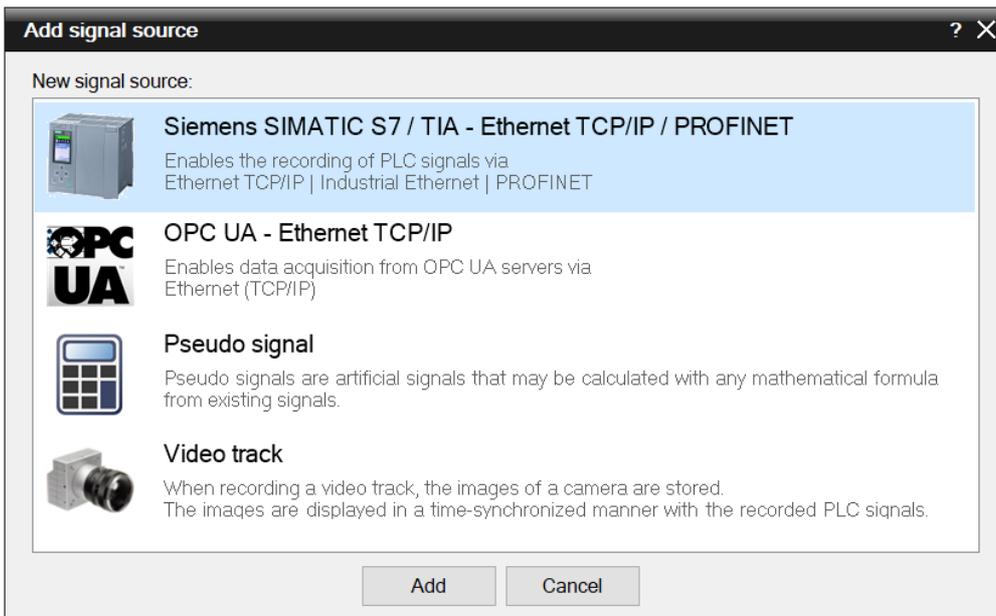


Fig. 4-5 Add Signal source

Edit signal source

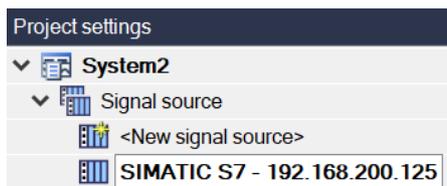


Fig. 4-6 Edit signal source

Double-click on the signal source whose settings you want to configure.. Alternatively, right-click on the signal source and select menu item *Properties*.

The settings of the respective signal sources are basically always the same. The settings of the driver are located on the left side in the upper area. In addition to the *name*, you can also set the communication *properties* for the selected signal source. These are different for each driver and are explained in the chapters of the respective [driver](#).

You also [enter the addresses](#) for recording and [scale](#) them.

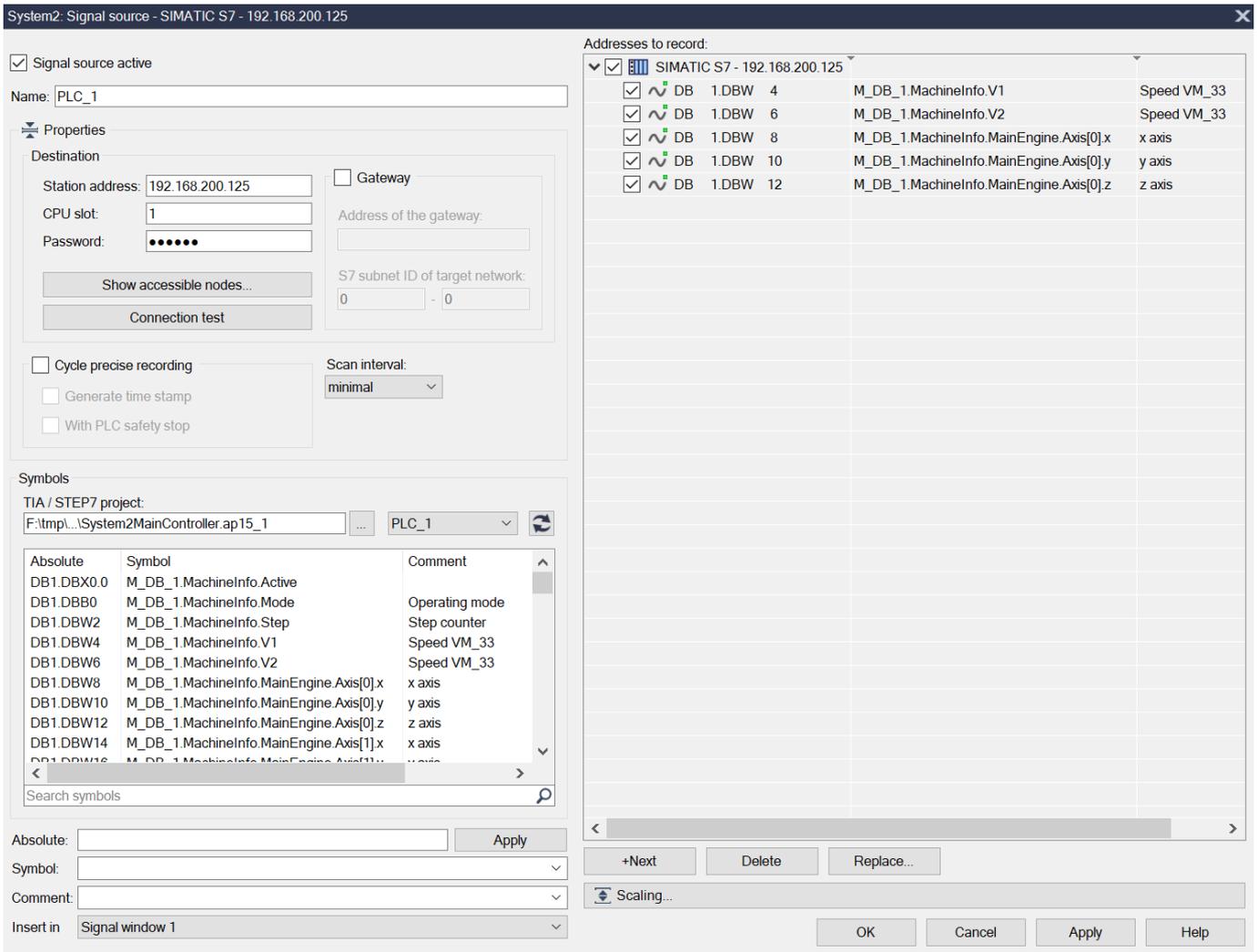


Fig. 4-7 Settings of a signal source

Address selection

Add PLC signals for data acquisition to the lower left of the settings window. The following options are available:

Manual address entry

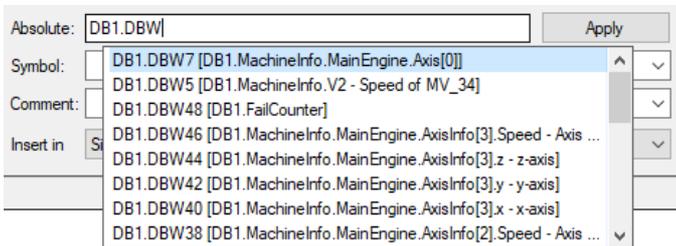


Fig. 4-8 Manual address entry

Under *Absolute* successively enter the desired addresses according to the syntax of the selected signal source. If a symbol file is loaded, the input is automatically completed and the *symbol* and *comment* are automatically taken over. The *symbol* or *comment* can also be entered manually. With *Apply* or <ENTER>, the entered address is included in the list *Addresses for data acquisition*.

Address entry via a symbol file

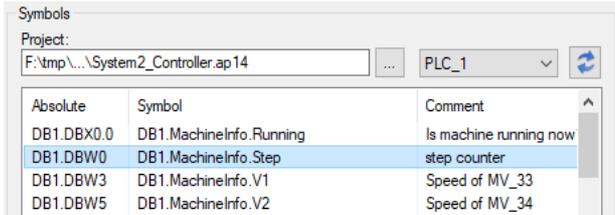


Fig. 4-9 Address entry via a symbol file

In the settings of most signal sources it is possible to load symbol files created by the respective programming software or the project of the programming software. The exact procedure can be found in the chapters of the respective [Driver](#).

If a symbol file is loaded, all symbolic identifiers and comments of the symbol file are listed under *symbols*.

Double-click a symbol address or drag it from the symbol list to the list *addresses for recording*.

To search for a specific symbol, enter the initial letters of the symbol in Symbol. A list with the corresponding search results is then displayed

Address set via the clipboard

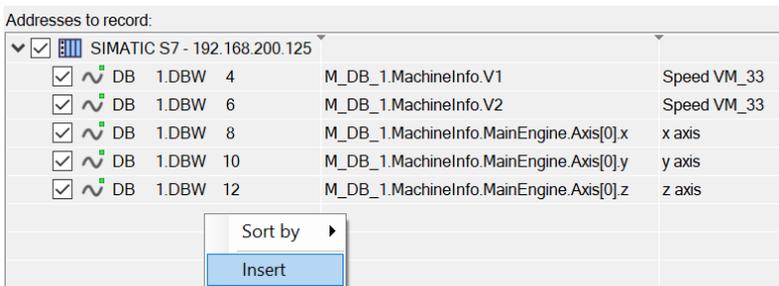


Fig. 4-10 Address set via the clipboard

Right-click in the list *addresses for recording*, and choose *Paste* to paste copied addresses from the clipboard.

You can copy addresses from the PLC ANALYZER pro 6 to the clipboard. To do this, select the desired signals in the list *Addresses for recording* and press <Ctrl> + <C> or right-click on them and choose *Copy*.

However, you can also copy from most PLC programming tools or Microsoft® Excel® address lists to the clipboard.

All selected addresses are included in the List *Addresses for recording*. Under *Paste* in the signal window, select where you want to insert the new addresses. If you want to enter several addresses in a sequence, e. g. EB0, EB1, EB2, click *+Next*.

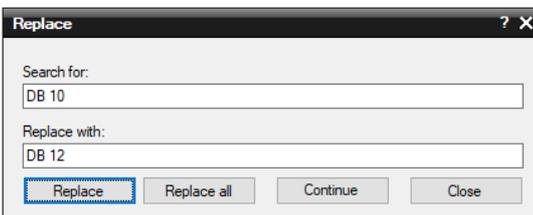


Fig. 4-11 Replace addresses

You can change addresses that have already been inserted by choosing *Replace*. Press *Remove* or <ENTF> to delete addresses of the selection.

+

NOTE

Some automation devices (e. g. SIMATIC S7) permit only byte-wise data acquisition. In this case, the PLC ANALYZER pro 6 automatically converts each bit address entered into a byte address. All bits are then available for display.

Scaling of register values

After the input, the *Scaling* of the selected addresses can be adjusted. To expand the scaling settings click on  or click  to close them again.

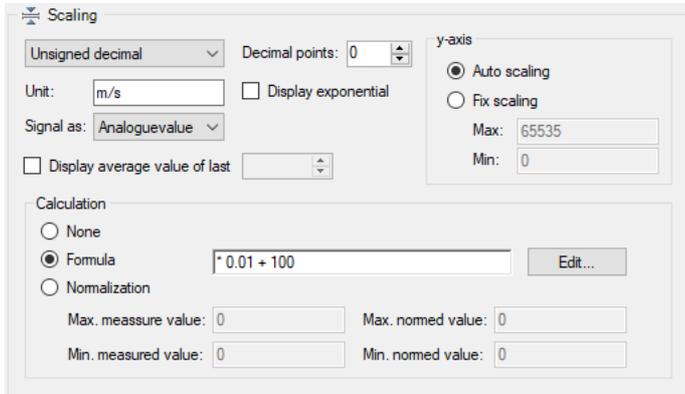


Fig. 4-12 Scaling of addresses

Scaling can be used to influence the display of analog values in terms of their numerical value and the graphical display. For example, you can convert signal values into physical units (m, N, kg, hPa ...) and display them on the screen. To do this, either create a *Formula* under *Calculation* or enter the *normalized values* for the measured values under *Normalization*.

Select the signal to be scaled in the list *Addresses for recording* and, if necessary, change the default values in the corresponding text fields.

First change the format of the selected address. Depending on the selected address, you can select different number formats. Specify the number of *decimal places*. By selecting *Exponential representation*, the numerical value of the address is displayed as a *floating point* number. If you are interested in the bit pattern of a measured value set *Display signal as bit pattern*. In addition, you can display the *average value* over the specified time.

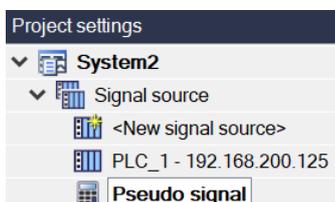
The vertical scale of an analog signal can be automatically adjusted by *autoscaling*. The signal is scaled so that the maximum value is at the upper end of the scale and the minimum value is at the lower end. The auto scaling is default.

If you want fixed scaling of the display values, select *Fixed scaling*. Enter the scaling limits for the display under *Min* and *Max*. The input is made according to the set number format. Fixed scaling is useful, for example, if you want to "zoom" small changes of a measured value in a certain value range. Please note that if the sign interpretation changes, the min-max values may be changed.

Pseudosignal

With PLC-ANALYZER pro 6 you have the possibility to generate pseudo signals from chosen addresses. Pseudo signals are artificial signals, which calculate themselves by means of any mathematical formula by existing signals.

According to the formula $P = U \times I$ (power = voltage x current) for example, the performance of a machine can permanently be calculated and indicated in pseudo signals, if voltage and current are present as measuring signals.



In the project tree select [<New signal source>](#) and insert a pseudo signal.

If pseudo signals already exist, double-click on *pseudo signals* in the project tree to edit or remove existing pseudo signals.

Fig. 4-13 Add Pseudosignal

In the *settings* window, you can *add* new pseudo signals or edit or *remove* existing pseudo signals. For newly created pseudo signals, specify in which [signal window](#) the pseudo signal is to be inserted.

In the address list, select the pseudo signal you want to edit.

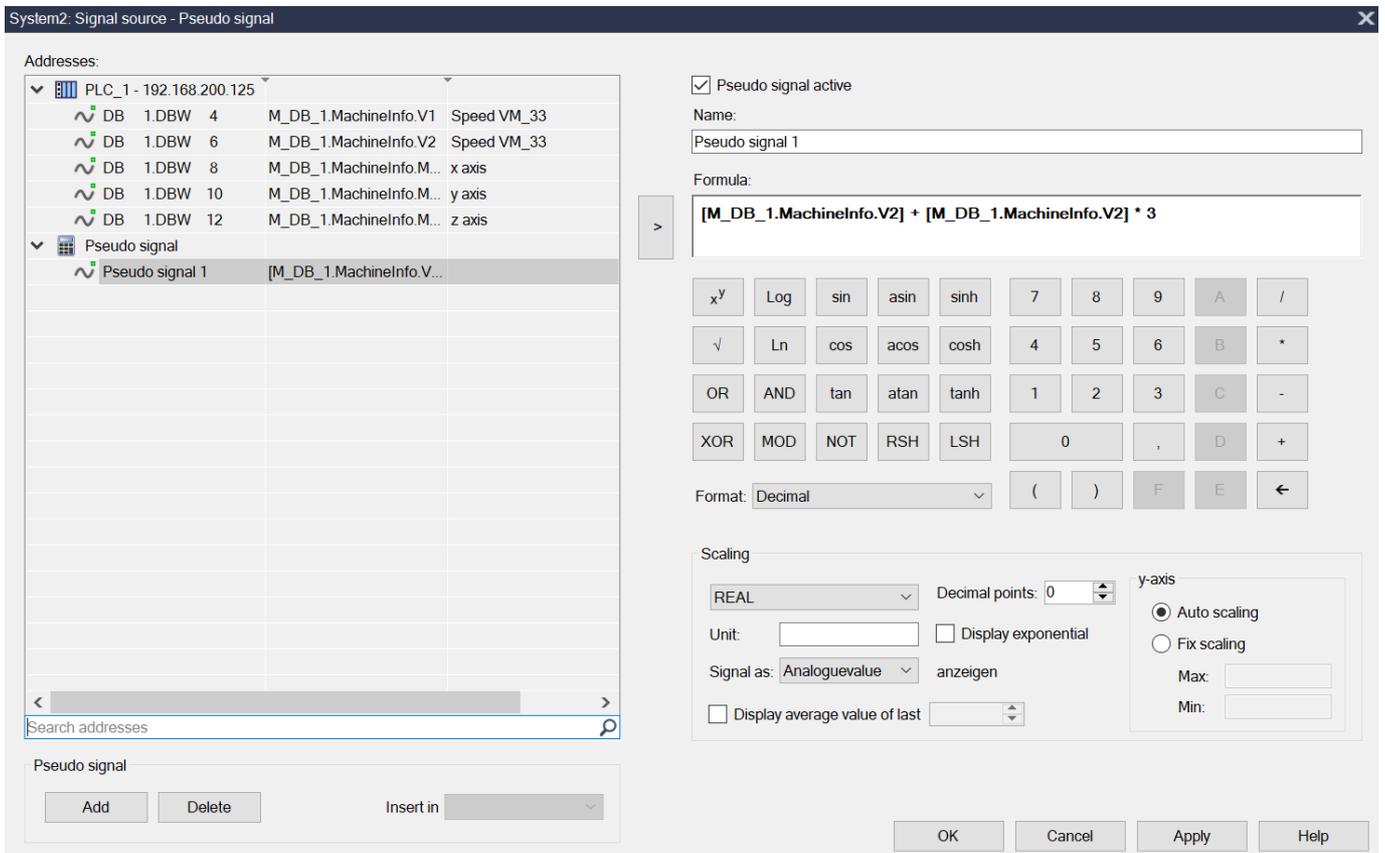


Fig. 4-14 Create pseudo signal

First, give the pseudo signal a meaningful *name*. Then create the pseudo signal *formula*. Add the individual operators and operands by selecting the corresponding buttons. Addresses can also be dragged with the mouse from the *address list* into the *formula*. You set the format of the numerical values under *Format*.

After you have created the pseudo signal, you can adjust the *scaling* of the pseudo signal (see [Edit signal source](#)).

Trigger

The PLC-ANALYZER pro 6 offers the possibility to trigger on certain events, e.g. sporadic errors. With this feature you can create signal files containing a specific event with the signal history leading up to and following the event. In addition, certain trigger actions - such as sending an e-mail - can be executed.

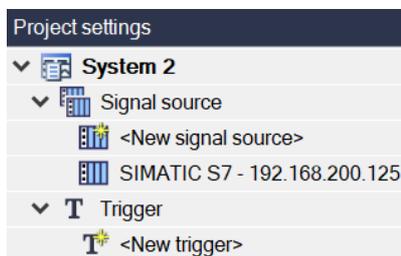
When capturing a large number of signals, and especially when recording a video track, a trigger can be used to considerably reduce the amount of data.

You can trigger on binary and analog signals. For binary signals you can trigger on either logical "0" or "1". For analog signals you can trigger on the relations greater than (>), less than (<), equal (=), and not equal (<>).

Trigger conditions can be combined with logical operators, enabling you to specify complex trigger conditions.

If a trigger is triggered during data acquisition, this is indicated in the [signal window](#) by a dashed vertical line. This is also entered as event (**T**) in the [event display](#).

Create new trigger



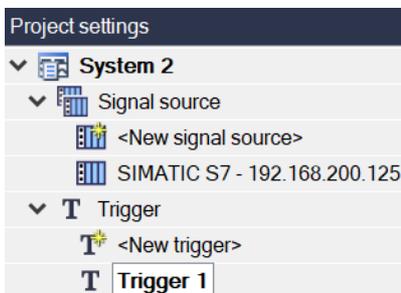
In the project settings, double-click <New Trigger> to insert a new trigger into the project

You can also add previously [copied trigger](#). To do this, right-click on *Trigger* and select the menu item *Insert*.

After inserting the trigger, the trigger [Configuration window](#) opens.

Fig. 4-15 New Trigger

Edit Trigger



Double-click the trigger whose settings you want to configure.

Alternatively, right-click on the trigger and select the menu item *Properties*.

Fig. 4-16 Edit Trigger

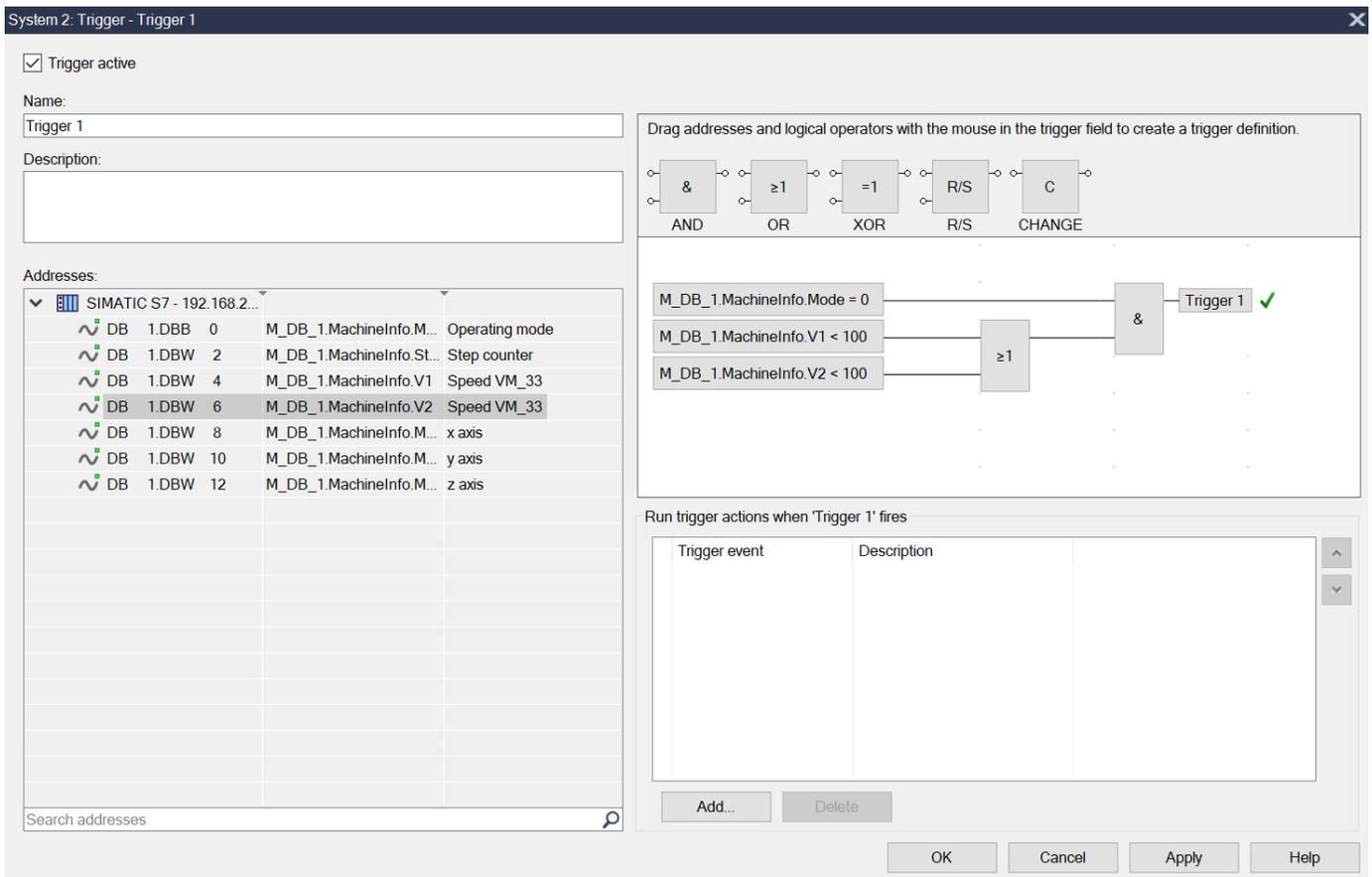


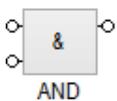
Fig. 4-17 Defining the trigger conditions

Enter a name for the trigger and assign a *description* to it. Use the check box *Trigger active* to activate or deactivate the trigger. If the trigger is deactivated, no trigger is triggered even if the trigger condition is fulfilled.

Now define the trigger condition. For this purpose, all signals from the *address* list and the logic operators AND, OR, XOR, R/S and CHANGE are available. Drag *addresses* or logic operators into the trigger field to define the trigger condition. Then connect the inputs and outputs of the modules with the mouse to a trigger condition.

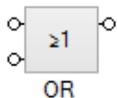
Logical operators

The trigger conditions can be linked with different logical operators, so that even complex trigger expressions can be defined. Use the mouse to drag the operators from the upper area into the trigger field.



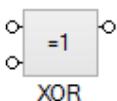
AND

The output is 1 = TRUE if all inputs are 1 = TRUE. If one or more inputs are 0 = FALSE, the output is also 0 = FALSE.



