TwinCAT 3 Engineering

Manual

TC3 Scope View

TwinCAT 3

Version:	**1.9**
Date:	**2019-03-20**
Order No.:	**TE13xx**

BECKHOFF

Table of contents

1 Foreword

1.1 Notes on the documentation

This description is only intended for the use of trained specialists in control and automation engineering who are familiar with the applicable national standards.
It is essential that the documentation and the following notes and explanations are followed when installing and commissioning the components.
It is the duty of the technical personnel to use the documentation published at the respective time of each installation and commissioning.

The responsible staff must ensure that the application or use of the products described satisfy all the requirements for safety, including all the relevant laws, regulations, guidelines and standards.

Disclaimer

The documentation has been prepared with care. The products described are, however, constantly under development.
We reserve the right to revise and change the documentation at any time and without prior announcement.
No claims for the modification of products that have already been supplied may be made on the basis of the data, diagrams and descriptions in this documentation.

Trademarks

Beckhoff®, TwinCAT®, EtherCAT®, Safety over EtherCAT®, TwinSAFE®, XFC® and XTS® are registered trademarks of and licensed by Beckhoff Automation GmbH.
Other designations used in this publication may be trademarks whose use by third parties for their own purposes could violate the rights of the owners.

Patent Pending

The EtherCAT Technology is covered, including but not limited to the following patent applications and patents:
EP1590927, EP1789857, DE102004044764, DE102007017835
with corresponding applications or registrations in various other countries.

The TwinCAT Technology is covered, including but not limited to the following patent applications and patents:
EP0851348, US6167425 with corresponding applications or registrations in various other countries.

EtherCAT® is registered trademark and patented technology, licensed by Beckhoff Automation GmbH, Germany

Copyright

BECKHOFF

1.2   Safety instructions

Safety regulations

Please note the following safety instructions and explanations!
Product-specific safety instructions can be found on following pages or in the areas mounting, wiring, commissioning etc.

Exclusion of liability

All the components are supplied in particular hardware and software configurations appropriate for the application. Modifications to hardware or software configurations other than those described in the documentation are not permitted, and nullify the liability of Beckhoff Automation GmbH & Co. KG.

Personnel qualification

This description is only intended for trained specialists in control, automation and drive engineering who are familiar with the applicable national standards.

Description of symbols

In this documentation the following symbols are used with an accompanying safety instruction or note. The safety instructions must be read carefully and followed without fail!

⚠ **DANGER**
Serious risk of injury!
Failure to follow the safety instructions associated with this symbol directly endangers the life and health of persons.

⚠ **WARNING**
Risk of injury!
Failure to follow the safety instructions associated with this symbol endangers the life and health of persons.

⚠ **CAUTION**
Personal injuries!
Failure to follow the safety instructions associated with this symbol can lead to injuries to persons.

NOTE
Damage to the environment or devices
Failure to follow the instructions associated with this symbol can lead to damage to the environment or equipment.

Tip or pointer

This symbol indicates information that contributes to better understanding.

2   Overview

TwinCAT 3 Scope is the charting and analysis tool for TwinCAT. Variables in TwinCAT can be recorded and displayed graphically. Sampling rates can be adjusted individually for each channel. Recordings in the µs range and long-term recordings over several days are equally possible. Due to division into a Scope View with multi-core support for the display of the signals and a Scope Server for the recording of the values, it is possible it to connect to servers distributed in the field from a central View. A tool is thus available not only for machine commissioning, but also for process monitoring. The TwinCAT 3 Scope functionality includes cursor tools and trigger functions.

TwinCAT 3 Scope also sets new standards for Engineering. Like TwinCAT 3 itself, it is integrated in Microsoft Visual Studio. It is thus possible to use TwinCAT 3 projects and Scope projects in parallel in a single solution. Variables can be shifted very simply from the TwinCAT 3 project to a Scope configuration. Of course, the Scope can also still be used as a standalone tool in the framework of the Visual Studio shell.

Due to the constantly increasing range of functions required of data analysis tools, the TwinCAT 3 Scope is subdivided into product levels with different ranges of features. The TwinCAT 3 Scope Base contains a license-free View and a license-free Server. Both are installed together with TwinCAT 3 XAE.

Product note

The TwinCAT 3 Scope software consists of two products:

- TwinCAT 3 Scope View is a TwinCAT 3 engineering product and supplies the graphic interface for the configuration of recordings and the display of signal curves. The View is available in different product levels and is licensed for the device on which it is displayed. This document contains the technical product description.
- TwinCAT 3 Scope Server is a TwinCAT 3 function and provides the software for data logging. The server sends the recorded data to the View. The software is installed on distributed devices or on the local target device. The server license is issued for the device on which the server runs. In most cases the Base version, which is installed together with TwinCAT XAE or the Scope View, is sufficient. The server license is only required if it is used independently, i.e. without View, e.g. for control via a PLC function block. The TwinCAT 3 Scope server is documented separately as part of the TwinCAT 3 functions.

Both components must be available in the system in order to be able to use the Scope.

Principle of operation

The principle of operation of the two main components View and Server is explained in detail in the Basic Concept [▶ 20] in the Technical Introduction.

Product level / feature list

The following table shows which functionalities are available with which TwinCAT Scope level and the corresponding licensing arrangement.

Features	Scope Base		Scope Server Full Licence		Scope View Professional	
	Server	View	Server		View	
			Full	7 days trial version	Full	7 days trial version
General:						
Free of charge	✔	✔	✘	✔	✘	✔
Local record	✔	✔	✔	✔	✔	✔
Remote Record using Target Server	✘	✔	✔	✔	✔	✔
Remote Record using Local Server	✔	✔	✔	✔	✔	✔
Scope Control Integration	-	✘	-	-	✔	✔
Long Time Records > 1h	-	✘	-	-	✔	✘
Ring Buffer	✔	✔	✔	✔	✔	✔
Auto Save	-	✘	-	-	✔	✔
Restart Record	-	✘	-	-	✔	✔
View Multicore Support	-	✔	-	-	✔	✔
Communication:						
ADS	✔	✔	✔	✔	✔	✔
OPC UA	✘	✘	✔	✔ (3 Nodes)	✔	✔ (3 Nodes)
Base:						
Configuration Wizard	-	✔	-	-	✔	✔
Target Browser Filter	-	✔	-	-	✔	✔
Individual Sample Rate	-	✔	-	-	✔	✔
Drag and Drop for Configuration	-	✔	-	-	✔	✔
Docking Windows	-	✔	-	-	✔	✔
YT Time based Charts	-	✔	-	-	✔	✔
XY Charts	-	✔	-	-	✔	✔
Array Bar Charts	-	✔	-	-	✔	✔
Overview Chart	-	✔	-	-	✔	✔
Stacked Y-Axes	-	✔	-	-	✔	✔

Features	Scope Base		Scope Server Full Licence		Scope View Professional	
Logarithmic Y-Axes	-	✔	-	-	✔	✔
Chart synchronisation	-	✔	-	-	✔	✔
Master Chart synchronisation	-	✔	-	-	✔	✔
Channel Marks	-	✔	-	-	✔	✔
Channel Offset	-	✔	-	-	✔	✔
Channel Scaling	-	✔	-	-	✔	✔
Panning	-	✔	-	-	✔	✔
Zoom	-	✔	-	-	✔	✔
Cursor	-	✔	-	-	✔	✔
Fillmode	-	✔	-	-	✔	✔
Print Report	-	✗	-	-	✔	✔
Data export wizard	-	✔	-	-	✔	✔
Export tool	✔	✔	✔	✔	✔	✔
Extended data export formats	-	✗	-	-	✔	✗
Trigger:						
Event Marker	-	✔	-	-	✔	✔
Start Record	-	✔	-	-	✔	✔
Stop Record	-	✔	-	-	✔	✔
Stop Display	-	✔	-	-	✔	✔
Restart Display	-	✔	-	-	✔	✔
Start Subsave	-	✗	-	-	✔	✔
Stop Subsave	-	✗	-	-	✔	✔
Units:						
Unit Property	-	✔	-	-	✔	✔
Unit Wizard	-	✔	-	-	✔	✔
Unit Dictionary Editor	-	✗	-	-	✔	✗
Unit generation by Terminal selection	-	✗	-	-	✔	✗
Parse Units from PLC Code	-	✔	-	-	✔	✔

BECKHOFF

Features	Scope Base		Scope Server Full Licence		Scope View Professional	
Units in Target Browser	-	✔	-	-	✔	✔
Units in Tooltips	-	✔	-	-	✔	✔
Units in Cursor Window	-	✔	-	-	✔	✔

3   Installation

3.1   System requirements

The following system requirements must be fulfilled for proper functioning of the TwinCAT 3 Scope View.