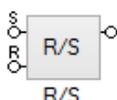
OR

The output is 1 = TRUE, if one or more inputs are 1 = TRUE. If all inputs are 0 = FALSE the output is also 0 = FALSE.



XOR

The output is 1 = TRUE if one input 1 = TRUE and the other input is 0 = FALSE. If both inputs are 1 = TRUE or both 0 = FALSE, the output is also 0 = FALSE.

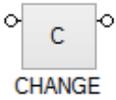


R/S-Flip flop

An RS flip-flop has two inputs designated R (reset) and S (set).

If the S-input is set to 1 = TRUE, the output will be set to 1 = TRUE. If the R input is set to 1 = TRUE, then the output is set to 0 = FALSE. If there is a 0 = FALSE signal at both inputs, then the output is

in the last set state. If there is a 1 = TRUE signal on both inputs, the output is also 0 = FALSE.



CHANGE

The output is 1 = TRUE if the input has changed since the last check (from 0 = FALSE to 1 = TRUE or vice versa).

Properties of a Trigger logical operator

Under *Triggering time*, set how long the logic operation must be completed so that the output of module becomes 1 = TRUE. Select the *number of inputs* that the module should have. By selecting the *Negate* control panels you can negate individual inputs and the output of the module.

Confirm your selection with *OK*.

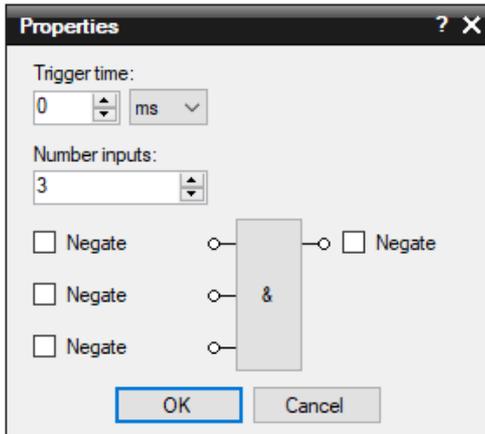
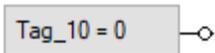


Fig. 4-18 Properties of a Trigger logical operator

Trigger-Address module

Drag addresses from the *address list* to the desired position in the trigger field.



Address

The output is 1 = TRUE if the value of the address meets the set condition.

Input of the trigger value of a binary signal

After dragging the bit signal address into the trigger field, the *trigger value* window opens. If the address module is already in the trigger field, double-click the address module to open the *trigger value* window.

Select either the trigger value 1 = TRUE or 0 = FALSE. The operator is permanently set to "=". Under *trigger time*, set how long the set condition must be fulfilled for the output of the module to become 1 = TRUE.

Confirm your selection with *OK*.

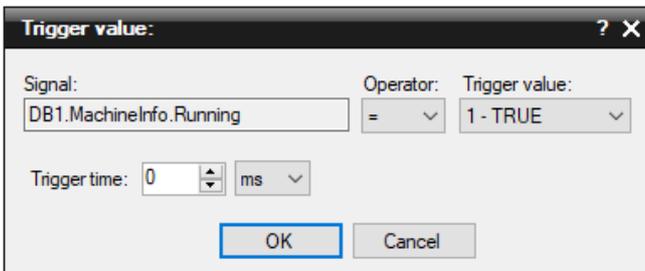


Fig. 4-19 Input of the trigger value of a binary signal

Input of the trigger value of an analog signal

After dragging the analog value address into the trigger field, the *trigger value* window opens. If the address module is already in the trigger field, double-click the address module to open the *trigger value* window

Under *Operator*, select one of the six comparison operators. Then enter the corresponding numerical value under *trigger value*.

A *hysteresis* can be set to avoid bounce of the trigger (multiple triggering) with small signal fluctuations in order to avoid the trigger value. The program uses the entered hysteresis to automatically determine the *upper* or *lower shutdown*. When triggering to equal "=" and unequal "< >", the entered hysteresis is used as a measure for the blur.

Under *trigger time*, set how long the set condition must be fulfilled for the output of the module to become 1 = TRUE.

Confirm your selection with *OK*.

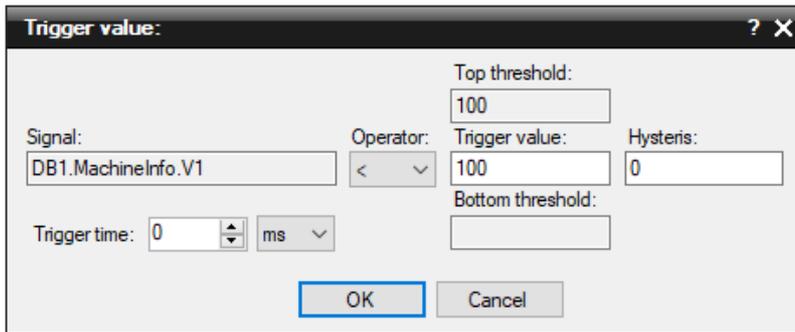


Fig. 4-20 Input of the trigger value of an analog signal

The following table explains the function of the various trigger operators:

Trigger operator	Description
<	The trigger is triggered when the value falls below the trigger value. A new trigger is only registered again if the upper cut-off has previously been exceeded.
<=	The trigger is triggered when the trigger value is reached or fallen below. A new trigger is not registered again until the upper cut-off has been exceeded.
>	The trigger is triggered when the trigger value is exceeded. A new trigger is only registered again when the value has fallen below the lower cut-off value.
>=	The trigger is triggered when the trigger value is reached or exceeded. A new trigger is only registered again when the value falls below the lower cut-off value.
=	The trigger is triggered when the measured value is in the range between upper and lower cut-off. A new trigger is only registered again if this range has previously been left.
< >	The trigger is triggered when the measured value leaves the range between upper and lower cut-off. A new trigger is only registered again if a measured value was previously within the range..

Tab. 4-1 Trigger operator

The following graphic illustrates the principle of analog triggering.

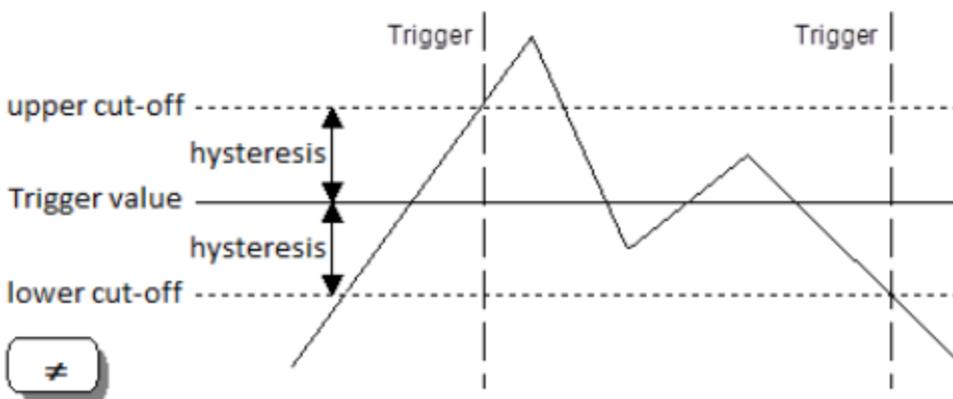
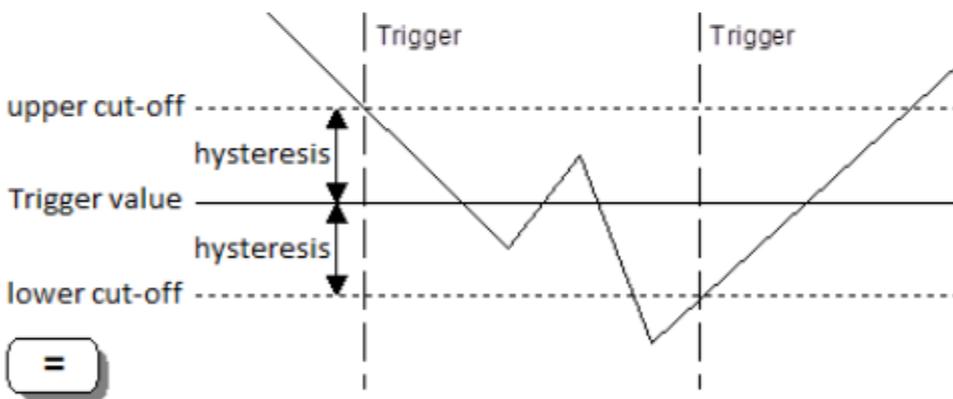
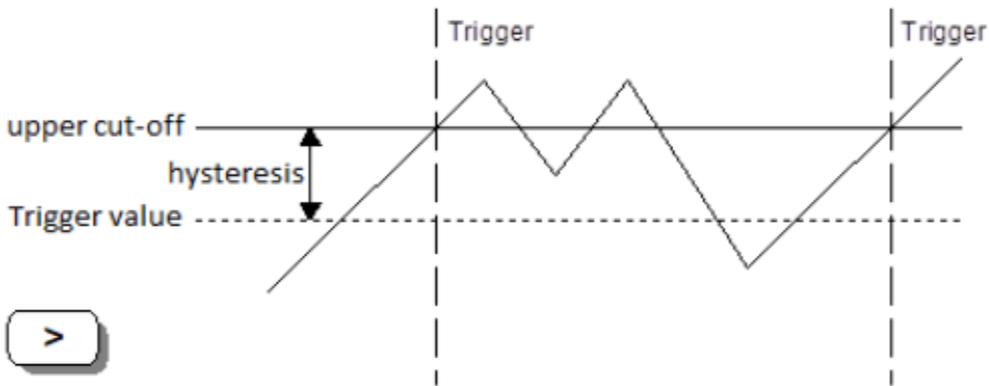
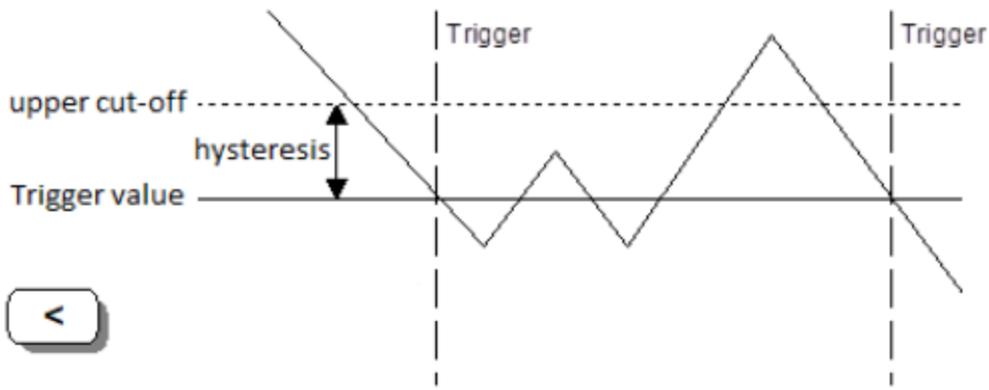


Fig. 4-21 Principles of analog triggering (<, >, =, \neq)

Trigger-Actions

The PLC ANALYZER pro 6 offers the possibility to perform different [Trigger actions](#) when a trigger is triggered. A prerequisite for this is that trigger actions have already been defined (see [Creating a new Trigger action](#)).

Click *Add* to open the window *Add Trigger Action*. Select the desired trigger actions and click *Add* to apply the selected actions.

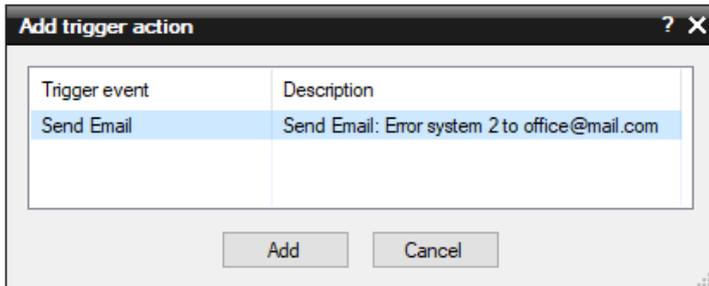


Fig. 4-22 Add Trigger-Action

Change the execution sequence of the trigger actions with  and . You can delete added trigger actions with *Delete*.

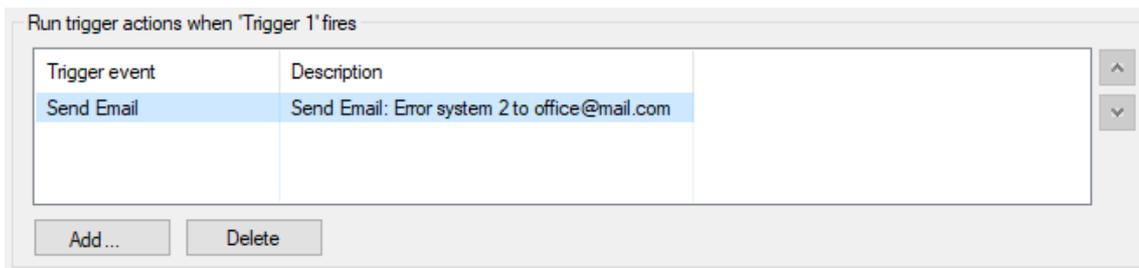
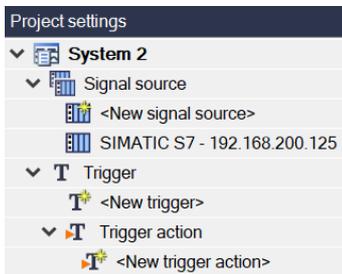


Fig. 4-23 Selected trigger actions

Trigger-Action

The PLC ANALYZER pro 6 offers the possibility to execute different trigger actions when a trigger is triggered.

Create new Trigger-Action



In the project settings, double-click <New Trigger Action> to insert a new trigger action into the project.

You can also add previously [copied Trigger-Action](#). To do this, right-click on Trigger Action and choose *Insert*.

Fig. 4-24 New Trigger-Action

In the window *Add Trigger Action* select the desired trigger action. With *Add*, the selected trigger action is transferred to the project and the [Configuration window](#) of the trigger action is opened.

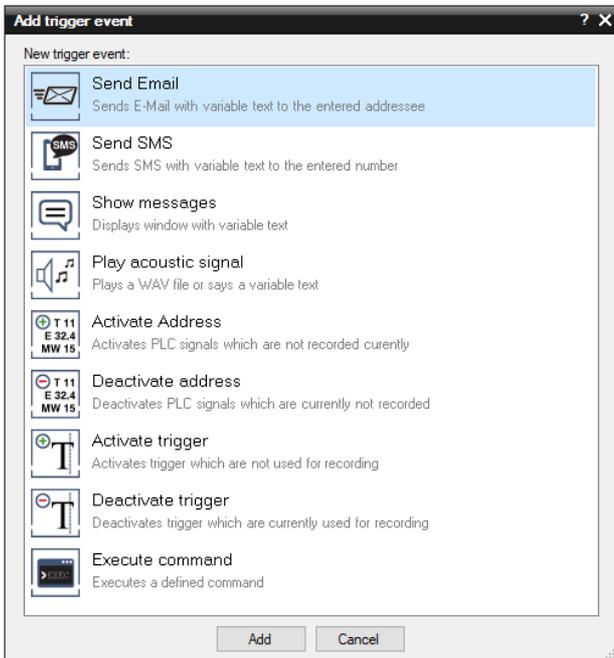
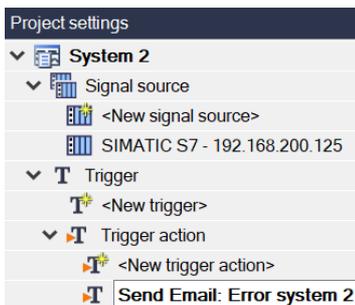


Fig. 4-25 Add Trigger-Action

Edit Trigger-Action



Double-click the trigger action whose settings you want to configure. Alternatively, right-click on the trigger action and choose *Properties*.

Fig. 4-26 Edit Trigger-Action

Depending on the selected trigger action, the respective settings window opens, in which you can adjust the individual parameters of the trigger action.

Send E-Mail

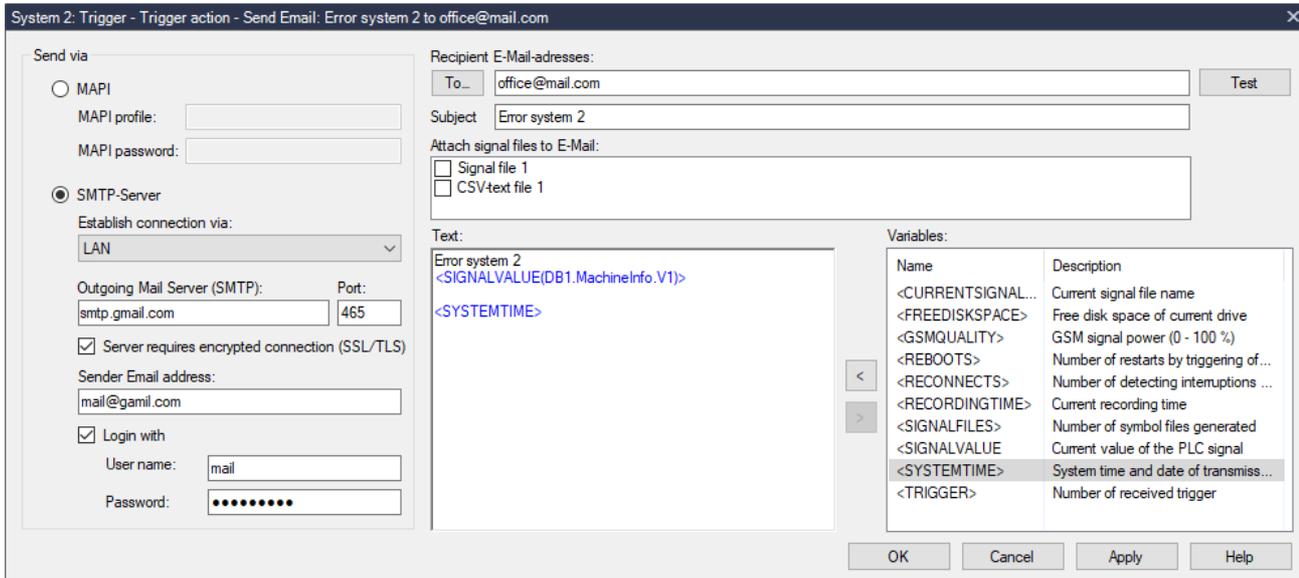


Fig. 4-27 Settings Trigger action "Send e-mail"

To configure E-Mail dispatch, select whether the e-mail is to be sent via MAPI or via an SMTP server.

When sending using MAPI, enter the existing MAPI profile and MAPI password.

If the E-Mail is to be sent via an *SMTP server*, specify which network connection you want to use to connect to the SMTP server. A prerequisite for this is that a network connection has already been set up under Windows. Please refer to the Windows Help for instructions on how to create such a connection.

In addition to the *outgoing mail server (SMTP)*, enter the *sender's E-Mail address* and, if necessary, a *user name* and a *password*. If the server requires a secure SSL connection, activate *Server requires encrypted connection (SSL/TLS)*.

Enter one or more *email addresses* and specify a subject line. Select signal files, which should be add to the email as an attachment in *Attach signal files to E-Mail*.

Now add the text to be sent.

Here you can insert variables in addition to normal text. To do this, simply drag a corresponding variable from the Variables list into the text field or use the buttons < and > .

Variable	Description
<CURRENTSIGNALFILE>	Current signal file
<FREEDISKSPACE>	Free disk space current drive
<GSMQUALITY>	GSM Signal quality (0 - 100 %)
<REBOOTS>	Number of reboots BLACKBOX (Watchdog)
<RECONNECTS>	Number of acquisition interruptions (Reconnection)
<RECORDTIME>	Current recording time
<SIGFILES>	Number of created signal files
<SIGVALUE(X)>	Current value of PLC-Signals X
<SYSTIME>	System time and date at the time of transmission
<TRIGGER>	Number of triggered triggers

With *Test* you can test whether all settings are correct and the e-mail is sent correctly.

Send SMS

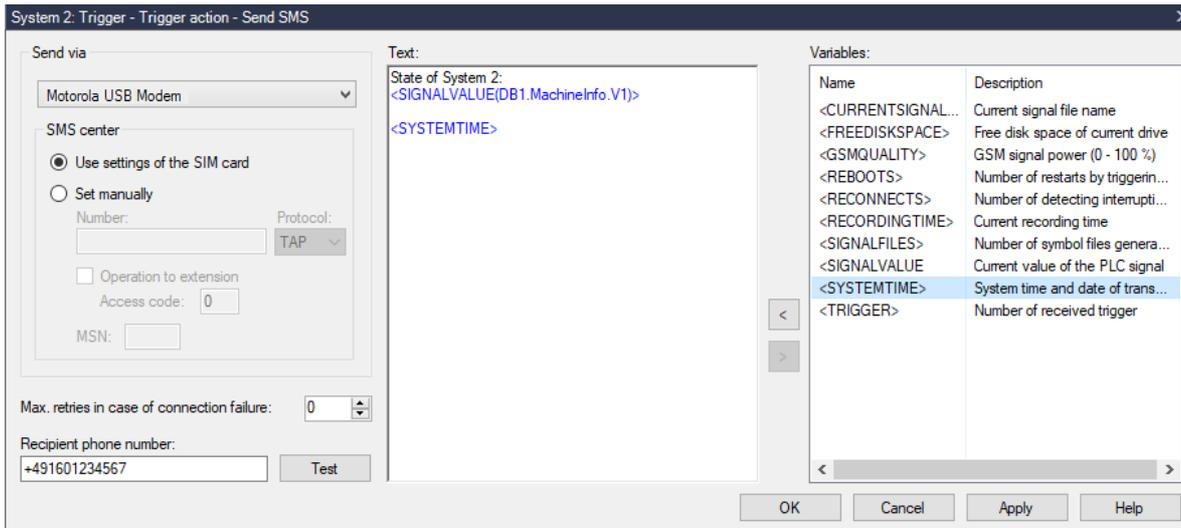


Fig. 4-28 Settings Trigger action "Send e-mail"

To configure SMS sending, first select the connection via which the SMS is to be sent. Prerequisite for this is the installation of a modem.

Now select the *SMS central*. You can use the *settings of the SIM card* or enter them *manually*. When entering manually, enter the *number* of the SMS center and the *protocol* supported by the SMS center. If necessary, enter your *MSN* number and activate the check box *Operation on Extension* if your modem is connected to an extension. If necessary, also enter an *outside access code*. Set the *Max. Repeats in case of connection error*.

Enter the recipient's number and insert the *text* to be sent.

Here you can insert variables in addition to normal text. To do this, simply drag a corresponding variable from the Variables list into the text field or use the buttons < and >.

Variable	Description
<CURRENTSIGNALFILE>	Current signal file
<FREEDISKSPACE>	Free disk space current drive
<GSMQUALITY>	GSM Signal quality (0 - 100 %)
<REBOOTS>	Number of reboots BLACKBOX (Watchdog)
<RECONNECTS>	Number of acquisition interruptions (Reconnection)
<RECORDTIME>	Current recording time
<SIGFILES>	Number of created signal files
<SIGVALUE(X)>	Current value of PLC-Signals X
<SYSTIME>	System time and date at the time of transmission
<TRIGGER>	Number of triggered triggers



NOTE

The length of the text to be sent is limited to 160 characters.

With *Test* you can test whether all settings are correct and the SMS is sent correctly.

Show Message

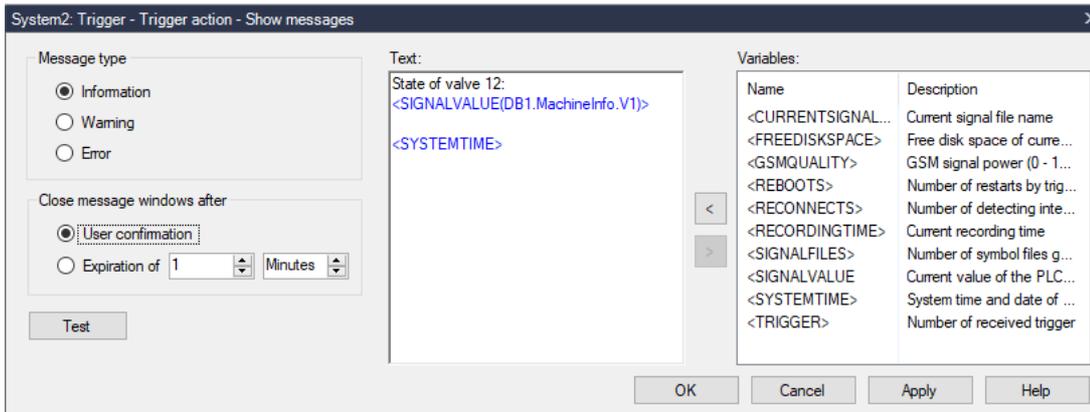


Fig. 4-29 Settings Trigger action "Display message"

Specify the *message type* and specify whether the message window should be closed with a *user confirmation* or automatically after the *Expiration of* the specified time.

Now insert the *text* to be displayed. Here you can insert variables in addition to normal text. To do this, simply drag a corresponding variable from the Variables list into the text field or use the buttons and .