Supported operating systems

Windows Embedded Standard 7, Windows 7, Windows 10

TwinCAT

Minimum is TwinCAT 3 ADS.

.NET Framework

.NET Framework 4.5.1 is required.

The TwinCAT 3 Scope Base is installed together with TwinCAT 3 XAE. The further product levels and features are enabled by licensing. If you use a newer Scope version outside a TwinCAT 3 XAE update, or if you want to install the Scope in stand-alone mode on a PC without TwinCAT 3 Engineering, you can download the corresponding setup file from the internet.

Scope View can be licensed as a full version or as a 7-day test version. For limitations of the test version, please refer to the product overview page.

3.2   Downloading the setup file

TwinCAT Scope View is available for download from the Beckhoff website. In each case the latest product version that can be licensed for each product level is made available. To download the setup file, perform the following steps:

1. Start a web browser of your choice and go to the Beckhoff website at www.beckhoff.de.

2. In the tree structure of the website, navigate to the node Automation > TwinCAT 3 > TE1xxx | TC3 Engineering > TE13xx | TC3 Scope View Professional.

3. Click on the Download link in order to place the software in the shopping cart. Then click on **Start download**.

4. (Optional) Transfer the downloaded file to the TwinCAT runtime system you wish to install the product on.

3.3   Installation

The following section describes how to install the TwinCAT 3 Function for Windows-based operating systems.

✓ The TwinCAT 3 Function setup file was downloaded from the Beckhoff website.

1. Run the setup file as administrator. To do this, select the command **Run as administrator** in the context menu of the file.

   ⇨ The installation dialog opens.