Variable	Description
<CURRENTSIGNALFILE>	Current signal file
<FREEDISKSPACE>	Free disk space current drive
<GSMQUALITY>	GSM Signal quality (0 - 100 %)
<REBOOTS>	Number of reboots BLACKBOX (Watchdog)
<RECONNECTS>	Number of acquisition interruptions (Reconnection)
<RECORDTIME>	Current recording time
<SIGFILES>	Number of created signal files
<SIGVALUE(X)>	Current value of PLC-Signals X
<SYSTIME>	System time and date at the time of transmission
<TRIGGER>	Number of triggered triggers

With *Test* you can test whether all settings are correct and the message window is displayed correctly.

Play acoustic signal

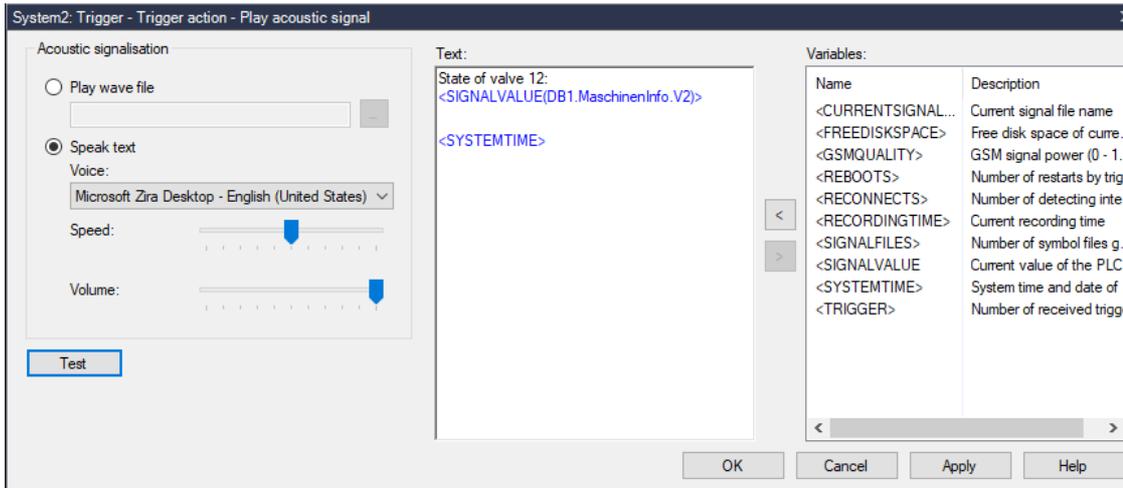


Fig. 4-30 Settings Trigger action "Playing acoustic signal"

Select whether you want to play a WAV file or announce a text.

To play a WAV file, select the appropriate file.

To announce a text, first select an installed voice. Then set the speed and volume of the voice. Now insert the text to be announced.

Here you can insert variables in addition to normal text. To do this, simply drag a corresponding variable from the Variables list into the text field or use the buttons < and > .

Variable	Description
<CURRENTSIGNALFILE>	Current signal file
<FREEDISKSPACE>	Free disk space current drive
<GSMQUALITY>	GSM Signal quality (0 - 100 %)
<REBOOTS>	Number of reboots BLACKBOX (Watchdog)
<RECONNECTS>	Number of acquisition interruptions (Reconnection)
<RECORDTIME>	Current recording time
<SIGFILES>	Number of created signal files
<SIGVALUE(X)>	Current value of PLC-Signals X
<SYSTIME>	System time and date at the time of transmission
<TRIGGER>	Number of triggered triggers

With *Test* you can test whether all settings are correct and the acoustic signal is played back correctly.

Activate / deactivate addresses

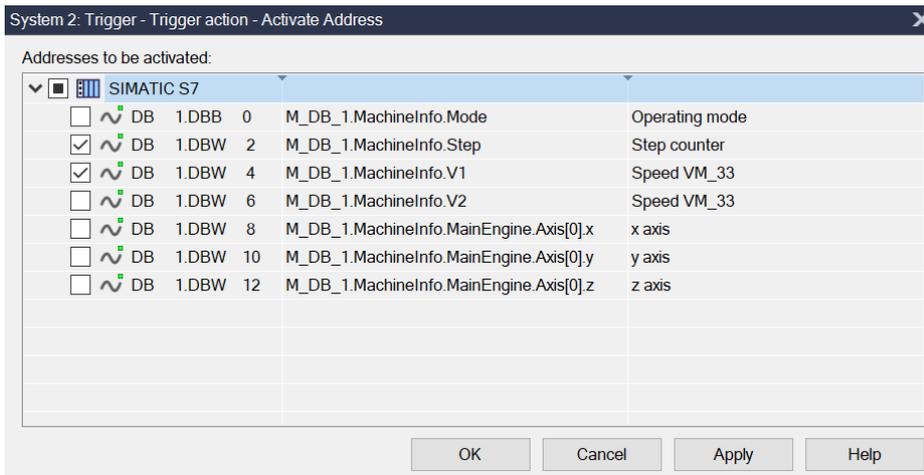


Fig. 4-31 Settings Trigger action "Activate / deactivate addresses"

If a trigger occurs, certain addresses can be activated or deactivated for acquisition. This is useful, for example, if you use a trigger condition to trigger a specific fault in a particular part of a system and in this case want to capture specific signals from the relevant part of the system and investigate them in more detail.

Select which addresses are to be activated or deactivated by the executed trigger action. Select the corresponding address by selecting the checkbox in front of the respective address.

If the trigger action "Activate addresses" is executed by a trigger, the selected addresses are activated. After triggering the trigger, the activated addresses are recorded.

If the trigger action "Deactivate addresses" is executed by a trigger, the selected addresses are deactivated. After triggering the trigger, the acquisition of the deactivated addresses is stopped.

Activate / deactivate trigger

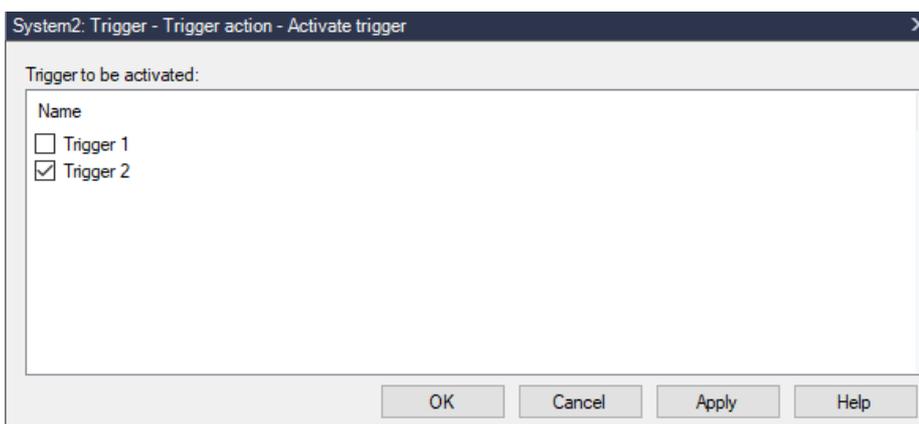


Fig. 4-32 Settings Trigger action "Activate / deactivate trigger"

Select which triggers are to be activated or deactivated by the executed trigger action. Select the corresponding trigger by selecting the check box in front of the respective trigger.

If the trigger action "Activate Trigger" is executed by a trigger, the selected triggers are activated. If the trigger action "Deactivate Trigger" is executed by a trigger, the selected triggers are deactivated.

Execute command

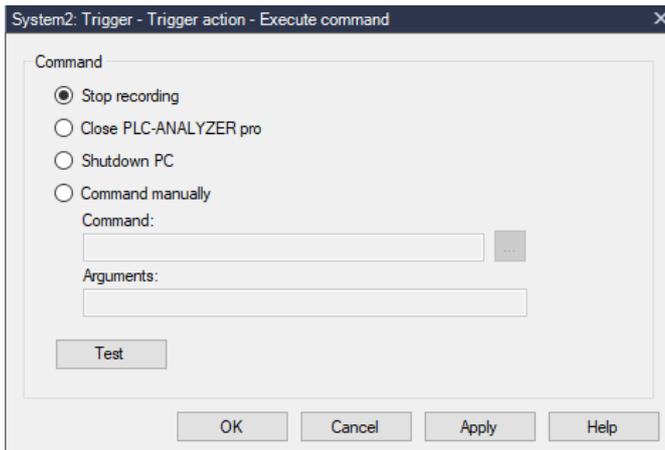


Fig. 4-33 Settings Trigger action "Execute command"

Select the *command* to be executed when a trigger executes this trigger action. The following table describes the selectable commands.

Command	Explanation
Stop acquisition	Terminates the current data acquisition and displays the last recorded signal file.
Exit PLC-ANALYZER pro	Terminates the current data acquisition and closes the PLC-ANALYZER pro 6.
Turn off PC	Terminates the current data acquisition and switches off the PC.
Manual command	Manually select a <i>command</i> here and enter any <i>arguments</i> you may need.

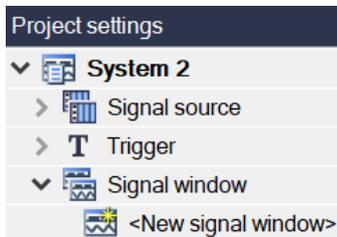
Table 4-2 Commands

You can use *Test command* to test whether all settings are correct and whether the command is executed correctly.

Signal window

The recorded data of selected PLC signals are displayed in a signal window. The recorded measured values are displayed over time in the form of a line recorder or oscilloscope or as a text table. There is also a video signal window for displaying images from a camera.

Create new signal window



In the project settings, double-click *<New signal window>* to insert a new signal window into the project

You can also add previously [copied signal window](#). To do this, right-click on *Signal window* and select the menu item *Insert*.

Fig. 4-34 New signal window

In the window *Add signal window*, select the desired window type.

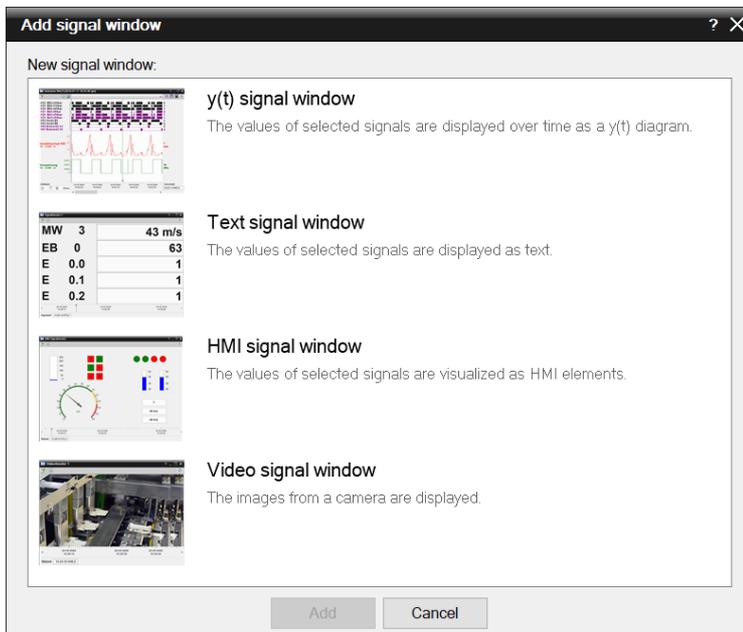


Fig. 4-35 Add signal window

y(t) signal window

In a y(t)-signal window, the measured values of selected signals are displayed over time according to the type of line recorder or oscilloscope (y(t)-diagram).

Text signal window

In a text signal window, the measured values of selected signals are displayed as text in tabular form.

HMI signal window

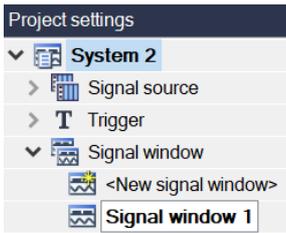
In an HMI signal window, the acquired measured values of selected signals are displayed as HMI elements.

Video signal window

In a video signal window, the images of a camera are displayed.

With *Add*, the selected signal window is transferred to the project and the [configuration window](#) of the window is opened.

Edit y(t) signal window



Double-click on the signal window whose settings you want to configure. Alternatively, right-click on the signal window and select the *Properties* menu item.

Clicking the button  in the signal window is another way to open the preferences window.

Fig. 4-36 Edit y(t) signal window

Give the signal window a meaningful name. Use *Display signal window* to specify whether the window is to be displayed.

All signals recorded with the current project are listed under *Addresses*. Drag all the addresses that are to be displayed with the mouse into the signal window. Alternatively, you can also use the buttons *All >* and *>*. Use the mouse to position the addresses in the signal window in the desired order.

Addresses that are no longer to be displayed in the signal window can either be dragged back into the settings window with the mouse or by using the *< All* or *<* buttons.

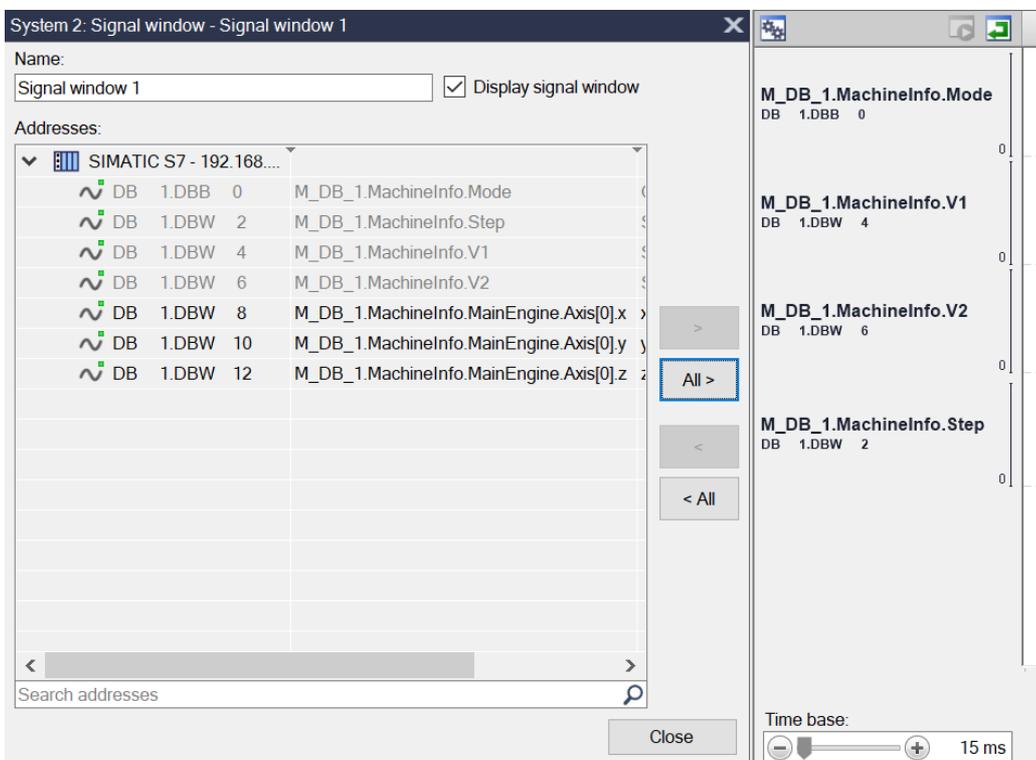


Fig. 4-37 Address selection for the y(t) signal window

Now adjust the signal window according to your wishes. The following setting options are available:

Absolute / relative time display

In the lower part of the signal window there is a time scale. The *cursor time* text field also displays the current time at the cursor position. You can choose between two different display options:

- Relative: Time display relative to the beginning of the signal file
- Absolute: System time (computer time) at the time the signal data is received.

You can change the time display via the menu item *Absolute time display* in the menu *View* or via the icon  in the toolbar of the signal window. The general format of the time display is: hh: mm: ss. xxx, y (hh = hour; mm = minute; ss = second; xxx, y = millisecond).

Display grid

By selecting the menu item *Grid* in the menu *View* or using the icon  in the toolbar of the signal window, a grid can be laid over the display area of the signal window.

Display scan points

Scan points are displayed as small dots on the signal curve. These mark the time at which the signal was scanned. Activate the display of the scan points via the menu item *Display scan points* in the menu *View* or via the icon  in the toolbar of the signal window.

Bar display

Specify whether bit signals are to be displayed in bar display or as lines. Activate the bar display via the *Bar Display* menu item in the menu *View* or via the icon  in the toolbar of the signal window.

Switch scroll / overwrite

During data acquisition, recorded signal values are displayed live in the signal window. Select whether the displayed signals should be overwritten after a single pass or whether they should be scrolled out of the window to the left. Activate the corresponding display type using the icon  or  in the toolbar of the signal window.

Switch live / analysis

You can already view and evaluate signal data during recording. Click the icon  in the toolbar of the signal window to pause the live view and switch to analysis view mode. The signal data from the signal file is now displayed and you can analyze and evaluate it. The data recording continues to run in the background. Press  to return to the live display mode

Change signal colour

In the address field on the left side of the signal window, select all signals whose colour you want to change. Simply click on the corresponding signal. By holding down the <Ctrl> or <Shift> keys and selecting several signals simultaneously with the mouse, several signals can be selected. Then press the right mouse button on a selected signal and select the menu item *Change signal colour*. In the *Change Signal Colour* window, select the desired *signal colour* and confirm with *OK*.

Change signal position and height

In the signal window, select all signals whose position is to be changed. Simply click on the corresponding signal. By holding down the <Ctrl> or <Shift> keys and selecting several signals simultaneously with the mouse, several signals can be selected. Then press the left mouse button on a selected signal and move the signals with the left mouse button pressed down.

The height of a selected analog signal can be changed by changing the position of the upper or lower end of the scale of the signal with the mouse. The height of a bit signal cannot be changed. The height always corresponds to the standard binary signal level set in the [options](#).

Synchronize cursors in signal windows

By selecting the symbol  in the toolbar, the cursors in all signal windows are displayed synchronously in time. If the cursor position is changed in one signal window, the cursor is also set to the corresponding time in the other signal windows of the signal file.

Change time base

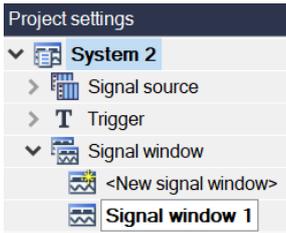


The time base in the lower left area of the signal window indicates the time per screen pixel. During a live data acquisition, the speed of updating the signals is changed with the time base, while in analysis display mode the display of the signal data is stretched or compressed over time.

Change the time base with  and  or by moving the slider  with the mouse.

Alternatively, you can use the mouse wheel with <Ctrl> key held down. 

Edit text signal window



Double-click on the signal window whose settings you want to configure. Alternatively, right-click on the signal window and select the *Properties* menu item.

Clicking the button  in the signal window is another way to open the preferences window.

Fig. 4-38 Edit text signal window

Give the signal window a meaningful name. Use *Display signal window* to specify whether the window is to be displayed.

All signals recorded with the current project are listed under *Addresses*. Drag all the addresses that are to be displayed with the mouse into the signal window. Alternatively, you can also use the buttons *All >* and *>*. Use the mouse to position the addresses in the signal window in the desired order.

Addresses that are no longer to be displayed in the signal window can either be dragged back into the settings window with the mouse or by using the *< All* or *<* buttons.

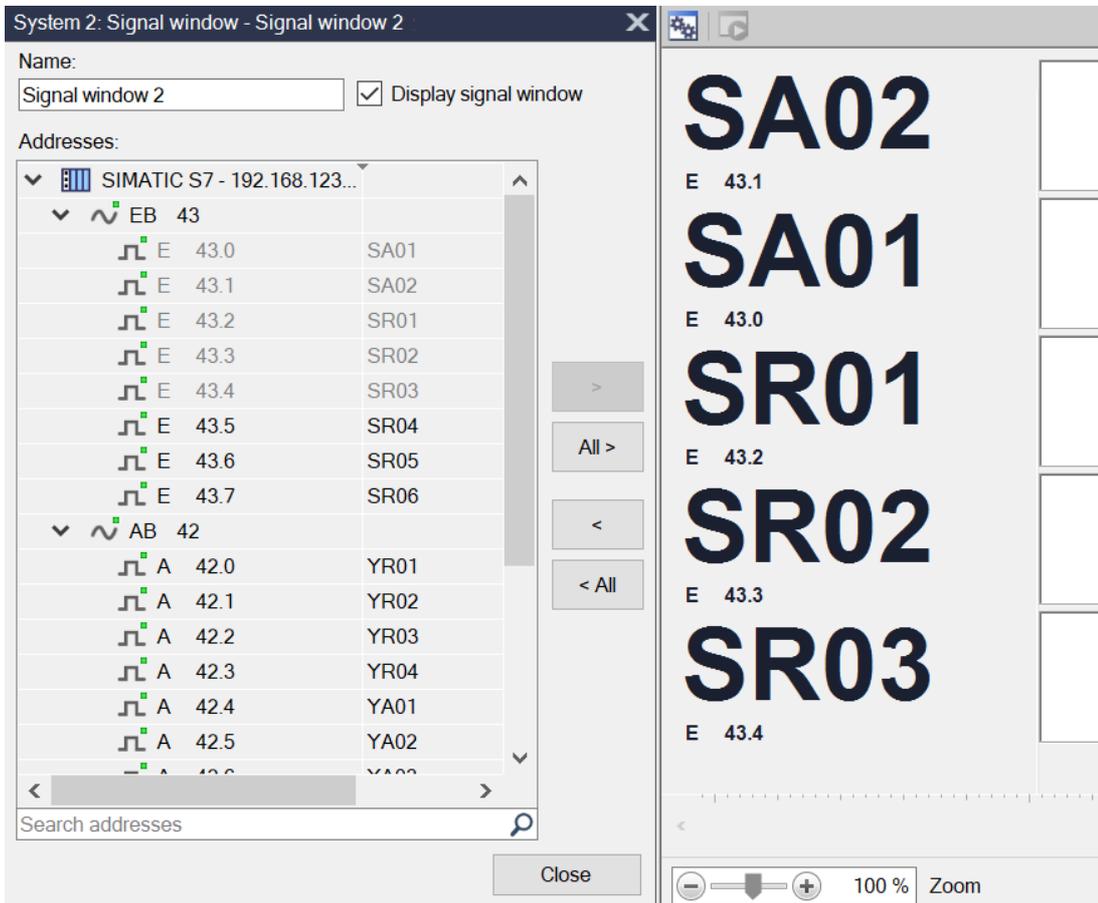


Fig. 4-39 Address selection for the text signal window

Now adjust the signal window according to your wishes. The following setting options are available:

Absolute / relative time display

In the lower part of the signal window there is a time scale. The *cursor time* text field also displays the current time at the cursor position in the time scale. You can choose between two different display options:

- Relative: Time display relative to the beginning of the signal file
- Absolute: System time (computer time) at the time the signal data is received.

You can change the time display via the menu item *Absolute time* display in the menu *View* or via the icon  in the toolbar of the signal window. The general format of the time display is: hh: mm: ss. xxx, y (hh = hour; mm = minute; ss = second; xxx, y = millisecond).

Switch live / analysis

You can already view and evaluate signal data during recording. Click the icon  in the toolbar of the signal window to pause the live view and switch to analysis view mode. The signal data from the signal file is now displayed and you can analyze and evaluate it. The data recording continues to run in the background. Press  to return to the live display mode

Change signal colour

In the address field on the left side of the signal window, select all signals whose colour you want to change. Simply click on the corresponding signal. By holding down the <Ctrl> or <Shift> keys and selecting several signals simultaneously with the mouse, several signals can be selected. Then press the right mouse button on a selected signal and select the menu item *Change signal colour*. In the *Change Signal Colour* window, select the desired *signal colour* and confirm with *OK*.

Change signal position

In the signal window, select all signals whose position is to be changed. Simply click on the corresponding signal. By holding down the <Ctrl> or <Shift> keys and selecting several signals simultaneously with the mouse, several signals can be selected. Then press the left mouse button on a selected signal and move the signals with the left mouse button pressed down.

Synchronize cursors in signal windows

By selecting the symbol  in the toolbar, the cursors in all signal windows are displayed synchronously in time. If the cursor position is changed in one signal window, the cursor is also set to the corresponding time in the other signal windows of the signal file.

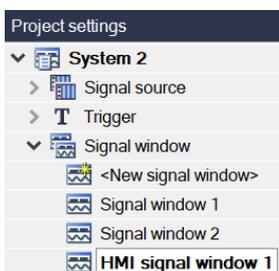
Change zoom



With the help of the zoom element you change the size of the signals in the text signal window. Change the zoom with  and  or by moving the slider  with the mouse. Alternatively, you can use the mouse wheel with <Ctrl> key held down.



Edit HMI signal window



Double-click on the signal window whose settings you want to configure. Alternatively, right-click on the signal window and select the *Properties* menu item.

Clicking the button  in the signal window is another way to open the preferences window.

Fig. 4-40 Edit HMI signal window

Give the HMI signal window a meaningful name. Use *Display signal* window to specify whether the window is to be displayed.

Drag all HMI elements that are to be displayed in the signal window into the signal window with the mouse. After releasing, a settings window appears in which you can assign an address to the HMI element. The value of this address represents the element. Colours and limit values can also be set. Elements that are no longer to be displayed in the signal window can be selected by clicking on them and then deleted with .

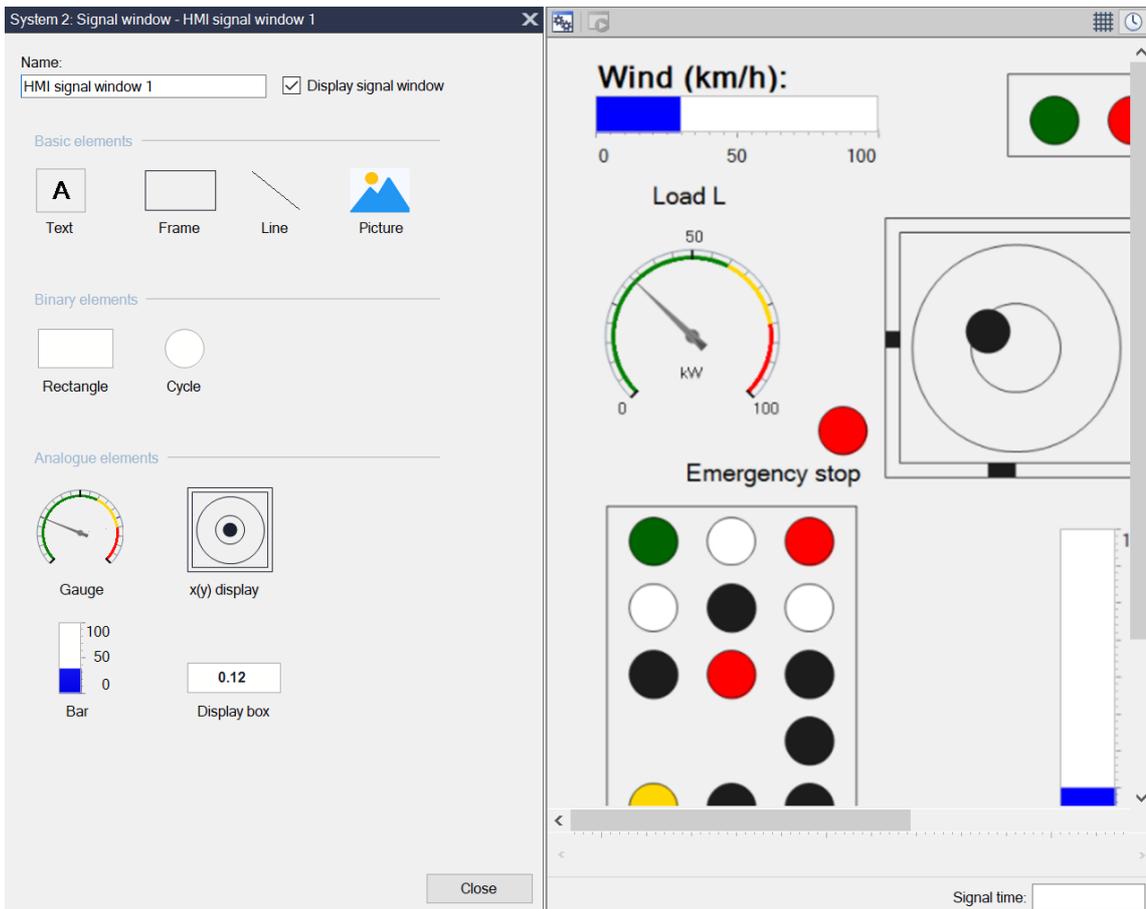


Fig. 4-41 Settings of the HMI signal window

Now adjust the signal window according to your wishes. The following setting options are available:

Absolute / relative time display

In the lower part of the signal window there is a time scale. The *cursor time* text field also displays the current time at the cursor position in the time scale. You can choose between two different display options:

- Relative: Time display relative to the beginning of the signal file
- Absolute: System time (computer time) at the time the signal data is received.

You can change the time display via the menu item *Absolute time display* in the menu *View* or via the icon  in the toolbar of the signal window. The general format of the time display is: hh: mm: ss. xxx, y (hh = hour; mm = minute; ss = second; xxx, y = millisecond).

Switch live / analysis

You can already view and evaluate signal data during recording. Click the icon  in the toolbar of the signal window to pause the live view and switch to analysis view mode. The signal data from the signal file is now displayed and you can analyze and evaluate it. The data recording continues to run in the background. Press  to return to the live display mode.

Edit video signal window

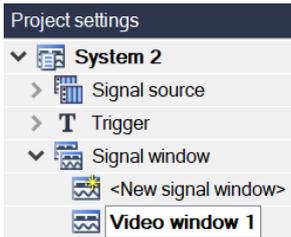


Fig. 4-42 Edit video signal window

Double-click on the video signal window whose settings you want to configure. Alternatively, right-click on the signal window and select the *Properties* menu item.

Clicking the button  in the signal window is another way to open the preferences window.

Give the signal window a meaningful name. Use *Display signal* window to specify whether the window is to be displayed.

Under Camera, select the camera whose images are to be displayed in the video signal window. The camera must be added [as a signal source](#) beforehand.

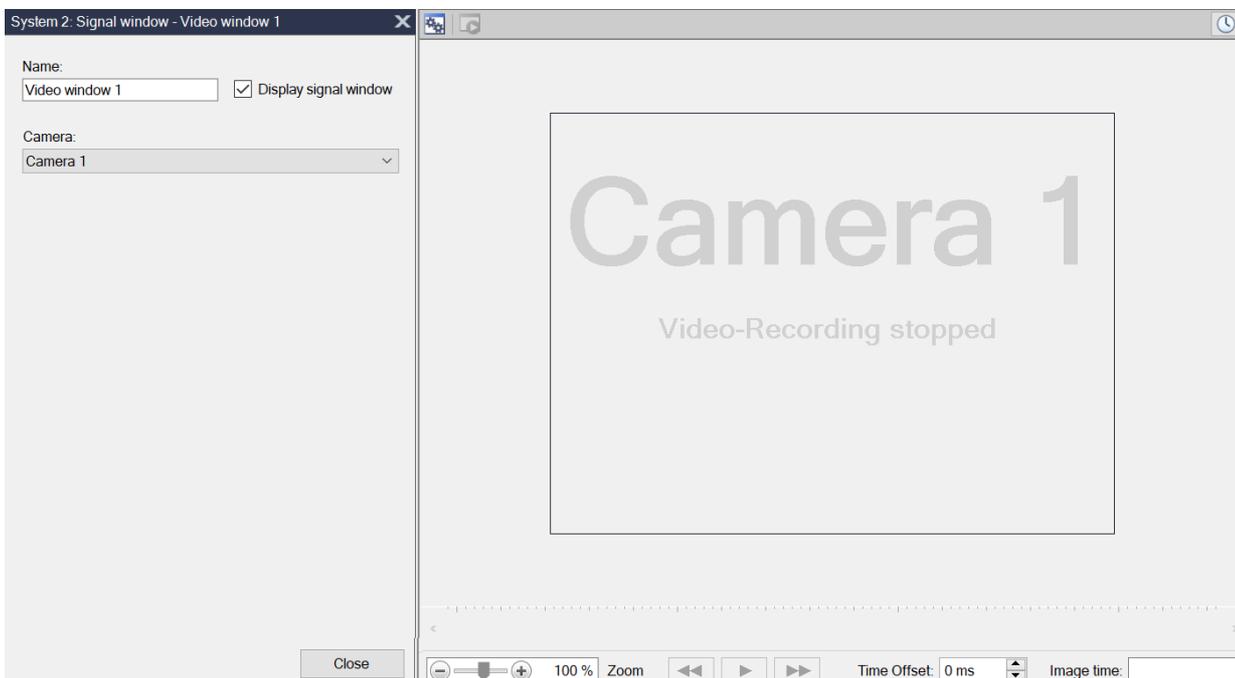


Fig. 4-43 Settings video signal window

Now adjust the video signal window according to your wishes. The following setting options are available:

Absolute / relative time display

In the lower part of the signal window there is a time scale. The *cursor time* text field also displays the current time at the cursor position in the time scale. You can choose between two different display options:

- Relative: Time display relative to the beginning of the signal file
- Absolute: System time (computer time) at the time the signal data is received.

You can change the time display via the menu item *Absolute time display* in the menu *View* or via the icon  in the toolbar of the signal window. The general format of the time display is: hh: mm: ss. xxx, y (hh = hour; mm = minute; ss = second; xxx, y = millisecond).

Switch live / analysis

You can already view and evaluate signal data during recording. Click the icon  in the toolbar of the signal window to pause the live view and switch to analysis view mode. The signal data from the signal file is now displayed and you can analyze and evaluate it. The data recording continues to run in the background. Press  to return to the live display mode

Change time offset

A time offset can be added to the time stamp of the images delivered by the camera. This makes it possible to compensate for any time difference between the camera signals and the recorded PLC signals.

Play back video data

Recorded video data can also be played back later. To do this, click the button  in the video signal window of a recorded video file. With  and  the images can be viewed in fast motion forwards or backwards. You can play the video faster by clicking the buttons several times.

Synchronize cursors in signal windows

By selecting the symbol  in the toolbar, the cursors in all signal windows are displayed synchronously in time. If the cursor position is changed in one signal window, the cursor is also set to the corresponding time in the other signal windows of the signal file.

Change zoom



With the help of the zoom element you change the size of the signals in the text signal window. Change the zoom with  and  or by moving the slider  with the mouse. Alternatively, you can use the mouse wheel with <Ctrl> key held down.



Destination

With the PLC ANALYZER pro 6 per recorded signal data can be stored in different destinations. By default, the data is stored in a [signal file](#).

Create new destination

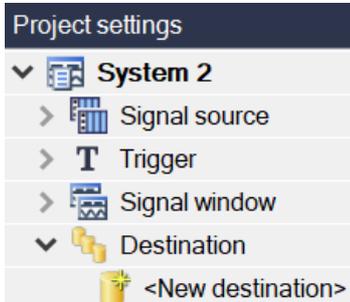


Fig. 4-44 Create new destination

In the project settings, double-click *<New Destination>* in order to insert a new data target into the project.

You can also add previously [copied destination](#). To do this, click on the right mouse button and select the menu item *Insert*.

In the window *Add Destination*, select the desired data destination. Select either a signal file or a CSV text file.

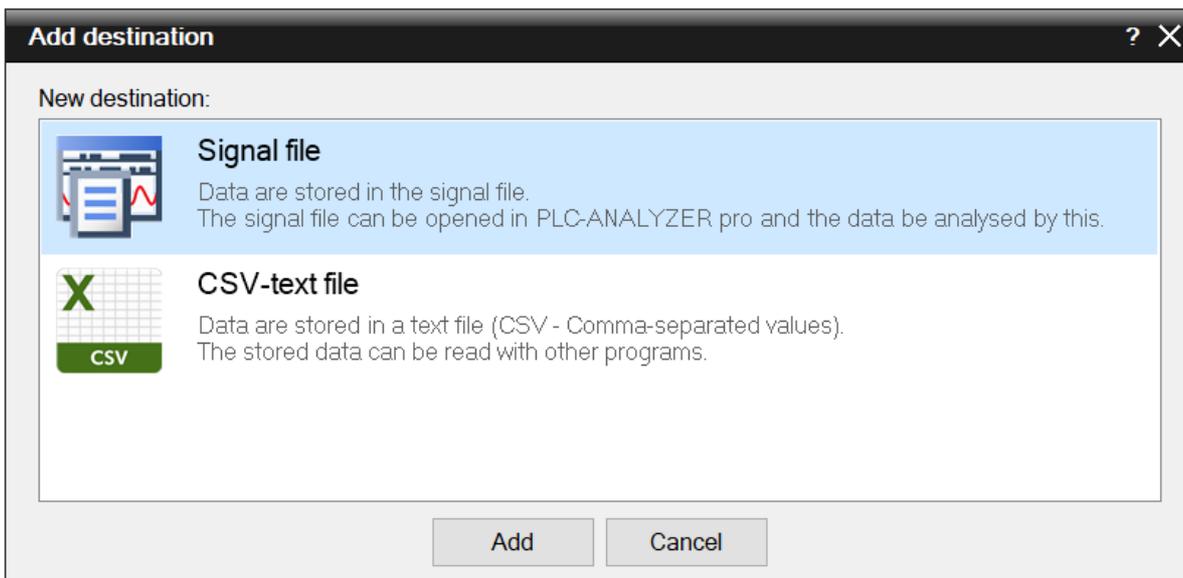


Fig. 4-45 New Destination

Signal file

By default, the signal data is stored in a signal file. In addition to the signal data, the data of the signal windows are also saved. Signal files can be opened by the PLC-ANALYZER pro 6 and the data can be analysed with it.

CSV-Text file

The signal data is stored in a text file (CSV = comma-separated values). The measured values are separated from each other by an adjustable separator. The CSV text file is automatically formatted in such a way that it can be processed with other programs such as Microsoft® Excel®.

With *Add*, the selected destination is transferred to the project and the [Configuration window](#) of the destination is opened.

Edit Destination



Double-click the destination whose settings you want to configure. Alternatively, right-click on the destination and choose *Properties*.

Fig. 4-46 Edit destination

Depending on the selected destination, the corresponding settings window opens in which you can adjust the individual parameters of the destination.

Signal file

By default, signal data is stored in a signal file. In addition to the signal data, the data of the signal windows are also saved. Signal files can be opened by the PLC-ANALYZER pro 6 and the data can be analysed with it.

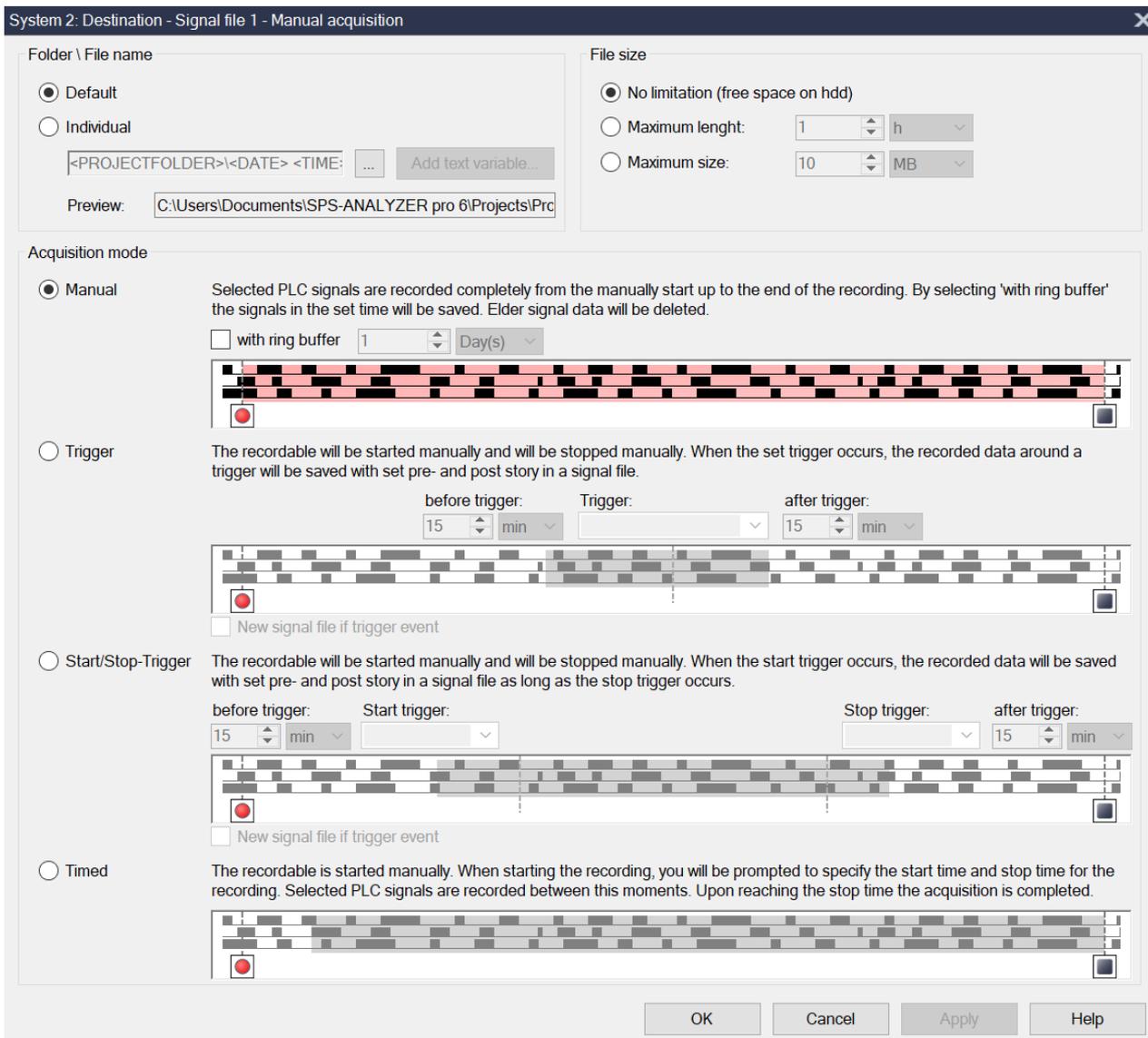


Fig. 4-47 Settings signal file

In the destination settings, specify the *location, file name, and file size*. Also set the *type of data recording*. The settings are described below:

Folder \ File name

Specify the destination's location. Select whether the destination is to be created under a *default* name or at an *individually* specified location. The *default name* composed of date and time and is saved in the project folder. You can also assign a name *individually*. In addition to any text, variables are also available for individual *Folder \ File names*. To add a new variable to the file name, choose [insert variable](#). Variables for date, time and signal values are available.

If a new target is created at start up or during data recording, the variables in the file name are replaced by the corresponding values of these variables.

File size

Under *File size*, set a file size limit for the data destination if necessary. When the set size is reached, the current data destination is saved and a new destination is automatically generated.

Acquisition mode

The signal data recorded by the PLC are stored in the destination, depending on the type of recording. The following table explains the differences:

Acquisition mode	Explanation
Manual	<p>The selected PLC signals are continuously recorded from manual start to the manual end of acquisition.</p> <p>By selecting the control field with <i>ring memory</i>, you activate the ring memory functionality. In this case, the signal data are stored in the set period of time. Signal data older than the set ring memory time are deleted.</p>
Trigger	<p>The recording standby is started manually and ended manually. When the set <i>trigger</i> occurs, the measurement data around this trigger are stored in the target with the set pre- and post history.</p> <p>By activating <i>new signal file on trigger event</i>, a new signal file is generated for every trigger that occurs.</p>
Start-/Stop-Trigger	<p>The recording standby is started manually and ended manually. When the set <i>start trigger</i> occurs, the measurement data are stored in the destination until the set <i>stop trigger</i> with the set previous and subsequent history occurs.</p> <p>By activating <i>new signal file on trigger event</i>, a new signal file is generated for every trigger that occurs.</p>
Timed	<p>The recording standby is started manually. When you start recording, the system prompts you to specify the <i>start time</i> and <i>stop time</i> for recording. The selected PLC signals are stored between these times. When the stop time is reached, the recording is terminated.</p>

Table 4-3 Various acquisition modes

CSV-Text file

In a CSV text file, the measured values are separated from each other by an adjustable separator. The CSV text file is automatically formatted in such a way that it can be processed with other programs such as Microsoft® Excel®.

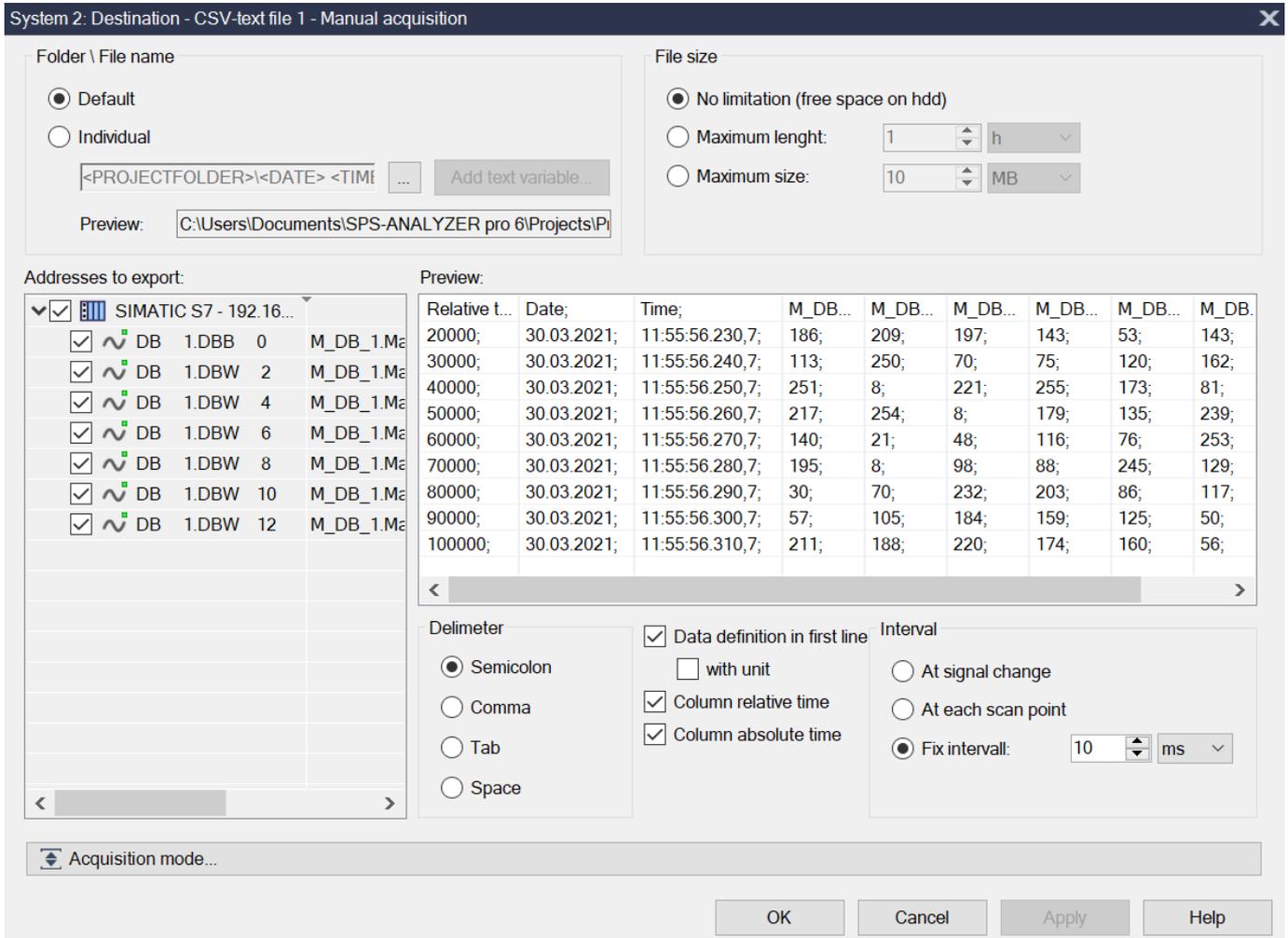


Fig. 4-48 Settings CSV-Text file

In the destination settings, specify the *location*, *file name*, and *file size*. Also set the *type of data acquisition*. For details, see [Settings for the signal file](#).

Folder \ File name

Specify the destination's location. Select whether the destination is to be created under a *default* name or at an *individually* specified location. The *default name* composed of date and time and is saved in the project folder. You can also assign a name *individually*. In addition to any text, variables are also available for individual *Folder \ File names*. To add a new variable to the file name, choose [insert variable](#). Variables for date, time and signal values are available.

If a new target is created at start up or during data recording, the variables in the file name are replaced by the corresponding values of these variables.

File size

Under file size, set a limit on the file size of the data destination. If the set size is reached, the current data destination is saved and a new destination is automatically generated.

In the list *Addresses to be exported*, you specify which signal values are to be saved in the CSV text file. The measured values of all addresses selected here are stored during the acquisition. The measured values are separated from each other by the set *separator*.

If you activate *data definition in the first row*, the columns are given headings. With export units, the corresponding units are saved in addition to the measured values. Specify whether you want to save a column for the *relative time* and for the *absolute time*.

With *Interval* you can set the time interval between the stored signals. If you only want to save the signal changes, choose *When Signal Changes*. This makes the file smaller. When selecting *With each scanning point*, data is exported at all scanning points.

The *Preview list* shows you how the settings affect the file to be created.

Video file

The image data of a video track is stored in a video file.