2. Accept the end user licensing agreement and click **Next**.
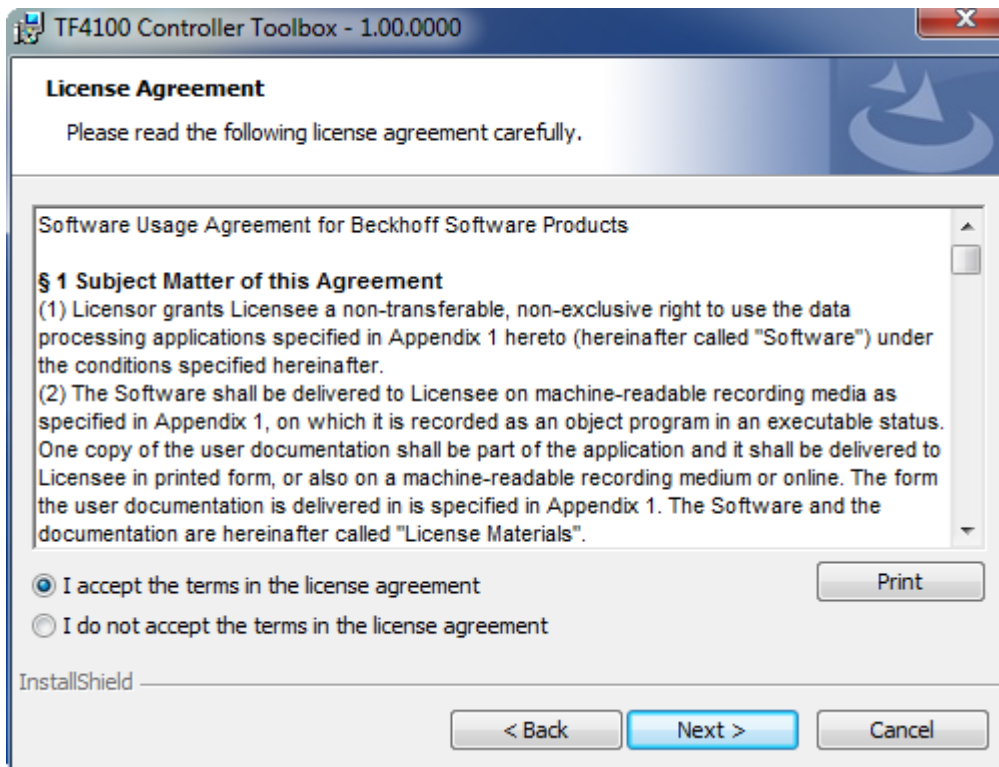
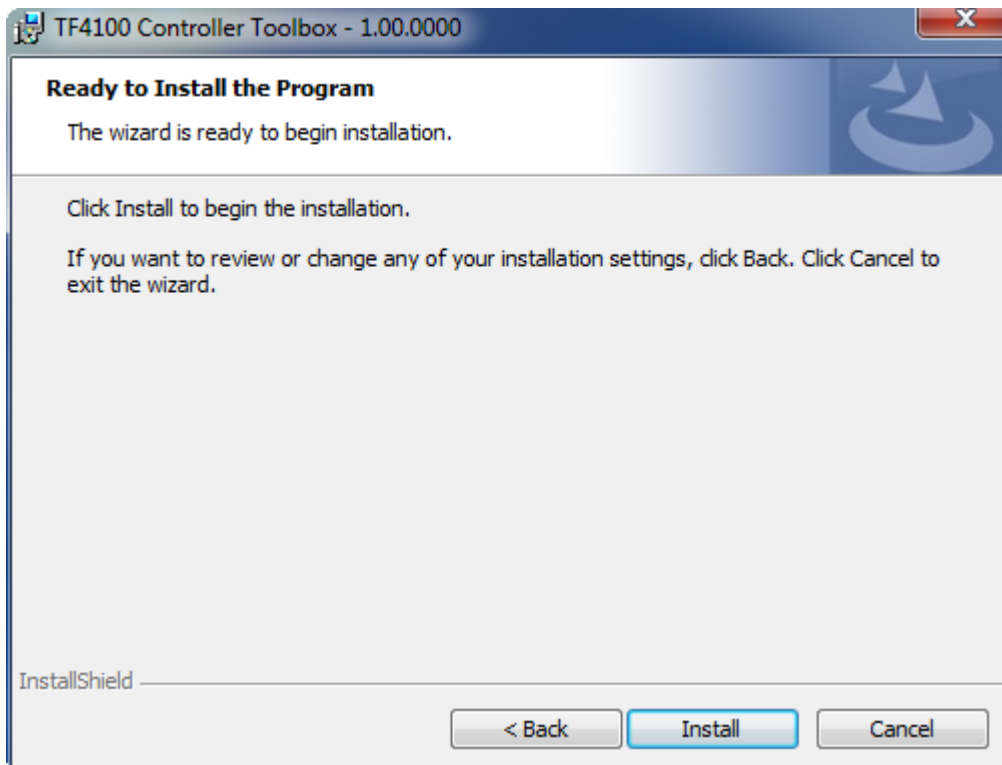
3. Enter your user data.
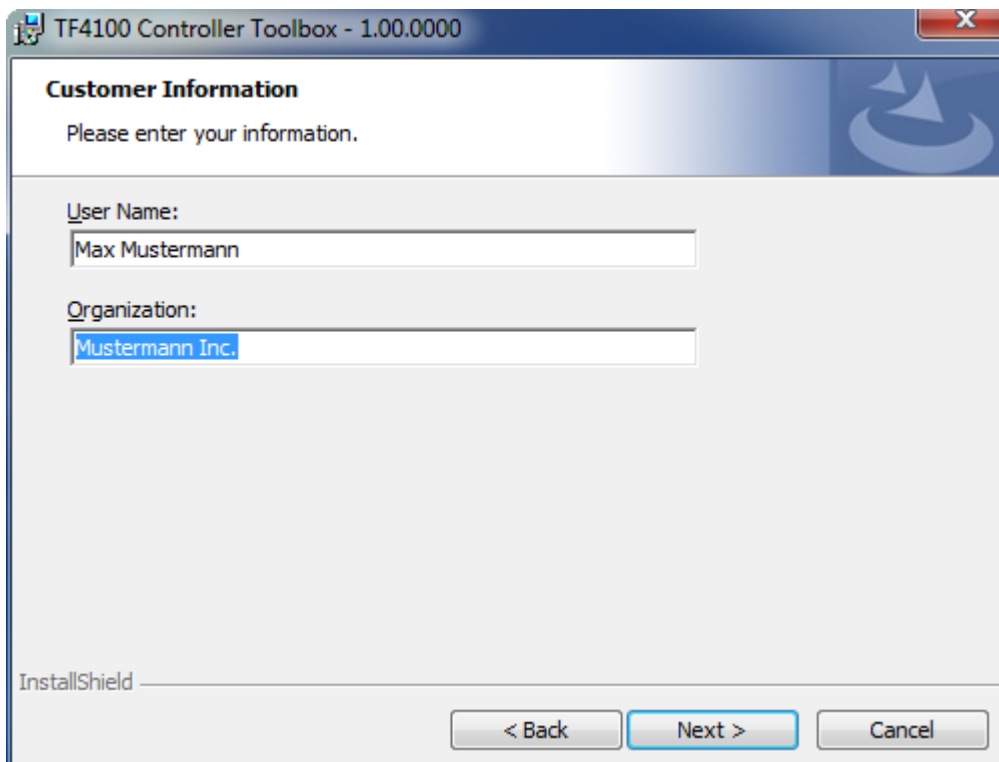