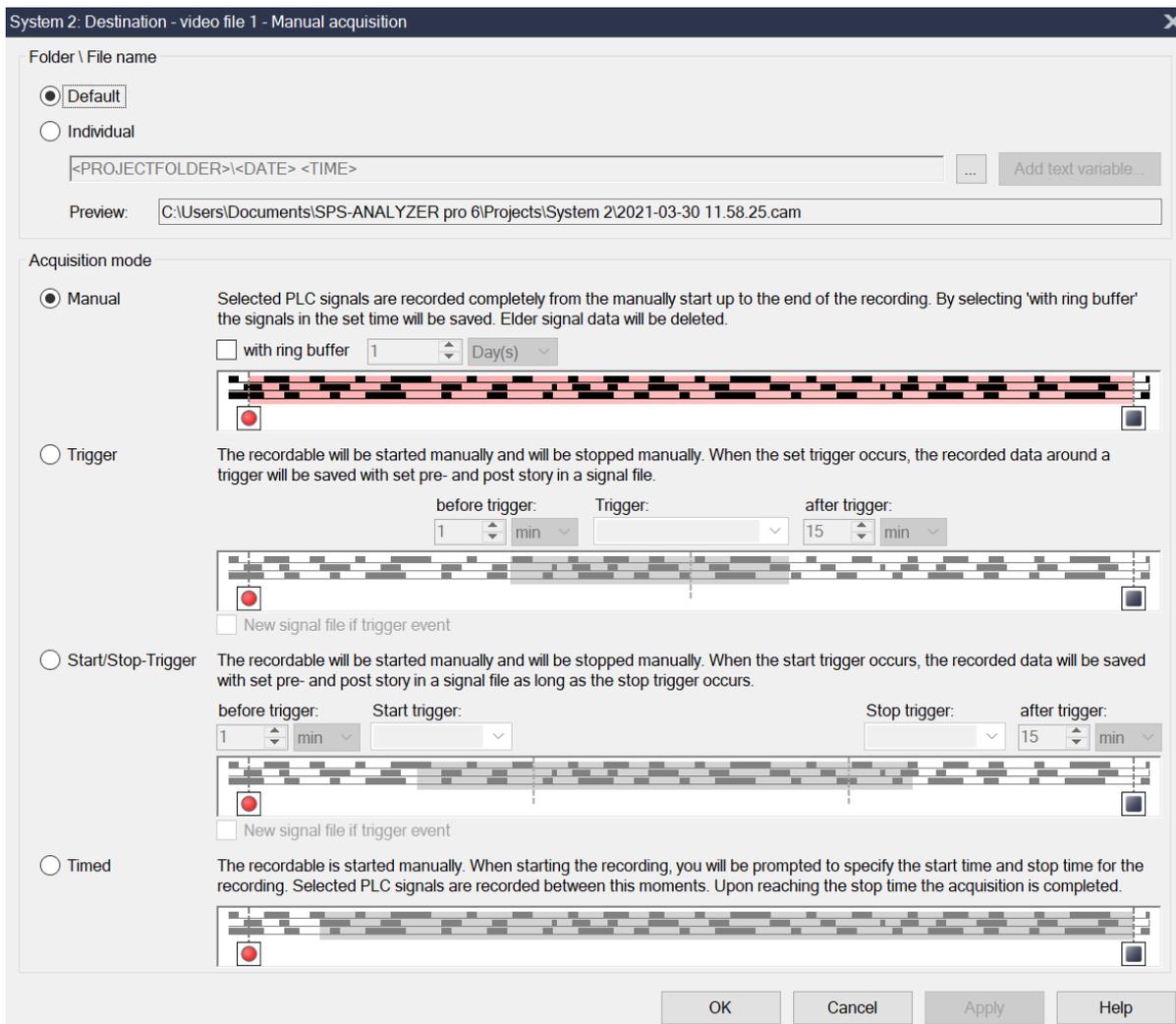


Fig. 4-49 Settings video file

In the destination settings, specify the *location* and the *file name*. Also set the *type of data acquisition*. For details, see [Settings for the signal file](#).

Folder \ File name

Specify the destination's location. Select whether the destination is to be created under a *default* name or at an *individually* specified location. The *default name* composed of date and time and is saved in the project folder. You can also assign a name *individually*. In addition to any text, variables are also available for individual *Folder \ File names*. To add a new variable to the file name, choose [insert variable](#). Variables for date, time and signal values are available.

If a new target is created at start up or during data recording, the variables in the file name are replaced by the corresponding values of these variables.

Acquisition mode

The signal data recorded by the PLC are stored in the destination, depending on the type of recording. The following table explains the differences:

Acquisition mode	Explanation
Manual	<p>The selected PLC signals are continuously recorded from manual start to the manual end of acquisition.</p> <p>By selecting the control field with <i>ring memory</i>, you activate the ring memory functionality. In this case, the signal data are stored in the set period of time. Signal data older than the set ring memory time are deleted.</p>
Trigger	<p>The recording standby is started manually and ended manually. When the set <i>trigger</i> occurs, the measurement data around this trigger are stored in the target with the set pre- and post history.</p> <p>By activating <i>new signal file on trigger event</i>, a new signal file is generated for every trigger that occurs.</p>
Start-/Stop-Trigger	<p>The recording standby is started manually and ended manually. When the set <i>start trigger</i> occurs, the measurement data are stored in the destination until the set <i>stop trigger</i> with the set previous and subsequent history occurs.</p> <p>By activating <i>new signal file on trigger event</i>, a new signal file is generated for every trigger that occurs.</p>
Timed	<p>The recording standby is started manually. When you start recording, the system prompts you to specify the <i>start time</i> and <i>stop time</i> for recording. The selected PLC signals are stored between these times. When the stop time is reached, the recording is terminated.</p>

Table 4-4 Various acquisition modes

Insert variable

The names of the signal files generated during recording can also contain variables. When the signal file is created, the variables in the file name are replaced by the values of the variables valid at that time. You define the file names in the [signal file settings](#).

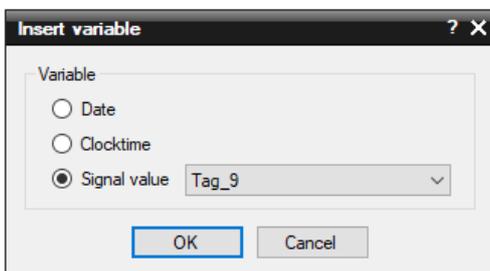


Fig. 4-50 Insert variable

Select a *variable* that you want to insert. The following variables are available:

- Date Inserts the current date in the format YYYY-MM-DD (e. g. 2019-01-21).
- Time Inserts the current time in HH.MM.SS format (e. g. 12.37.52).
- Signal value Inserts the signal value of the selected address

Confirm your selection with *OK*.

Start of recording

Once you have correctly selected all the recording parameters, the signal windows you have defined will appear on the screen. Save your project via *Save Project* in the menu *File* or with the toolbar  so that you can use it again later.

Start recording via the icon  in the toolbar or press the <F5> key. The PLC-ANALYZER pro 6 first checks the communication with the PLC and determines special features of the connected PLC or its CPU.

+

NOTE

If the message "Error during connection setup" appears, please check the settings of the [signal source](#) and the hardware coupling to the PLC.

After a connection has been established successfully, the PLC ANALYZER pro 6 now records the selected PLC signal data and generates one or more signal files, depending on the [set destinations](#).

If you have [signal window](#), the measured values are displayed continuously according to the selected [settings](#).

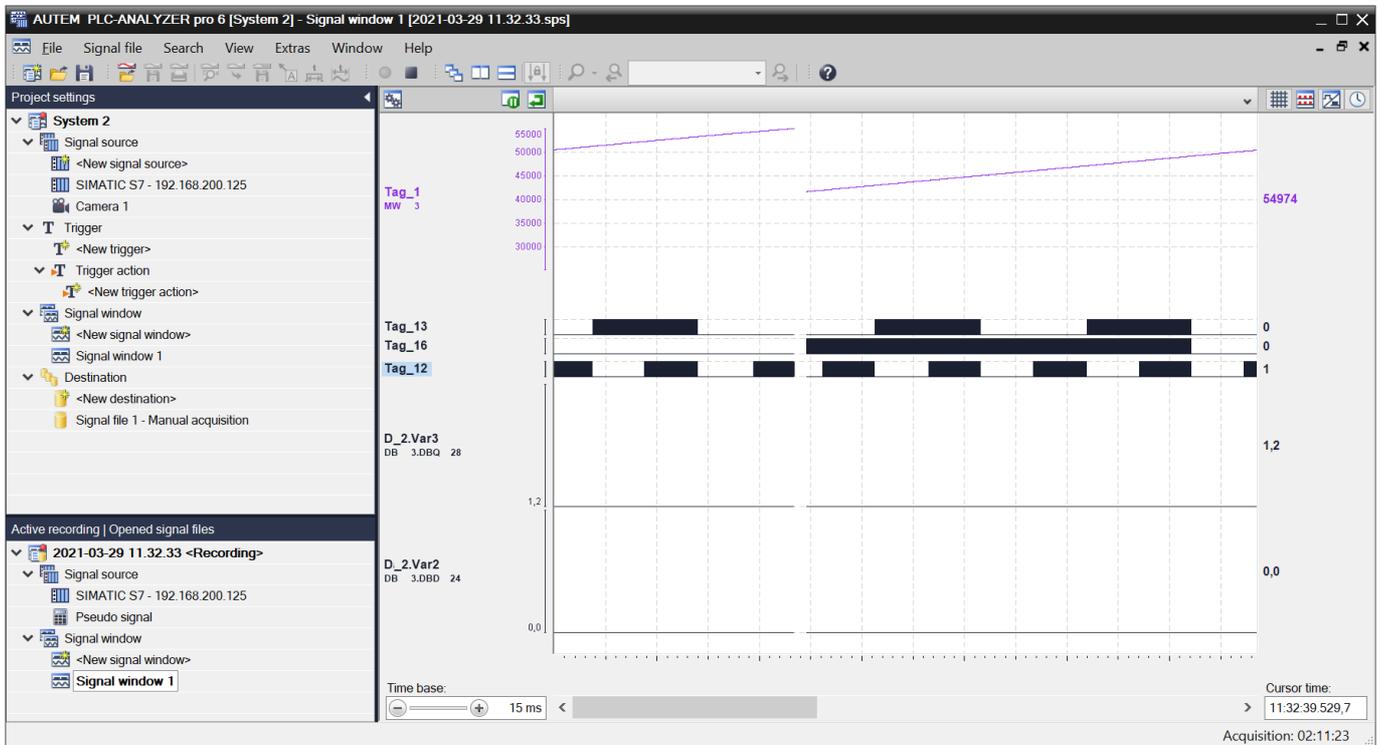


Fig. 4-51 Signal recording

+

NOTE

An analog signal is displayed in grey, if the signal falls outside the range of the scaling limits. If autoscaling is selected, scaling is constantly rescaled during recording.

The signal colour also changes to grey in the event of communication faults. In this case, the instantaneous values are not displayed.

The name of a signal file depends on the selected [destination settings](#).

The recording is terminated by selecting the symbol  in the toolbar or by pressing the <F6> key. The last generated signal file is automatically displayed after the acquisition is finished.

Project-Info

To document a measurement project, the PLC ANALYZER pro 6 can generate a complete overview of all project settings.

Double-click on the project name in the tree *Project Settings* or right-click here on it and select *Info*. The *Info* window displays all project data. You also assign a *description* to the project here.

Selecting the *Save* button saves the project settings to a text file. Choose *Print* to print the project settings.

Project: Info

Name: System2 Path: C:\Users\Documents\SPS-ANALYZER pro 6\Projects\Project126

Description:
Problems with Machine in System 2
Siemens S7 - 192.168.200.125
12.04.2019

PLC driver:
PLC_1 - 192.168.200.125

Addresses:

Absolute	Symbol	Comment	Driver
DB 1.DBX 0.0	DB1.MachineInfo.Running	Is machine running now?	PLC_1 - 192.168.200.125
DB 1.DBW 3	DB1.MachineInfo.V1	Speed of MV_33	PLC_1 - 192.168.200.125
DB 1.DBW 5	DB1.MachineInfo.V2	Speed of MV_34	PLC_1 - 192.168.200.125
DB 1.DBW 16	DB1.MachineInfo.MainEngine.AxisInfo[0].x	x-axis	PLC_1 - 192.168.200.125
DB 1.DBW 18	DB1.MachineInfo.MainEngine.AxisInfo[0].y	y-axis	PLC_1 - 192.168.200.125
DB 1.DBW 20	DB1.MachineInfo.MainEngine.AxisInfo[0].z	z-axis	PLC_1 - 192.168.200.125

Buttons: Save, Print..., OK, Cancel, Apply, Help

Fig. 4-52 Project info

Signal display and analysis

This chapter describes the analysis of the recorded signal data. For a detailed analysis, the PLC-ANALYZER pro 6 provides you with various tools, such as [time measurement](#), extensive [search functions](#) and [signal statistics](#).

Open signal file

After termination of a recording, the PLC-ANALYZER pro 6 always automatically displays the last generated signal file.

Manually open a signal file in the menu *File*, either via the history list (contains the eight most recently loaded signal files) or via *Open signal file....* Alternatively, click the icon  in the toolbar.

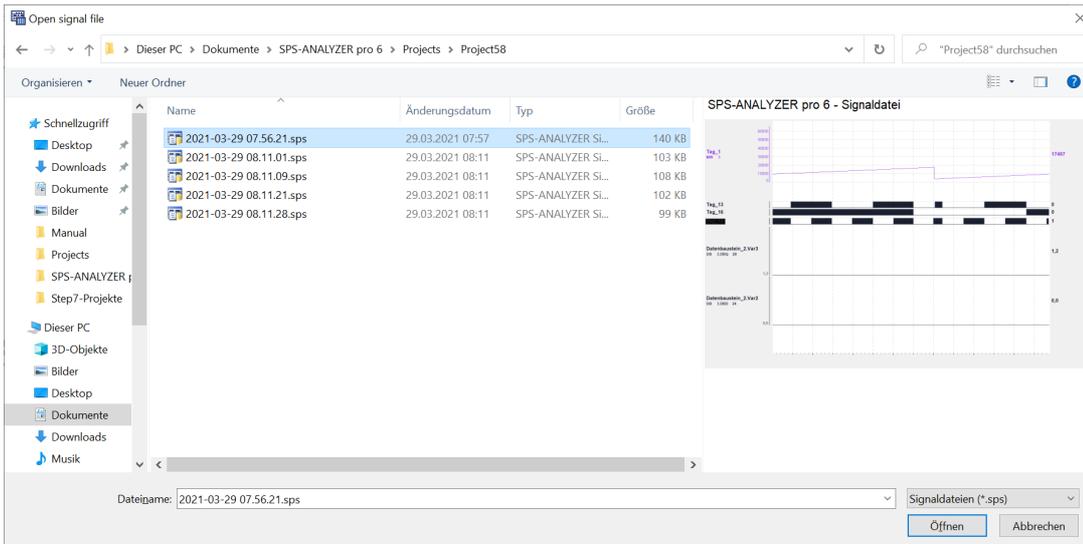
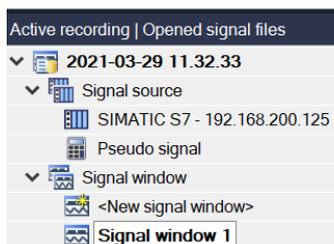


Fig. 5-1 Open a signal file

The file selection window lists the signal files in chronological order of their recording. Information of the selected signal file is displayed in the preview window of the selection window.

To open a signal file, select it and press the *Open* button.



After opening the signal file, all objects of the opened signal file are displayed in a tree structure in the left area of the PLC ANALYZER pro 6. All currently open signal files are listed under *Active Recording / Opened Signal Files*. All settings of the signal files can be made here.

Fig. 5-2 Signal file settings

Save signal file

To save modified signal files, click on the symbol  in the toolbar or select the menu *File* the item *Save signal file* or *Save signal file as*.

Regardless of this, depending on the [recording mode](#) selected, signal files are automatically saved when a data recording is terminated.

Save a signal file section

You can also use the PLC ANALYZER pro to save segments of a signal file. Simply select the desired section (see section [Mark section](#)) and select *Section / Save* from the menu *Signal File*. Enter a name for the portal and choose *Save*.

Adjusting the signal display

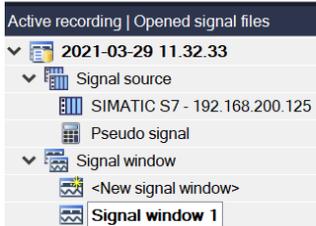


Fig. 5-3 Signal file settings

All open signal files are listed under *Active Recording / Opened Signal Files*. All settings of the signal files can be made here.

The individual objects are edited here in the same way as with the corresponding [Project settings](#).

Signal selection

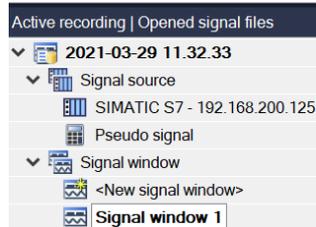


Fig. 5-4 Edit signal window

Under *Active Recording / Open Signal Files*, double-click the signal window whose settings you want to configure. Alternatively, right-click on the signal window and select *Properties*.

Clicking the button  in the signal window is another way to open the preferences window.

Now select addresses that should be displayed in the signal window or removed from it. Proceed as described in [Edit signal window](#).

Signal scaling

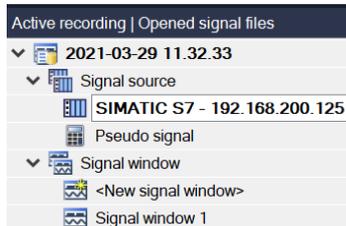


Fig. 5-5 Edit signal source

Double-click the signal source whose settings you want to configure. Alternatively, right-click on the signal source and select the menu item *Properties*.

In the settings window, change the scaling of the signals (see [Scaling of register values](#)).

Change symbols and comments of signals

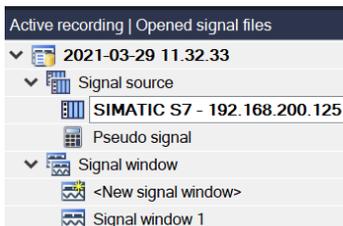


Fig. 5-6 Edit signal source

Double-click the signal source whose settings you want to configure.

Alternatively, right-click on the signal source and select the menu item *Properties*.

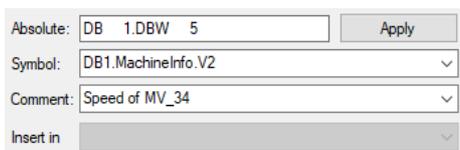


Fig. 5-7 Change symbols and comments

In the list *Addresses for recording*, select the address whose symbolic identifier or comment you want to change.

The absolute address of the selected signal appears in the text field *Absolute*. Enter the desired *symbol* and the *comment* of the address. Choose *Apply* to accept the change and copy it to the list *Addresses for recording*.

Moving signale

In the signal window, select all signals whose position is to be changed. Simply click on the corresponding signal. By holding down the <Ctrl> or <Shift> keys and selecting several signals simultaneously with the mouse, several signals can be selected. Then press the left mouse button on a selected signal and move the signals with the left mouse button pressed down.

The height of a selected analog signal can be changed by changing the position of the upper or lower end of the scale of the signal with the mouse. The height of a bit signal cannot be changed. The height always corresponds to the standard binary signal level set in the [settings](#).

Insert note

Click the right mouse button in the signal window of the signal file and select the menu item *Note / New*. Alternatively, select this menu item from the signal file menu.

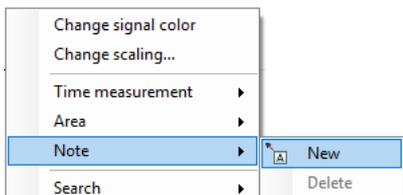


Fig. 5-8 Menu "New note"

Click on the position where you want the note to appear. Enter any text in the text field that appears. With <ENTER> you go to the next line. Click with the mouse next to the text field or <ESC> to accept the note.

Existing notes can be moved as required. To do this, drag it to the new position while holding down the left mouse button. The position of the arrowhead of the note can also be changed this way.

You can change a note by double-clicking on it.

If you want to delete a note, press the right mouse button on it and select the menu option *Note / Delete*.

Change signal color

In the address field on the left side of the signal window, select all signals whose colour you want to change. Simply click on the corresponding signal. By holding down the <Ctrl> or <Shift> keys and selecting several signals simultaneously with the mouse, several signals can be selected. Then press the right mouse button on a selected signal and select the menu item *Change signal color* and select the desired signal color.

Mark Section

For certain functions of the PLC ANALYZER pro 6 it is necessary to mark a section of the signal file. This is necessary, for example, if you want to save, export, print, zoom or statistically analyze a certain section of the signal file.

To do this, position the [signal cursor](#) at the beginning of the section to be marked. Click the right mouse button in the signal window of the signal file and select *Section / Select*. Alternatively, select this menu item from the signal file menu. Now move the mouse pointer to the end of the desired section. The marked area is displayed in blue. Use the left mouse button to end the marking process.

You can also hold down the shift key (Shift \hat{u}) and move the signal cursor with the mouse.

Press the right mouse button in the marked area to select further actions under the menu item *Section*.

Time information

The lower part of the $y(t)$ -signal window contains important information and setting options for the time display.



Fig. 5-10 Time information

The *cursor time* text field displays the current time at the position of the signal cursor. The time scale also shows the time at the respective position. Depending on the [settings](#), either the relative time or the absolute time is displayed here.

With the time base you set the time per screen pixel, i. e. the zoom of the signal data over time. By changing the time base, the display of the signal data is stretched or compressed over time.

Change the time base with \oplus and \ominus or by moving the slider \blacksquare with the mouse. Alternatively, you can use the mouse wheel with <Ctrl>-key held down.

Signal cursor

The vertical green line (blinking) is the signal cursor. All values shown in the display fields - e. g. instantaneous value and time - always refer to the current cursor position. Simply navigate the signal cursor to an interesting position in the signal path and read the measured values valid there.

The signal cursor can be moved elegantly by mouse or keyboard. For example, you can grab the cursor with the mouse and drag it to any position. When you reach the right or left limit of the signal display window with the cursor, the displayed signal file section scrolls slowly forwards or backwards.

The following table explains the keyboard operation of the cursor.

Key	Function
\Rightarrow	Cursor forward stepwise
\Leftarrow	Cursor backward stepwise
CTRL + \Rightarrow	Picture in steps of 10 pixels forward (cursor stops)
CTRL + \Leftarrow	Picture in steps of 10 pixels backward (cursor stops)
Page \uparrow	Screen page back
Page \downarrow	Screen page forward
Pos1	Cursor at the beginning of the screen
End	Cursor at the end of the screen
CTRL + Pos1	Cursor at the beginning of the signal file
CTRL + End	Cursor at the end of the signal file

Table 5.1 Operation via keyboard

Time measurement

The PLC-ANALYZER pro 6 has the capability of time difference measurement and bit measurement for the temporal measurement of signals.

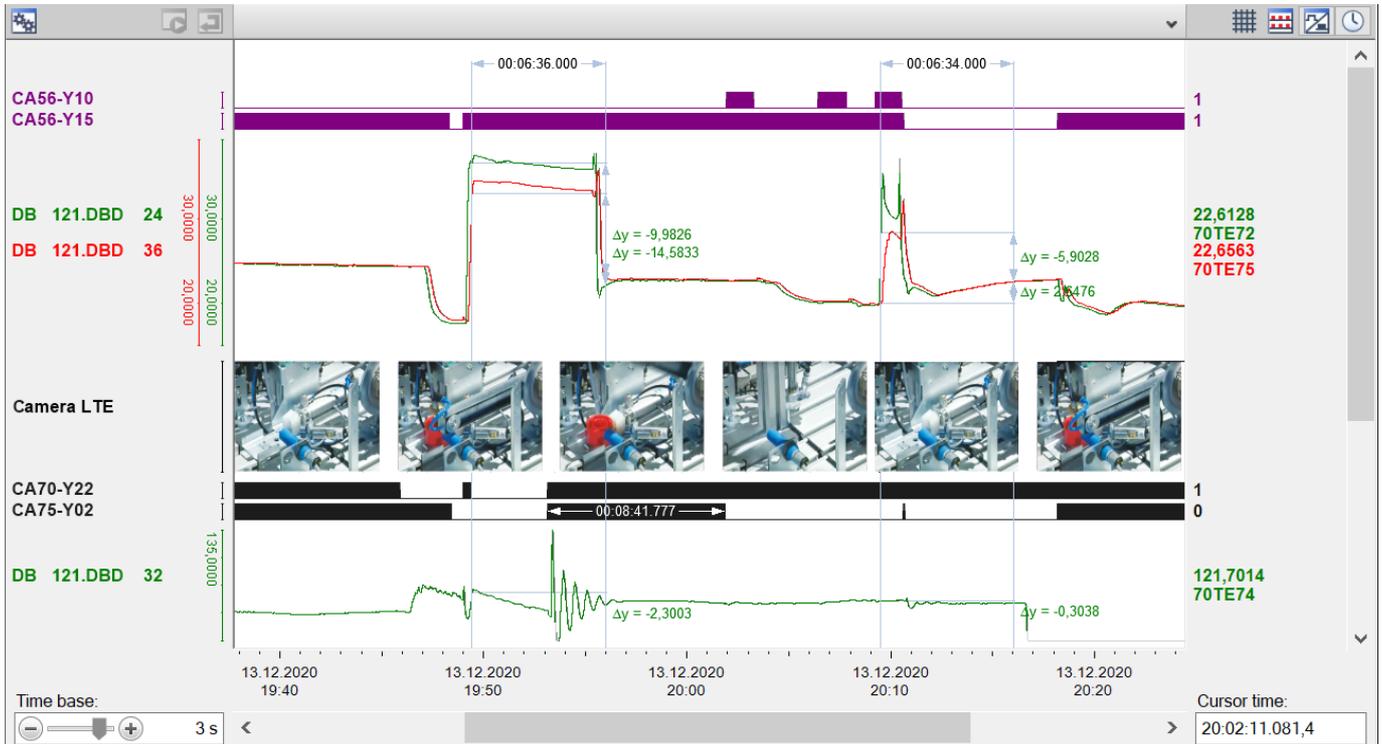


Fig. 5-11 Time difference measurement

Time difference measurement

You can use the time difference measurement to measure the time difference between any two positions. To do this, first position the signal cursor on a reference point. Activate the time difference measurement by selecting menu option *Time measurement / Time difference* in the menu *Signal File* or select the icon  in the toolbar. The reference point is now marked by a reference cursor. Now click with the mouse on the second measuring point. In addition to the time difference, the differences of the signal value are also displayed for analog values.

To change the measuring points afterwards, drag them to the desired new position by keeping the left mouse button pressed down.

Bit measurement

Bit measurement is an elegant way to measure bit signals. The time between the two edges of the bit signal is measured. To do this, first activate the ready for measurement mode by selecting the menu option *Time measurement / Bit measurement on* in the *Signal File* menu or by selecting the symbol  in the toolbar.

Now click on the bit signal whose length is to be measured. The time difference is now automatically entered into the bit signal. You can now measure bit signals until the ready state is terminated by selecting the menu item *Time measurement / Bit measurement off* in the *Signal file* menu or by selecting the symbol  in the toolbar.

To remove a time measurement, highlight it with the mouse and press <Delete> or select *Time measurement / Delete* from the *Signal File* menu.

Search funktions

The PLC ANALYZER pro 6 supports the search for specific signal values, triggers, times or notes within a signal file. Activate the search functions via the *Search* menu, by pressing the right mouse button in the display area of the signal window or via the toolbar:

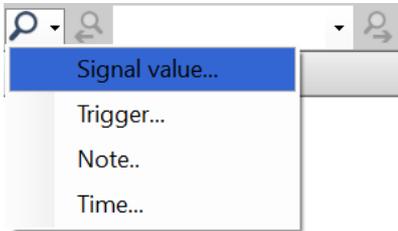


Fig. 5-12 Define new search

Searches that have already been performed can be selected in the search selection in the toolbar and executed again. Choose an existing search and start the search with or .



Fig. 5-13 Start existing search

Search signal value

To find certain signal values, use the signal value search. In the Search menu, select the *Signal value...* menu item to access the *Search signals values* window. Alternatively, you can access the *Search signals values* window via the [toolbar](#) or via the context menu (right mouse button in the Signal window).

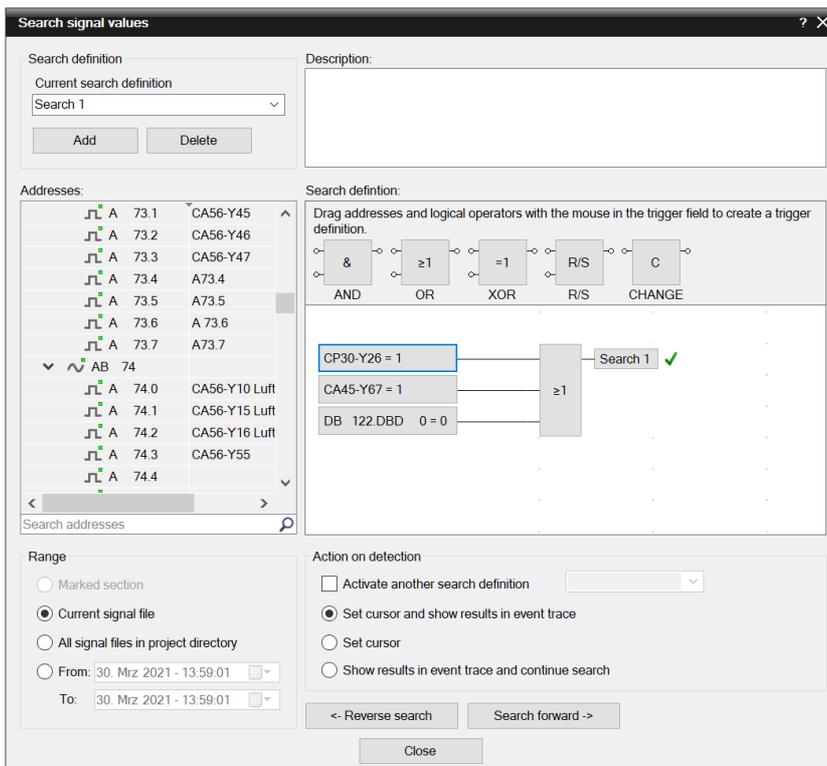


Fig. 5-14 Search signal values

Under *Current search definition*, enter a meaningful name for the search definition. You can also select other searches here. With *Add* and *Remove* you can add or delete further search definitions. Assign any *description* to the search definition.

Now define the *search definition*. For this purpose, all signals from the list *Addresses* and the logical operators

AND, OR, XOR, R/S and CHANGE are available. To define the search definition, drag *addresses* or logical operators into the definition field with the mouse. Then connect the inputs and outputs of the modules with the mouse to a search definition. Creating the search definition is identical to creating a trigger condition. For a detailed description of the available logical operators and their properties, see [Edit Trigger](#).

Specify the *Range* in which the search is to be performed:

Marked section

The [marked section](#) of the signal file is searched.

Current signal file

The entire signal file is searched from the current cursor position. If the end of the signal file is reached, the search continues at the beginning of the file until the start time of the search.

All signal files in the project directory

Starting from the current cursor position, all signal files in the project directory are searched chronologically. If the end of the last signal file is reached, the search continues at the beginning of the first signal file.

Any period of time

Select any search period. All signal files located in the project directory and within the selected time period are searched in chronological order. If the end of the last signal file is reached in this time period, the search continues at the beginning of the first signal file during this time period.

Under *Action on find* you determine what should happen if the search is successful:

Set cursor and display hits in event track

The search is started with <- *Search backwards* or *Forward search*->. In the case of a find, the search is stopped and the signal cursor is positioned on the location found. The hit is displayed in the [event track](#). The search can be continued with <- *Search backwards* and *Search forwards*->.

Set cursor

The search is started with <- *Search backwards* or *Forward search*->. In the case of a find, the search is stopped and the signal cursor is positioned on the location found. The search can be continued with <- *Search backwards* and *Search forwards*->.

Show hit in event track and continue search

The search is started with <- *Search backwards* or *Forward search*->. The set area is then searched from the cursor position onwards. Finds are displayed in the [event track](#).

Activate other search definition

In the case of a find, another previously defined search definition is activated. From this point on, the search will continue with the search definition selected here.

The search function remains active even after *closing* the window and can be restarted with  or  via the [Toolbar](#).

Search trigger

Use the trigger search to find specific triggers. Select the menu item *Trigger...* from the menu *Search* to open the *Search Trigger* window. Alternatively, you can access the window via the [toolbar](#).

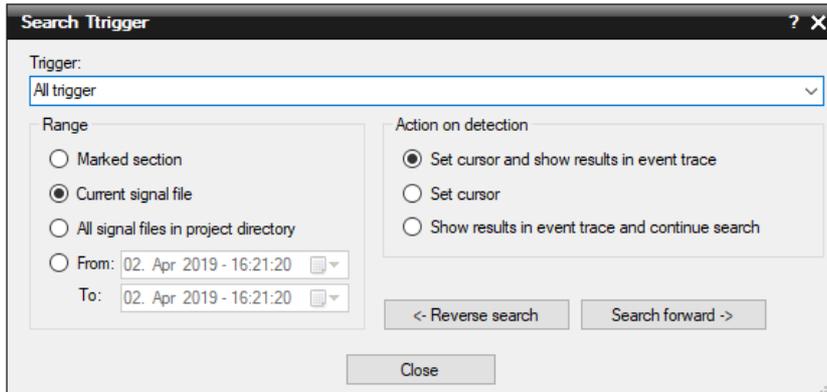


Fig. 5-15 Search trigger

In the text field *Trigger*, set the name of the trigger you want to search for. If "All trigger" is specified, the system searches for all trigger that occur.

Specify the *Range* in which the search is to be performed:

Marked section

The [marked section](#) of the signal file is searched.

Current signal file

The entire signal file is searched from the current cursor position. If the end of the signal file is reached, the search continues at the beginning of the file until the start time of the search.

All signal files in the project directory

Starting from the current cursor position, all signal files in the project directory are searched chronologically. If the end of the last signal file is reached, the search continues at the beginning of the first signal file.

Any period of time

Select any search period. All signal files located in the project directory and within the selected time period are searched in chronological order. If the end of the last signal file is reached in this time period, the search continues at the beginning of the first signal file during this time period.

Under *Action on find* you determine what should happen if the search is successful:

Set cursor and display hits in event track

The search is started with *<- Search backwards* or *Forward search->*. In the case of a find, the search is stopped and the signal cursor is positioned on the location found. The hit is displayed in the [event track](#). The search can be continued with *<- Search backwards* and *Search forwards->*.

Set cursor

The search is started with *<- Search backwards* or *Forward search->*. In the case of a find, the search is stopped and the signal cursor is positioned on the location found. The search can be continued with *<- Search backwards* and *Search forwards->*.

Show hit in event track and continue search

The search is started with *<- Search backwards* or *Forward search->*. The set area is then searched from the cursor position onwards. Finds are displayed in the [event track](#).

The search function remains active even after *closing* the window and can be restarted with  or  via the [Toolbar](#).

Search note

Use the note search to find specific notes. In the menu *Search*, select menu item *Note...* to open the *Search Note* window. Alternatively, you can access the window via the [toolbar](#).

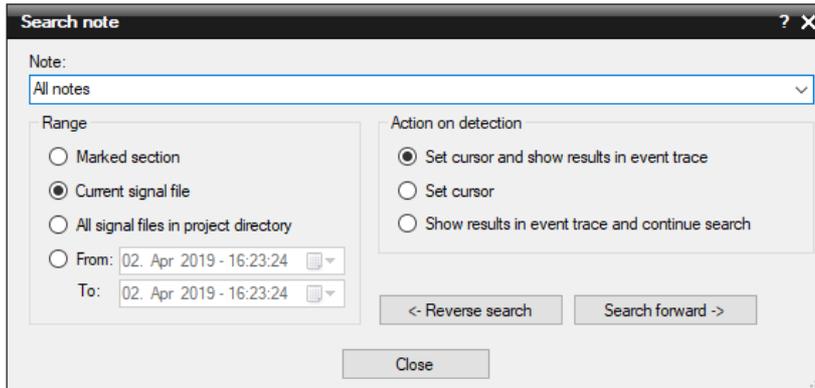


Fig. 5-16 Search note

First select the *note* you want to search for. The setting "All notes" searches for all notes that occur.

Specify the *Range* in which the search is to be performed:

Marked section

The [marked section](#) of the signal file is searched.

Current signal file

The entire signal file is searched from the current cursor position. If the end of the signal file is reached, the search continues at the beginning of the file until the start time of the search.

All signal files in the project directory

Starting from the current cursor position, all signal files in the project directory are searched chronologically. If the end of the last signal file is reached, the search continues at the beginning of the first signal file.

Any period of time

Select any search period. All signal files located in the project directory and within the selected time period are searched in chronological order. If the end of the last signal file is reached in this time period, the search continues at the beginning of the first signal file during this time period.

Under *Action on find* you determine what should happen if the search is successful:

Set cursor and display hits in event track

The search is started with *<- Search backwards* or *Forward search->*. In the case of a find, the search is stopped and the signal cursor is positioned on the location found. The hit is displayed in the [event track](#). The search can be continued with *<- Search backwards* and *Search forwards->*.

Set cursor

The search is started with *<- Search backwards* or *Forward search->*. In the case of a find, the search is stopped and the signal cursor is positioned on the location found. The search can be continued with *<- Search backwards* and *Search forwards->*.

Show hit in event track and continue search

The search is started with *<- Search backwards* or *Forward search->*. The set area is then searched from the cursor position onwards. Finds are displayed in the [event track](#).

The search function remains active even after *closing* the window and can be restarted with  or  via the [Toolbar](#).

Search time

Use the time search to find a specific time. In the menu *Search*, choose the menu option *Time...* to open the *Search Time* window. Alternatively, you can use the [toolbar](#) to access the window.

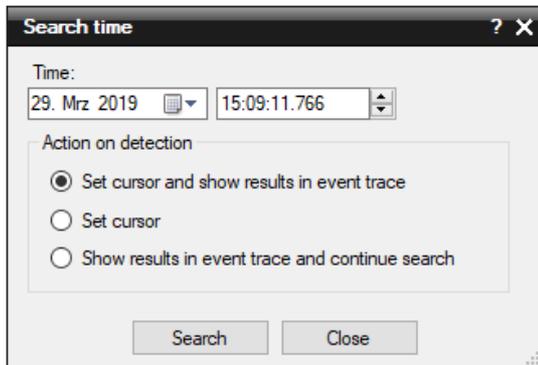


Fig. 5-17 Search Time

Enter the time to be searched for. Alternatively, you can also change the desired time by pressing the spin box fields.

Under *Action on find* you determine what should happen if the search is successful:

Set cursor and display hits in event track

The search is started with *<- Search backwards* or *Forward search->*. In the case of a find, the search is stopped and the signal cursor is positioned on the location found. The hit is displayed in the event track. The search can be continued with *<- Search backwards* and *Search forwards->*.

Set cursor

The search is started with *<- Search backwards* or *Forward search->*. In the case of a find, the search is stopped and the signal cursor is positioned on the location found. The search can be continued with *<- Search backwards* and *Search forwards->*.

Show hit in event track and continue search

The search is started with *<- Search backwards* or *Forward search->*. The set area is then searched from the cursor position onwards. Finds are displayed in the event track.

The search function remains active even after *closing* the window and can be restarted with  or  via the [Toolbar](#).

Compare signal files

The PLC-ANALYZER pro 6 offers you an elegant possibility to compare signal files with each other. This can be used, for example, to determine creeping wear on a system ("Condition Monitoring").

Open the signal files, which you like to compare, one after another. Now choose *Compare* in the signal file menu. The signals of both signal files will be shown superimposed in one signal window. If several signal files are open, you will be prompted to select the signal window to compare.

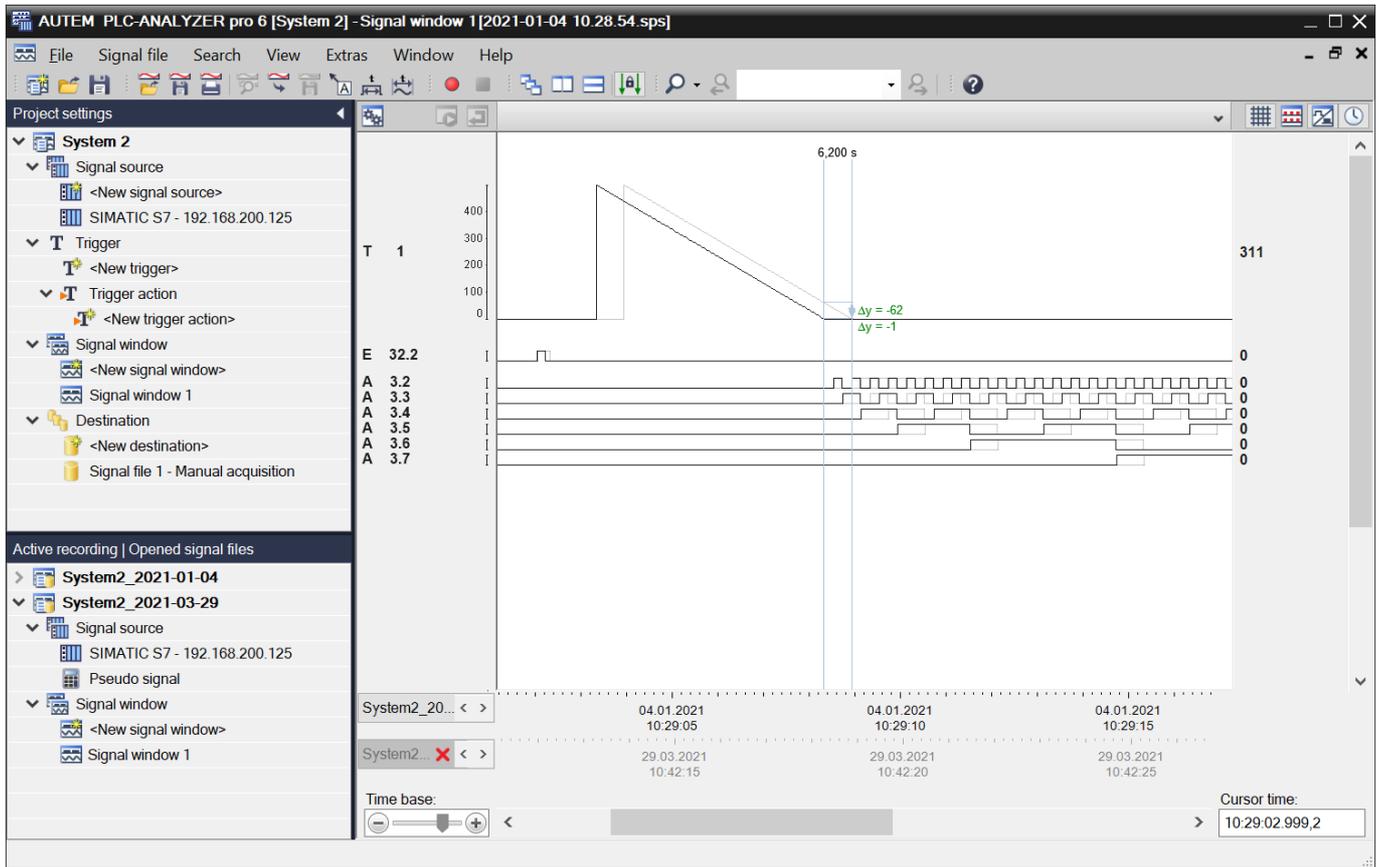


Fig. 5-18 Compare signal files

The signals of the inactive signal file are displayed in grey while the signals of the active signal file appear in the selected colour. All signal file functions, e. g. cursor time, time difference, search functions, refer to the active signal file



For each signal file to be compared, there is a button on the left side of the signal window with which you can activate the respective signal file. You can also use **<** or **>** to move the signals of the signal file horizontally. Alternatively, you can also move the respective scale by keeping the mouse button pressed down. Use **✖** to remove the signal file from the signal window and terminate the comparison.

Merge signal files

The PLC-ANALYZER pro 6 offers you the possibility to combine signal files recorded separately from each other. The content of the *Source signal files* is then correctly fitted into the *New signal file*. For example, the signal files of a project whose [destination settings](#) have limited the signal file size can be reassembled.

To do this, select *Merge signal files* from the *File* menu. In the window *Merge signal file*, select the signal files whose data you want to transfer to the *New signal file*.

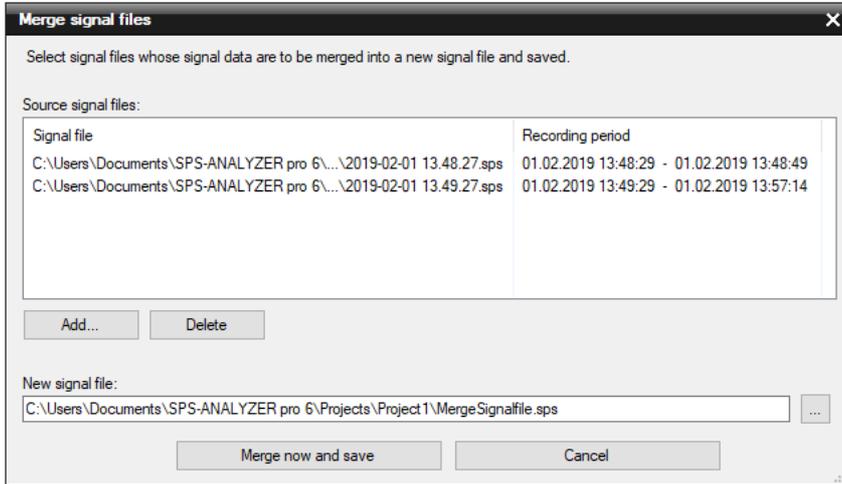


Fig. 5-19 Merge signal files

Press *Merge now and save* to save the data of the selected signal files to *New signal file*.

Event bar

The event bar at the top of the signal window gives you an overview of all events that have occurred during recording. The events are displayed by corresponding symbols in the event bar. If you move the mouse pointer over an icon, further information about this event is displayed and the position of the event is displayed in the signal window.



Fig. 5-20 Event bar in signal window

Click to  to fold down the event list. All events of the signal file are listed in the event list. In addition to the times, the exact descriptions of the respective events are also displayed. Click on  to close the list so that only the event track remains visible.

The following icons are displayed in the event track:

-  Information on data recording, such as the start of recording.
-  Warnings
-  Error during data recording, e. g. interruption of data acquisition, because the PLC no longer responds.
-  Trigger event occurred
-  Search condition found

Signal statistics

The PLC-ANALYZER pro 6 supports the statistical evaluation of signal characteristics. Select *Signal statistics...* in the signal file menu to obtain extensive information on a specific signal.

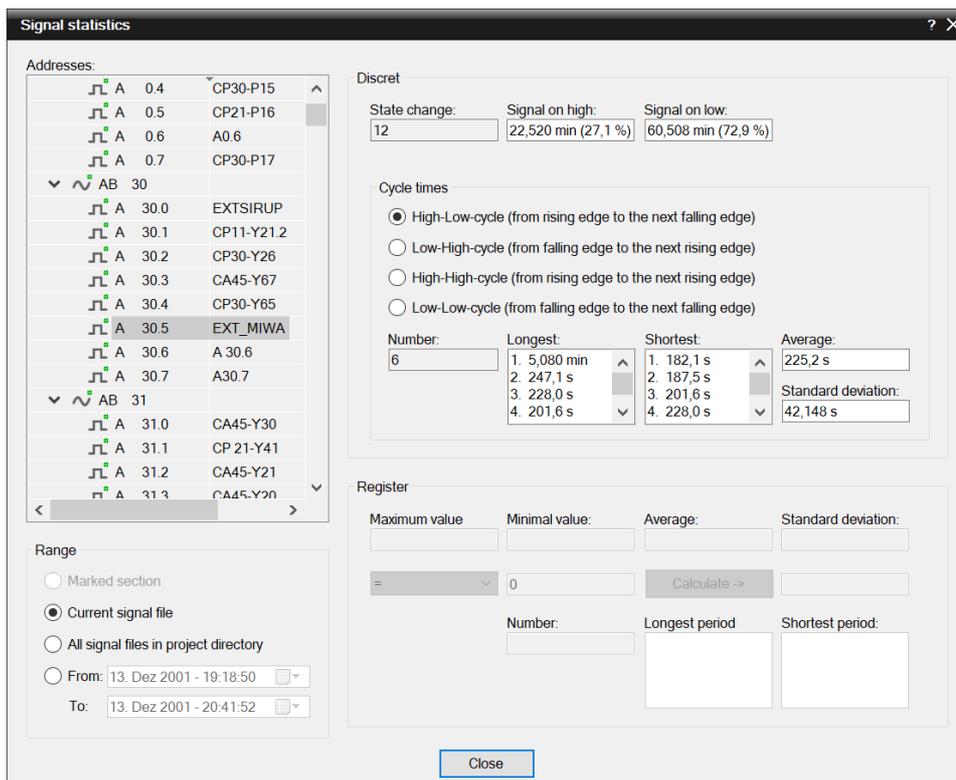


Fig. 5-21 Signal statistics

Under *Addresses*, select the address for displaying statistical information. Set the desired *Range* to which the analysis should refer.

Statistical evaluation distinguishes between discrete signals (bit addresses) and register values (byte, word or double word):

Statistical analysis for discrete signal

State change indicates the number of edge changes in the selected *range*. It also shows how long the *signal* was *on high* or the *signal on low*.

Statistical analysis of selected *cycle types* is displayed under *cycle times*. First set the type of bar:

- High-Low-Cycle - from rising edge to the next falling edge
- Low-High- Cycle - from falling edge to the next rising edge
- High-High- Cycle - from rising edge to the next rising edge
- Low-Low- Cycle - from falling edge to the next falling edge

After selection, the *number* of cycles appearing in the selected range is displayed. The longest and shortest cycle are listed under *longest* and *shortest*. The mean value and *standard deviation* of the cycles are also displayed.

Statistical analysis for register value

When a register value is selected, its *maximum value*, *minimum value*, average value and *standard deviation* are displayed.

Register			
Maximum value	Minimal value:	Average:	Standard deviation:
65533	0	31577	18496
>	15000	Calculate ->	3,391 min (76,1 %)
Number:	Longest period	Shortest period:	
5	53,975 s	1,416 s	

Fig. 5-22 Signal statistics - Register value

You can also calculate the rate of occurrence of a certain signal value. To do this, enter an operand and enter a signal value. Pressing *Calculate* displays the result. The set condition was fulfilled during the time displayed here.

The *number*, the *longest phase* and the *shortest phase* of the fulfilled conditions in the selected period are also displayed.

Print a signal file

You can either print the entire signal files or extracts of the signal files. Via the menu item *Print...* in the *File* menu the printer selection window will be opened.

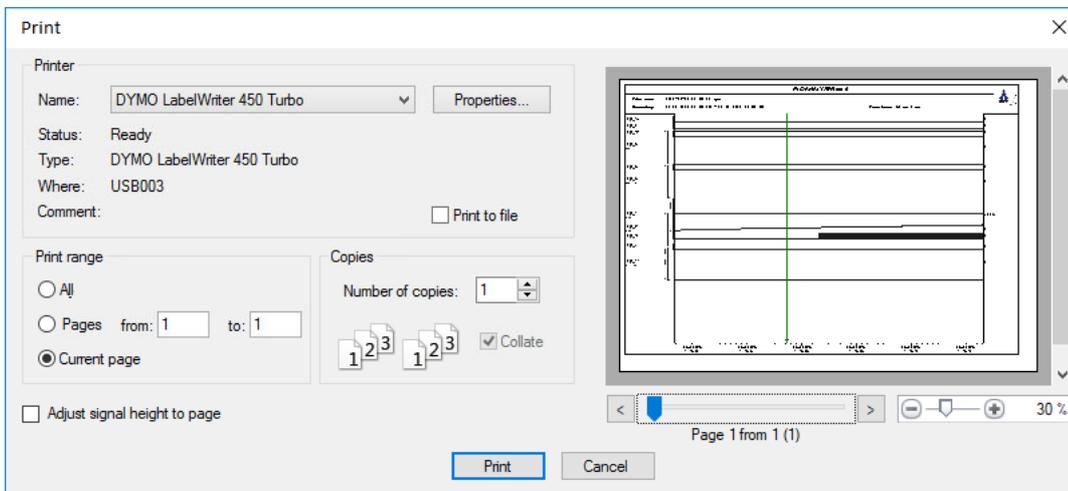


Fig. 5-23 Printer selection window

Under *Printer* select the desired printer. In the *Print range* option field, specify the desired section of the signal file to be printed. Select *All* to print the entire signal file with the time base set in the current signal window. By specifying *pages*, you can print certain pages. If you have previously marked a specific section in the signal file, you can also print only the [selected area](#) of the signal file.

Under *Copies*, specify how often the signal file is to be printed.

By selecting *adjust signal height to page*, the height of the PLC signals is automatically selected in such a way that all signals can be printed on one sheet.

On the right side of the window you will see a preview of the printout. Start the printout by choosing *OK*.

Export a signal file

Signal data can be exported as a text (CSV = comma separated values), as a PDF-file, as image file or as a HTML page. Export as text is suitable, for example, for further processing measurement data in a spreadsheet (Microsoft EXCEL®). The export in HTML format, on the other hand, is ideally suited for third parties to view the signal file with any web browser (e.g. Internet Explorer).

Via the menu item *Export...* in the menu *File* the export selection window will be opened.

Export as Text (CSV)

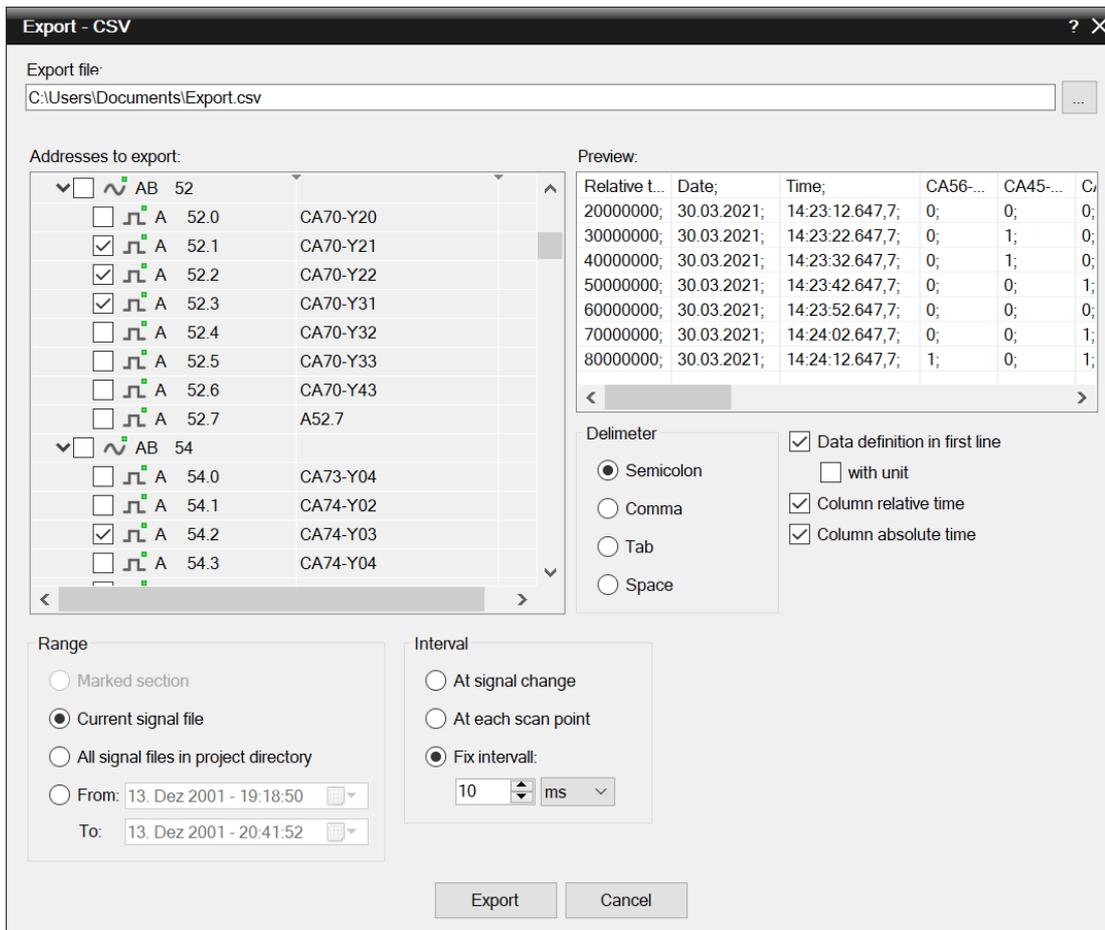


Fig. 5-24 Export as text (CSV)

Under *Export file*, specify the name of the export file. Under *Addresses to be exported*, select the PLC signals that you want to export to a CSV file. The selected addresses also appear in the *preview*. The order of the columns can easily be changed by dragging with the mouse.

Specify the *separator* between the individual measured values. If you activate *data definition in the first row*, the columns are given headings. Specify whether the header of the data column should also be specified with unit. By selecting *Relative time column* and *Absolute time column*, columns for the corresponding times are exported.

Specify the range to be exported:

Marked section

The previously [marked section](#) is exported.

Current signal file

The signal file opened in the current signal window is exported.

All signal files in the project directory

All signal files in the project directory are exported in chronological order.

Any period of time

All signal files located in the project directory and within the selected time period are exported in chronological order.

Use *Interval* to set the time interval between the exported signals. If you only want to export the signal changes, choose *When Signal Changes*. This makes the export file smaller. When selecting *with each sampling point*, data is exported at all sampling points.

The signal file is exported by pressing the *Export button*. The export file is automatically formatted in such a way that it can be processed meaningfully with Microsoft® Excel®, for example.

Export as PDF

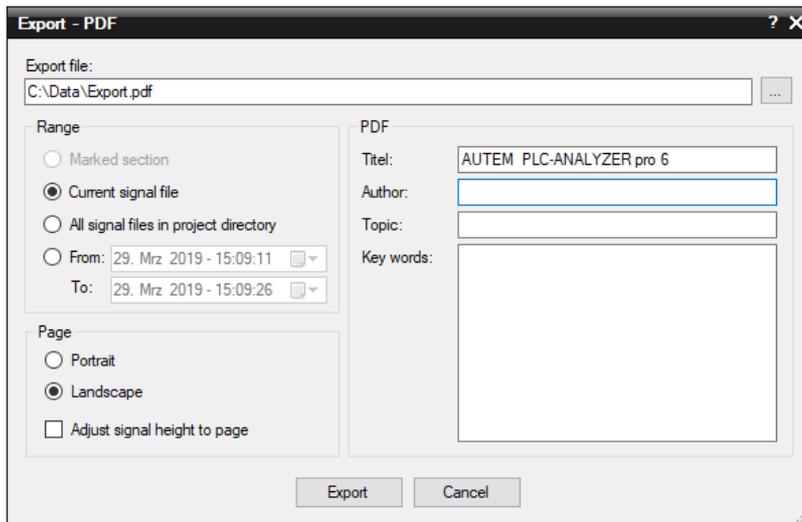


Fig. 5-25 Export as PDF

Under *Export file*, specify the name of the export file.

Specify the range to be exported:

Marked section

The previously [marked section](#) is exported.

Current signal file

The signal file opened in the current signal window is exported.

All signal files in the project directory

All signal files in the project directory are exported in chronological order.

Any period of time

All signal files located in the project directory and within the selected time period are exported in chronological order.

Indicate whether the page should be exported in portrait or landscape format and whether the signal height should be adjusted to fit the page.

Under PDF enter the title, the author, the topic and the appropriate keywords.

By pressing the *Export* button, the signal file is exported as a PDF file according to your specifications.

Export as Graphic

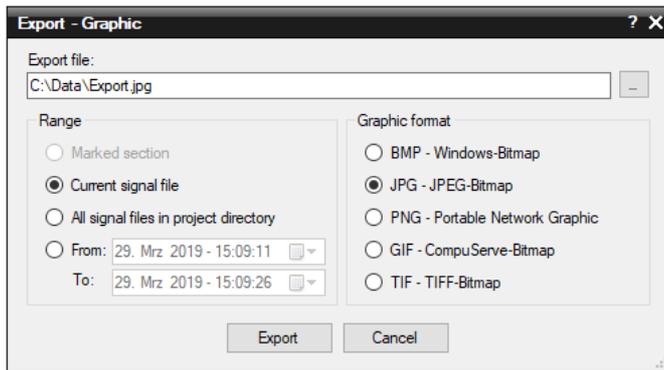


Fig. 5-26 Export as graphic

Under *Export file*, specify the name of the export file and select the desired *image format*.

Specify the range to be exported:

Marked section

The previously [marked section](#) is exported.

Current signal file

The signal file opened in the current signal window is exported.

All signal files in the project directory

All signal files in the project directory are exported in chronological order.

Any period of time

All signal files located in the project directory and within the selected time period are exported in chronological order.

Pressing the *Export* button exports the signal file as a graphic in the selected format.

Export as HTML page

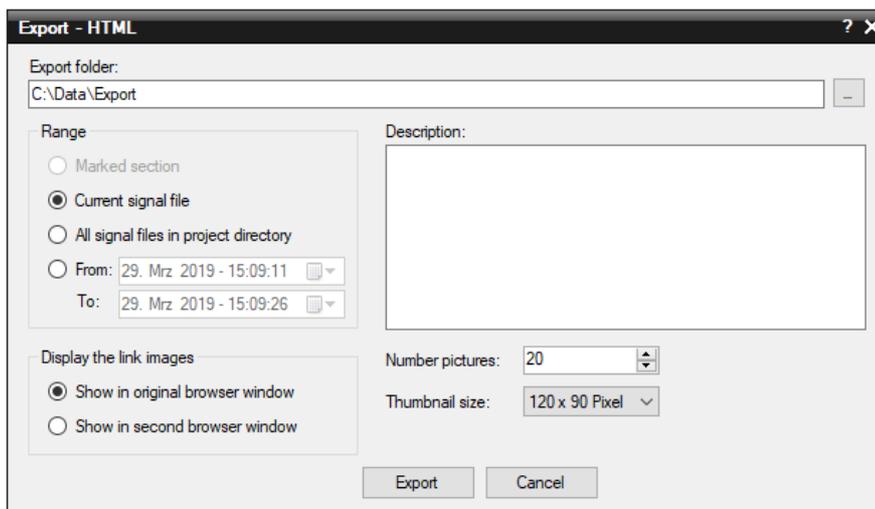


Fig. 5-27 Export as HTML page

Under *Export Folder*, select the storage path of the HTML site and the corresponding JPEG images (signal file sections).

Specify the range to be exported:

Marked section

The previously [marked section](#) is exported.

Current signal file

The signal file opened in the current signal window is exported.

All signal files in the project directory

All signal files in the project directory are exported in chronological order.

Any period of time

All signal files located in the project directory and within the selected time period are exported in chronological order.

Enter a *description* that will be displayed later on the HTML main site. Specify the *thumbnail size* and the *number of images* to split the signal file into. Under Display Link Images, specify whether the individual signal windows are to be displayed in the original browser window or always in a new browser window.

Pressing the *Export* button generates an HTML site and exports the corresponding parts of the signal file as JPEG images.

Open the HTML site with a web browser. The generated HTML site serves as an overview and displays the signal file - divided into sections - as small thumbnails. Click on the thumbnails to view the pictures in their original size.

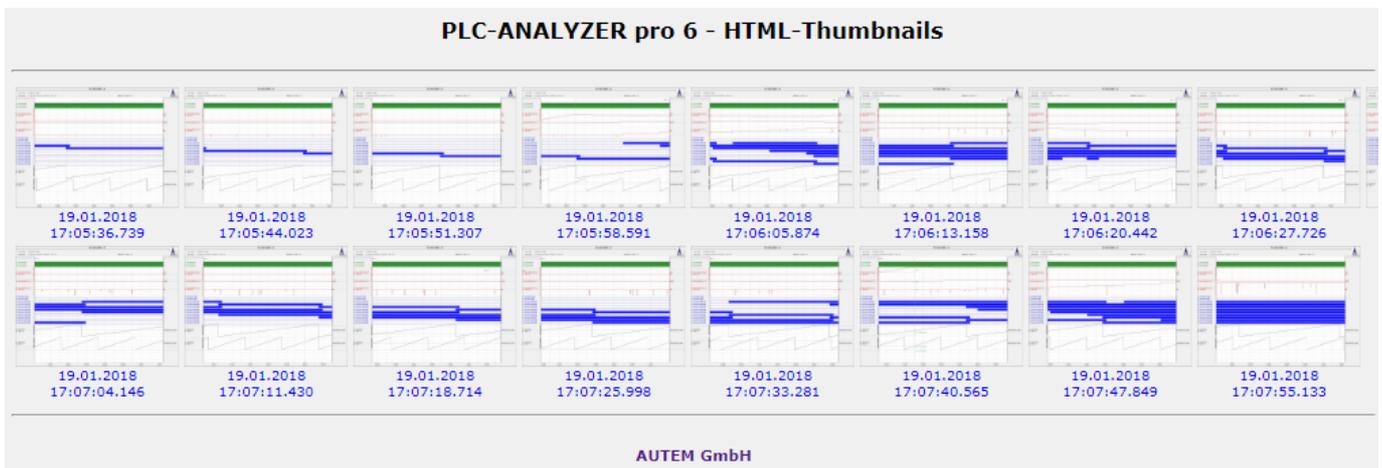


Fig. 5-28 Signal file as HTML page

Import von Signaldaten

The PLC-ANALYZER pro 6 also offers you the possibility to import data in text format. The import format is structured in the same way as the format for exporting signal files (see [Export as Text \(CSV\)](#)).

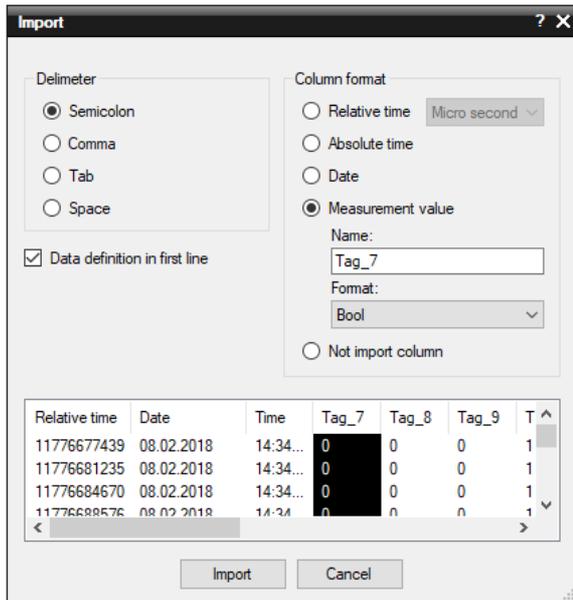


Fig. 5-29 Data import

Start the data import by selecting the menu item *Import...* in the file menu. Select a data file to be imported and click *Open*. The import window appears, in which you can make settings for the data import.

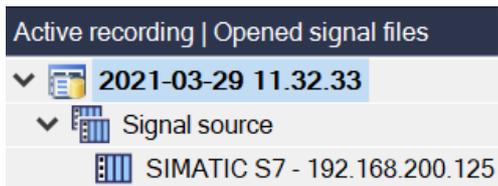
In the lower part of the window there is a data preview. Here you can see the effects of the settings you have made.

Set the *separator* that separates the measured values in the data file. Activating *data definition in the first row* means that the first row is ignored during the import, since the headings of the data columns can be located there.

Under *Column format* , you assign the corresponding format to each column. In the data preview, select the column whose format you want to change. Now change the appropriate format. A column can contain the *relative time*, *absolute time*, a *date* or a *measured value*. For a measured value, enter the *name* of the measured value and determine the *format*. Choose *Do not import column* if you want to ignore a column.

Click on *Import*. The PLC-ANALYZER pro 6 now imports the data from the text file and displays it in the signal window. You can change and save the generated signal file as usual.

Signal file info



Under *Active Recording / Opened Signal Files*, double-click the signal file whose info window you want to display. Alternatively, click on it with the right mouse button and choose *Info...*

Fig. 5-30 Open Signal file info

The *Info* window displays some information about the signal file. In addition to the *name*, *path* and *size*, the *recording time* as well as the number of signals and triggers are also listed.

Under *Description* you have the possibility to add a comment to the signal file or change it.

In the lower area of the window there is a list of all occurred [events](#) of the signal file.

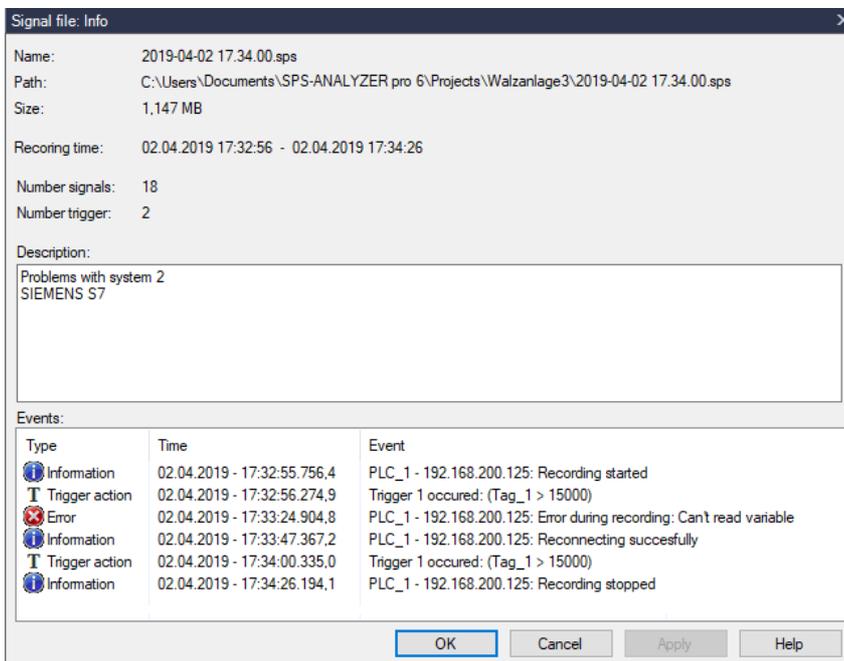


Fig. 5-31 Signal file info

Signal source

Siemens SIMATIC S7

This driver addendum describes the particularities of the following PLC drivers and gives you hints on using them.

- Siemens SIMATIC S7 - Industrial Ethernet TCP/IP / PROFINET - cycle-precise
- Siemens SIMATIC S7 - MPI / PPI / PROFIBUS - cycle-precise

With the PLC driver Siemens SIMATIC S7 - Industrial Ethernet TCP/IP / PROFINET PLC signals can be acquired via Industrial Ethernet (TCP/IP) or PROFINET. The PLC driver Siemens SIMATIC S7 - MPI / PPI / PROFIBUS enables the acquisition of PLC signals via PROFIBUS or via the multi-point MPI interface of the SIMATIC S7.

Both S7 drivers can load STEP 7 and TIA projects directly. The contained symbols are available for convenient address selection.

In addition to normal acquisition, cycle exact data acquisition is possible for many controllers. Chapter [Cycle-precise acquisition](#) explains special characteristics of this recording method.

It is important that you read through the driver addendum before using a PLC driver. Please pay attention to the WARNINGS that advise you on possible dangers when using PLC-ANALYZER pro.



WARNING

Errors that may occur in the automated facility, endangering humans or causing large-scale material damage, must be prevented by additional precautions. These precautions (e.g. independent limit monitors, mechanical interlocks) must guarantee safe operation, even in case of dangerous errors.

Installation

The PLC driver can be [added to the project as a new signal source](#). If the driver you want is not yet in the list of available signal sources, you must first activate the license for the PLC-driver with the AUTEM LicenseManager on your computer.

Installing additional hardware

If you have already connected your PG/PC to the PLC via MPI, PROFIBUS or Ethernet (LAN) for programming under STEP7 / TIA, you normally do not need to do anything else. Practically all common interfaces connections are supported.

Installing additional software

For new CPUs of types S7-1200 and S7-1500 no additional software is required .

For older controllers of the S7 series (S7-200/S7-300/S7-400) STEP7 or the TIA Portal must be installed on your computer.

Configuration

[Open driver settings](#) to set important parameters for data recording. If you have added the driver to the project several times, you can set the properties individually for each individual driver.

The screenshot shows the configuration interface for a Siemens SIMATIC S7 driver. The main window title is "System 2: Signal source - SIMATIC S7 - 192.168.200.125".

Properties Section:

- Signal source active
- Name:
- Destination:
 - Station address: Gateway
 - CPU slot: Address of the gateway:
 - Password:
 - Buttons: Show accessible nodes..., Connection test
 - S7 subnet ID of target network: -
- Cycle precise recording
 - Generate time stamp
 - With PLC safety stop
- Scan interval:

Symbols Section:

TIA / STEP7 project: ...

Absolute	Symbol	Comment
DB1.DBX0.0	M_DB_1.MachineInfo.Active	
DB1.DBB0	M_DB_1.MachineInfo.Mode	Operating mode

Addresses to record:

SIMATIC S7 - 192.168.200.125					
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	DB	1.DBW	4	M_DB_1.MachineInfo.V1
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	DB	1.DBW	6	M_DB_1.MachineInfo.V2
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	DB	1.DBW	8	M_DB_1.MachineInfo.MainEngine.Axis[0].x
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	DB	1.DBW	10	M_DB_1.MachineInfo.MainEngine.Axis[0].y
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	DB	1.DBW	12	M_DB_1.MachineInfo.MainEngine.Axis[0].z

Fig. 1-1 Settings Siemens SIMATIC S7

Load an existing TIA project under *TIA / Step 7 project*, apply the communication settings and select the desired variables conveniently by drag & drop for recording.

The settings can also be entered manually. First give the driver a meaningful *Name*. Set the *Station address* and the *slot number* of the desired CPU. Depending on the PLC driver, the *Station address* can be an MPI/PPI/PROFIBUS- or an Ethernet address. Under *Symbols* load an existing TIA project to load the contained settings of the target station directly. Enter a *password* if the CPU is password-protected.

Show accessible nodes provides you with an overview of reachable nodes. Use *Connection test* to check whether a connection to the controller can be established successfully.



NOTE

For SIMATIC S7 Ethernet-driver you can enter either the TCP/IP-address or the MAC-address of the CP. Pay attention to enter under slot the slot of the CPU and not the slot of the CP.

If the station can only be reached via a gateway, activate *Gateway*. Specify the *station address of the network transfer* and the *S7 subnet ID of the target network*. Activate *Gateway* only if there indeed a cross over occurs, e.g. from Ethernet to PROFIBUS. Refer to the hardware configuration of your STEP7 project for these settings:

If the target controller can only be reached via a network gateway, activate *Gateway*. Enter the *Address of the gateway* and the *S7 subnet ID of target network*. The gateway may only be activated if a transition actually takes place, e.g. from Ethernet to PROFIBUS. The settings for the gateway transition can be found in the hardware configuration of your STEP7/TIA project:

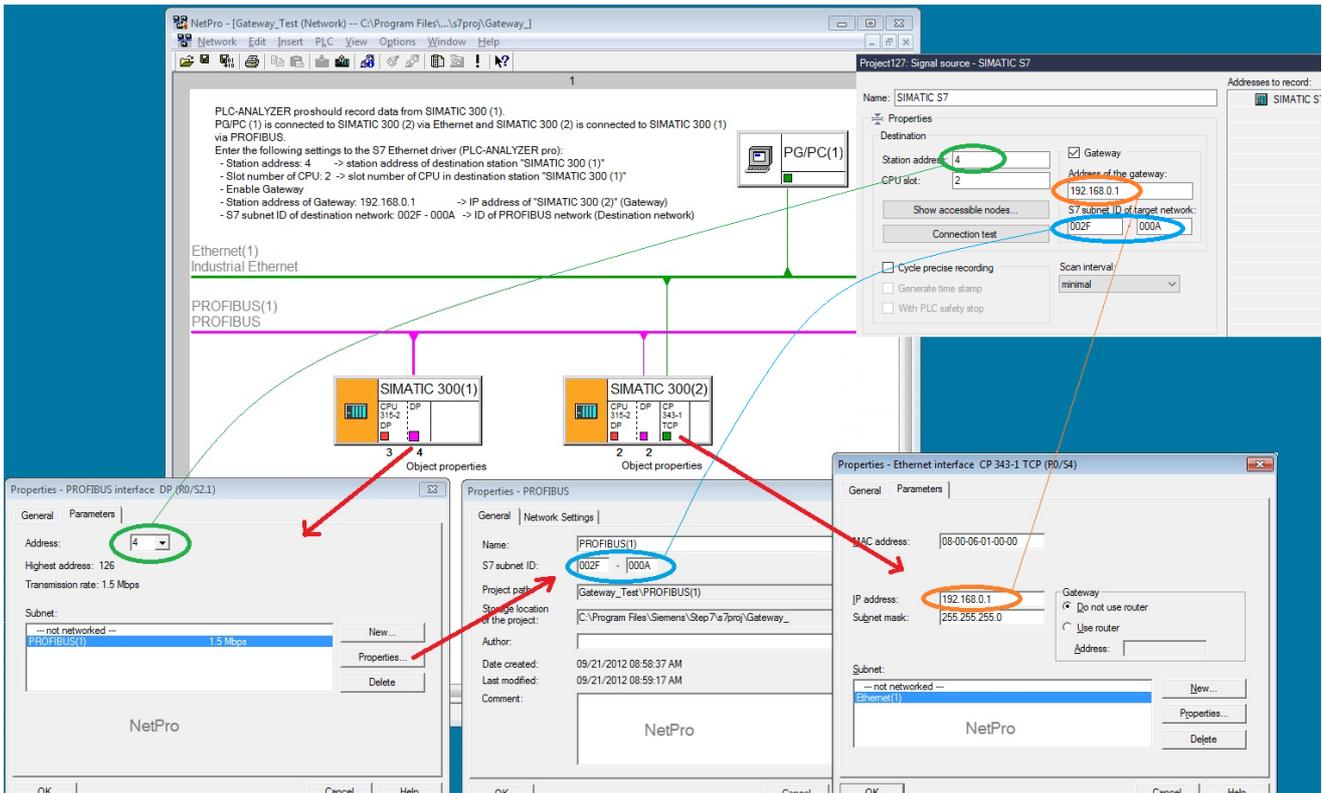


Fig. 1-2 Settings Gateway

Under *Scan interval* you specify the time interval at which measured values are read out from the PLC. A longer sampling interval can be selected for signal paths that are not time-critical, e. g. temperature. As a result, the generated signal files become smaller.

To ensure that even very short signal changes are reliably detected, activate [Cycle-precise recording](#).

Under *Symbols*, select a STEP 7- or TIA project to make the symbols of this project available for address selection. This makes it possible to use [symbolic identifiers when entering addresses](#). In addition to the absolute address, the symbolic identifier and comment are also displayed and stored in a signal- or project file.

After setting the communication properties, add the [PLC signals to be recorded](#). When a STEP7 or TIA project is loaded, the signals to be recorded can be conveniently selected from the symbol list by double-click or drag and drop.

Data acquisition

Supported PLC models and CPUs

The SIMATIC S7-Driver supports the CPUs of SIMATIC S7-200, S7-300, S7-400, S7-1200, S7-1500, M7, C7, SINUMERIK (S7), SAIA xx7, VIPA S7 and S7-PLCSIM.

Recordable PLC addresses

The following table shows the recordable addresses and the corresponding address syntax:

Syntax	Type of address	Example
Qx.z	Output byte x, bit z	Q32.4
QBx	Output byte x	QB9
QWx	Output word x	QW14
QDx	Output double word x	QD98
Ix.z	Input byte x, bit z	I17.0
IBx	Input byte x	IB127
IWx	Input word x	IW12
IDx	Input double word x	ID124
Fx.z	Flag byte x, bit z	F3.7
FBx	Flag byte x	FB250
FWx	Flag word x	FW24
FDx	Flag double word x	FD134
FBx	Flag byte x	FB250
Tx	Timer x	T2
Cx	Counter x	C5
DByDBXx.z	Data byte x, bit z from data block y	DB23DBX2.5
DByDBBx	Data byte x from data block y	DB2DBB5
DByDBWx	Data word x from data block y	DB12DBW5
DByDBDx	Data double word x from data block y	DB27DBD0
PIB x	Peripheral input byte x	PEB 231
PID x	Peripheral input double word x	PED 304

Table 1-1: SIMATIC S7 address syntax

Number of recordable addresses

A maximum of 16 million addresses can be acquired from up to 250 signal sources.

Time behaviour and particularities



NOTE

Acquiring data with PLC-ANALYZER pro 6 results in a small increase in cycle time in the automation device to the same extent as with STEP7/TIA in the monitor operating mode.

In the TIA Portal, the value for the CPU property "*Communication load - Cycle load due to communication*" should not be set too high. To keep the cycle time extension as low as possible, the value 20% is recommended here.

The intervals between scan transfers from the SIMATIC PLC to the computer depend on the PLC-CPU, the number of acquired signals and kind of connection. In addition, the scanning distance is influenced by the size of the network and the selected transmission speed.

With an S7-300, the scan distance for acquisition via MPI/PROFIBUS for one byte is approx. 25-30 ms, i.e. from a cycle time > 30 ms, one scan is received for each cycle. With each additional byte acquired, the scan distance increases by about 2 ms.

A scan interval of at least 1 ms can be achieved for the detection of an S7-1500 via PROFINET/Industrial Ethernet (TCP/IP).

Cycle-precise recording



WARNING

The PLC ANALYZER pro 6 programs a small addition to the PLC program in the controller for cycle-precise acquisition. We would like to point out that an influence of this modification on the mode of operation of the PLC or the PLC program cannot be completely excluded.

The cycle-precise signal acquisition enables the continuous measurement of selected signals in each PLC cycle without gaps.

With cycle-accurate signal acquisition, a limited number of signals are pre-measured within the PLC. The selected signals are stored in the memory of the PLC in each PLC cycle and transferred to the PC by means of intelligent procedures in such a way that continuous cycle-accurate acquisition is possible.

As a user, you do not see any visible difference from the normal recording. Even live display is possible as usual. Cycle-precise signal acquisition is available for many controllers of the S7 family.

Configuration of PLC-driver for cycle-precise acquisition

In the [Settings](#) window of the S7 driver, activate the *cycle-precise acquisition*.



Fig. 1-3 Cycle-precise acquisition (SIMATIC S7)

For recording of very brief signal changes, activate *Cycle-precise recording*.

Generate time stamps is used to provide the data with time stamps during cycle-precise recording. Deactivate this option if you have so much data to record that data loss cannot be prevented. In this case no time information is available during the signal analysis.

You also specify here whether the PLC is to be stopped for safety reasons before and after the transfer of the data blocks.



NOTE

If more addresses are stored to the memory than can be read from the PLC at one time, an overflow of the circular memory will occur and data will be lost. This can be avoided by reducing the number of addresses to be recorded.

Input of addresses

Up to 200 addresses (byte-, word- or double word-values) can be acquired simultaneously in the cycle-precise acquisition mode. This restriction is the result of the restricted memory capacity of the PLC and the transmission speed of the MPI-, PROFIBUS- or Ethernet-Interface. If the number of signals to be recorded is too large, a recording of the data without gaps is not guaranteed in every case. The number of addresses that can be recorded without gaps depends on the following factors:

- Cycle time of the PLC program
- Transmission speed of the Interface
- Recording with/without time stamp

With an S7-300 (CPU315-2 DP), for example, with a cycle time of 10 ms, about 50 bytes can be recorded without gaps.

Start acquisition



WARNING

It is essential to ensure that the system is brought into a safe condition before the modification for cycle precise recording is carried out. If "With PLC safety stop" is set, the PLC ANALYZER pro 6 switches the controller to the STOP state for a short time to transfer or modify the blocks. The process after the end of the recording is analogous.

Now start the [Acquisition](#). Depending on the default settings you have selected in the [Settings](#) window of the PLC driver, the blocks are transferred either during operation or after the controller has been stopped. One of the following message windows appears:

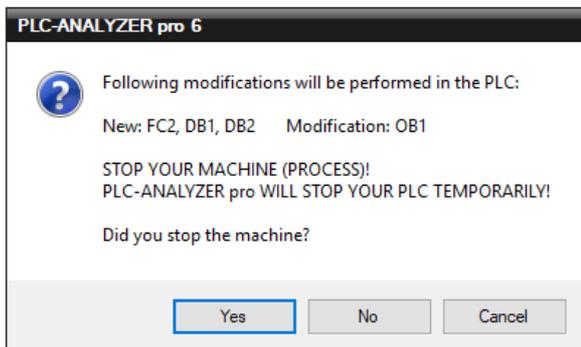


Fig. 1-4 Message before modifications in the PLC for pre-setting „With PLC safety stop“

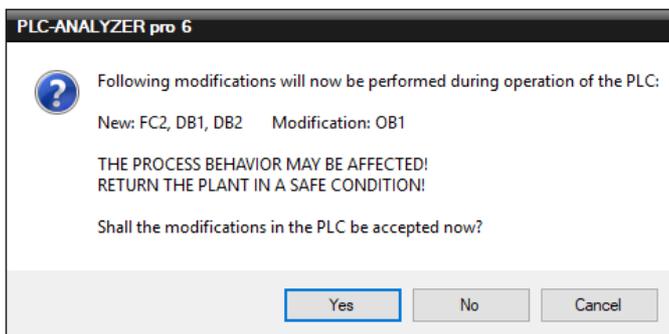


Fig. 1-5 Message before modifications in the PLC for pre-setting „No PLC safety stop“

Confirm with *Yes* only after you have stopped the process or if it is in a safe state. Make sure that damages to person or property by impairment of the function of the control is impossible!

The PLC-ANALYZER pro 6 searches for free block numbers in the PLC and generates one function block and two data blocks for data recording. In addition, a call to the new function block is added to the end of OB1.

The controller is in the RUN state or is now switched back to the RUN state. The cycle-precise acquisition begins. The signal changes are now displayed live on the screen.

Recording is stopped with *Stop acquisition*. You should now stop your system (process) or bring the system into a safe condition. Removal of the modifications is now done analogously in the STOP state or online. One of the following message windows will appear:

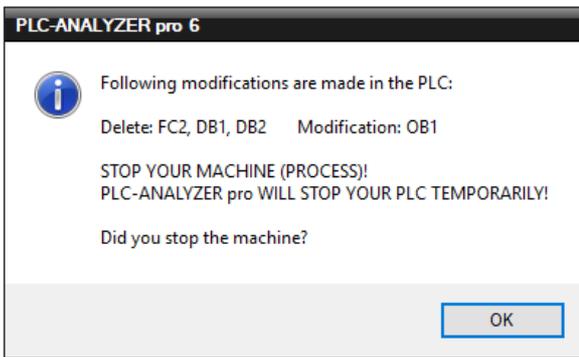


Fig. 1-6 Message before modifications in the PLC for pre-setting „With PLC safety stop“

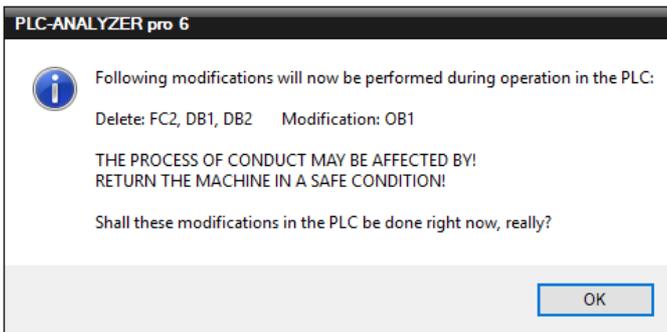


Fig. 1-7 Message before modifications in the PLC for pre-setting „No PLC safety stop“

Confirm the message after you have stopped your system or bring it into a safe condition. The original state in the PLC is restored now.

After the end of acquisition, the last signal file created is automatically opened for display.

Particularities in signal display and analysis

The evaluation of signal files acquired with cycle accuracy is practically identical to that of normal signal files. However, if no time stamp is generated during recording (see configuration of the S7 driver), no time is assigned to the data. In this case, the time is specified in cycles. The time base here is "ZP" (cycles per pixel).

Example: With a set time base of 0.1 ZP, a PLC cycle is displayed over 10 screen pixels.

If more addresses are recorded than can be continuously read from the PLC, the PLC ring buffer overflows. This causes data to be lost. These recording gaps during acquisition are displayed as a grey line.

Menu commands

File

New Project...

Create a new project.

Shortcut:

Toolbar: 

Keyboard: <Ctrl> + <N>

See also:

[Create a new project](#)

Open project...

Opens an existing project.

Shortcut:

Toolbar: 

Keyboard: <Ctrl> + <O>

See also:

[Edit project](#)

Close project

Use this menu command to close an open project.

Save project

Saves an open project with the current name.

Shortcut:

Toolbar: 

Keyboard: <Ctrl> + <S>

Save project under...

Saves an open project under a new name.

Open signal file...

Opens an existing signal file.

Shortcut:

Toolbar: 

See also:

[Open signal file](#)

Close signal file

This menu command closes the active signal file.

Save signal file

Saves the active signal file.

Shortcut:

Toolbar: 

See also:

[Save signal file](#)

Save signal file as...

Saves the active signal file under a new name.

See also:

[Signaldatei speichern](#)

Print...

Before you select this command, activate the signal window whose signal selection is to be printed.

Use this command to print a signal file. After selecting this command, the print window appears in which you can change the settings for printing.

Shortcut:

Toolbar: 

See also:

[Print a signal file](#)

Export

With this menu command you can export a signal file in different formats.

See also:

[Export a signal file](#)

Import...

Use this menu item to import data in text format available into a signal file.

See also:

[Import a signal file](#)

Close

Closes all open signal files and exits the PLC-ANALYZER pro.

Signal file

Signal statistic...

This command provides extensive information about the signals of the current signal file.

See also:

[Signal statistics](#)

Compare...

This command compares signal files with each other.

Shortcuts:

Toolbar: 

See also:

[Compare signal files](#)

Merge...

Merge the data of two signal files into one.

See also:

[Merge signal files](#)

Time measurement

With this menu command you can perform time measurements in the signal file.

Shortcuts:

Toolbar:

Time difference measurement: 

Bit measurement: 

See also:

[Time measurement](#)

Area

This menu item allows you to select, save, print and export an area.

See also:

[Mark area](#)

[Export a signal file](#)

[Print a signal file](#)

Note

This menu item allows you to add or remove notes.

See also:

[Add note](#)

Search

Signal value...

Use this menu command to find specific signal values.

Shortcuts:

Keyboard: <Ctrl> + <F>

See also:

[Search signal value](#)

Trigger...

To search for specific triggers, use the trigger search.

Shortcuts:

Keyboard: <Ctrl> + <T>

See also:

[Search trigger](#)

Note...

To find specific notes, use the note search.

Shortcuts:

Keyboard: <Ctrl> + <H>

See also:

[Search Note](#)

Time...

Use this menu command to find a specific time.

Shortcuts:

Keyboard: <Ctrl> + <Z>

See also:

[Search time](#)

View

Grid

Use this command to create a grid over the signal window. This enables you to read more accurately.

Shortcuts:

Toolbar: 

See also:

[Edit signal window](#)

Absolute time display

Switches between absolute and relative time display.

Shortcuts:

Toolbar: 

See also:

[Edit signal window](#)

Show sample points

Activates or deactivates the display of scan points in the active signal window.

Shortcuts:

Toolbar: 

See also:

[Edit signal window](#)

Bar display

Switches bar display on or off for bit signals in the active signal window.

Shortcuts:

Toolbar: 

See also:

[Edit signal window](#)

Tool bars

Use this menu command to switch the toolbar on or off.

The toolbar is located below the menu bar. The toolbar buttons can be used to quickly execute frequently used menu commands without having to select them from the menu.

Status bar

Turns the status bar on or off.

Extras

Options...

Changes the settings of the PLC-ANALYZER pro.

See also:

[Options](#)

Window

Cascade

With this command, all open signal windows are displayed overlapping.

Shortcuts:

Toolbar: 

See also:

[Signal display and analysis](#)

Tile vertical

This command displays all open signal windows side by side.

Shortcuts:

Toolbar: 

See also:

[Signal display and analysis](#)

Help

Show help...

Opens the help window of the PLC-ANALYZER pro and displays a list of the help topics under which you can choose.

Shortcuts:

Toolbar: 

Keyboard: <F1>

Search for updates...

Searches the Internet for current program versions of the PLC-ANALYZER pro

Info...

Provides information about licenses, edition and copyright.

Toolbar symbols

Symbols of the tool bar:

	New Project (Menu File)
	Open Project (Menu File)
	Save Projekt (Menu File)
	Open signal file (Menu File)
	Save signal file (Menu File)
	Print (Menu File)
	Export (Menu File)
	Compare (Menu Signal file)
	Area - Save (Menu Signal file)
	Note - Insert (Menu Signal file)
	Time measurement - Time difference (Menu Signal file)
	Time measurement - Bit measurement (Menu Signal file)
	Start Recording
	Stop Recording
	Window cascade (Menu Window)
	Window tile (Menu Window)
	Search signal value (Menu Search)
	Current search backwards
	Current search forwards
	Show help (Menu Help)

Symbols of the [y\(t\)-signal window](#):

	Edit settings of the signal window
	Change between live- and analysis display
	Overwrite or scroll live data
	Absolute or relative time display
	Display grid
	Display scan points
	Bar display

Help

Frequently Asked Questions - FAQ

Installation und License management

USB License-Key Dongle does not work

- Test whether the dongle is recognized by another USB port.
- Install the latest dongle driver (WIBU-KEY Runtime). Download under <https://www.wibu.com/downloads-user-software.html>.

Start of PLC-ANALYZER pro

PLC-ANALYZER pro cannot be started

Has the licence of the PLC-ANALYZER pro been activated?

No drivers can be loaded

- Make sure that the drivers are present (see info-window)
- Check whether the licenses from the driver are on the PC (see [Installation](#))

Connection to the PLC and data acquisition

SIMATIC S7: Error connecting via MPI/PROFIBUS

- STEP7 (min version 3.0) has to be installed.
- If you are not able to make a connection to you PC by STEP7, check the hardware connection between PLC and PC.
- Check, if the right settings for the connection under „Set PG/PC interface“ (STEP7) have been made. Do not use the “AUTO” connection here, but use the direct connection, e.g. CP5611 (MPI) and not CP5611 (AUTO) .
- If you have installed TIA-Portal, you must start “Set PG/PC interface” from windows control panel and the corresponding connection type has to be set there.
- Have you set the correct PLC address (MPI, PROFIBUS or TCP/IP address) in the [settings of the S7 driver](#) in the PLC-ANALYZER pro?
- Have you set the correct CPU slot in the PLC-ANALYZER pro in [settings of the S7 driver](#)
- The [gateway](#) may only be activated if there really is a cross over, e. g. from PROFIBUS to Ethernet

SIMATIC S7: Error connecting via Ethernet TCP/IP

- For a correct connection via Ethernet Softnet has to be installed
- If you made the right settings for an „Industrial Ethernet“ connection under „Set PG/PC interface“, you have to enter the MAC-address in the S7-driver of the PLC-ANALYZER pro (Format: xx.xx.xx.xx.xx.xx). If you use the connection „TCP/IP“ a TCP/IP-address has to be entered in the S7-driver of PLC-ANALYZER pro (Format: xxx.xxx.xxx.xxx)
- If the TIA portal is installed, *PG/PC interface* has to be set, the Windows control panel must be called and the corresponding connection type must be set there.
- In the [settings of the S7 driver](#), the correct *slot* of the CPU (not the CP) must be entered under Slot.
- The [gateway](#) may only be activated if there really is a cross over, e. g. from Ethernet to PROFIBUS

SIMATIC S7-1200: Connection cannot be made

- Make sure that you have entered slot = 1 in the [driver properties](#).

SIMATIC S7-1500: Connection cannot be made

- Make sure that you have entered slot = 1 in the [driver properties](#).

SIMATIC S7-200: Connection cannot be made

- STEP7 has to be installed (S7-Micro-Win is not enough)
- Choose under “Set PG/PC-interface” an MPI-connection or a PPI-connection
- If you use MPI you have to adjust the MPI-speed to 19200 Baud.
- Check, if you can see the S7-200 via the „accessible notes“ in STEP7. Use the shown address and enter it in the PLC-driver of the PLC-ANALYZER pro under the station address.

SIMATIC S5: Connection cannot be made

- If you are not able to make a connection to your PLC via Step5, check the hardware connection between PLC and PC.
- Did you choose the right COM-port under [Setting of S5-Treibers?](#)
- Close all programs, which interfere with the chosen COM-interface (e. g. STEP5)

Allen-Bradley: A connection via DH+ cannot be made

- Did you insert the right DH+-address settings of that card into the [settings of the Allen-Bradley driver?](#)
- KTXD-Card: Enter the address, which you entered on the card.
- PCMK-Card: Open in the system control the program PCMKInfo. Enter the driver properties shown in the PCMK-Info.

CoDeSys: Connection cannot be made

- Under Online / communication parameter the CoDeSys programming software lists all parameter necessary for a correct communication with the PLC. These parameter have to be set in the [settings of the CoDeSys driver](#) as well.
- For data recording, it is absolutely necessary that the [variable export has been activated](#) and that the symbols are available in the PLC.
- The target system must be able to transmit the symbolism. In CoDeSys, the option Send symbol file must be activated in the target system settings.

AD_USB-Box: Interruptions during recording

- Perhaps there are problems with other USB-units you use. Remove all other USB-units from your PC (e.g. your USB-mouse etc.) and try it again.
- Check whether your computer's energy-saving options disable the USB interfaces after a certain amount of time.

General help for problems with connecting to the PLC

- If it is not possible to make a connection to your PLC via your programming software (Concept, Medoc, Modsoft...) check the hardware connection first.
- Did you make the right settings in the [signal source](#) (COM-Port, Baud rate, PLC-Address...)?

- Close all programs which are using the same interface, these were set in your [signal source](#).
- Error „Invalid address“ means that there is a connection to a PLC but some addresses you have entered are not available in your PLC. Please delete all addresses which are not available in the [signal source](#).

There are interruptions during the recording

- Interruptions while using cycle-precise recording be possible. Reduce the number of variables you like to record and/or use a faster PLC-PC connection(see [cyclus exact recording](#)).
- Deactivate the Power manager via the system control of your PC.
- Check the hardware connection. Take care for electromagnetic shielding against electrical and magnetical fields

General questions

How many addresses can be recorded at most

Any number of addresses can be recorded with the SPS-ANALYZER pro. However, there are drivers that support a smaller number. Please refer to the chapter "Data acquisition" of the corresponding driver.

How many drivers can be loaded at the same time

- A maximum of 250 drivers can be used at the same time
- During the cycle-accurate recording (S5 and S7), recording is only allowed with one driver.

Technical Support

If you have any questions about the PLC-ANALYZER pro, first refer to the online help and the manual. If you don't find an answer to your question here, either look at the AUTEM website www.autem.de or contact:

AUTEM GmbH

Dithmarscher Straße 29
26723 Emden
Germany

Tel.: +49 (0)4921 / 9610-0

Fax: +49 (0)4921 / 9610-96

E-Mail: support@autem.de