4. If you want to install the full version of the TwinCAT 3 Function, select **Complete** as installation type. If you want to install the TwinCAT 3 Function components separately, select **Custom**.
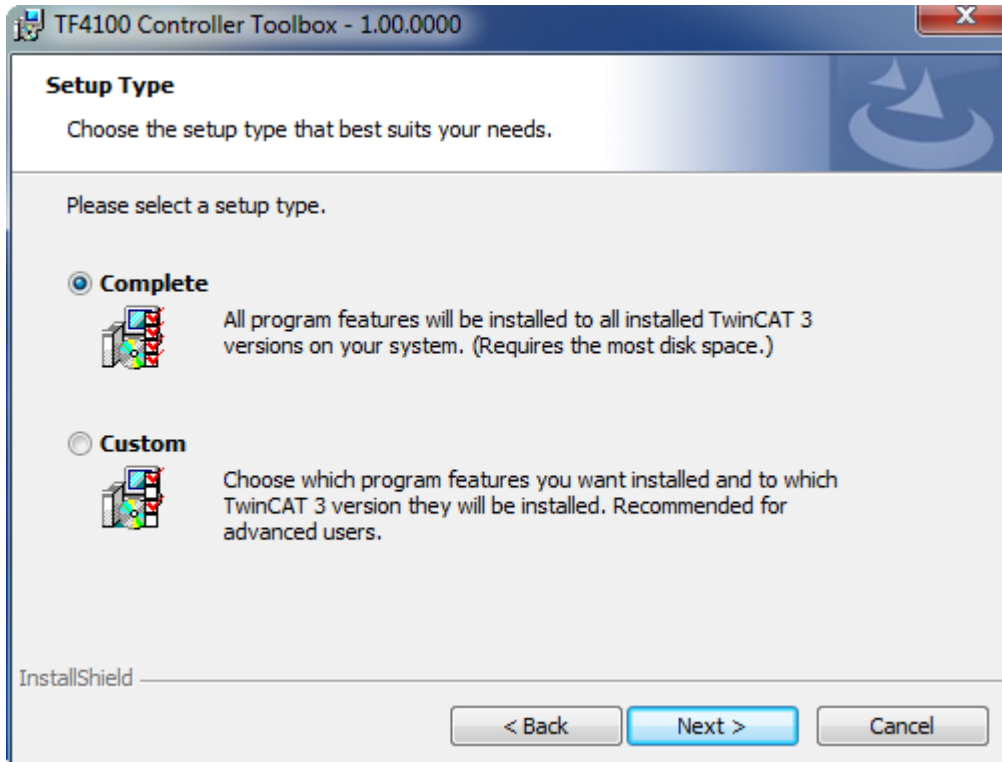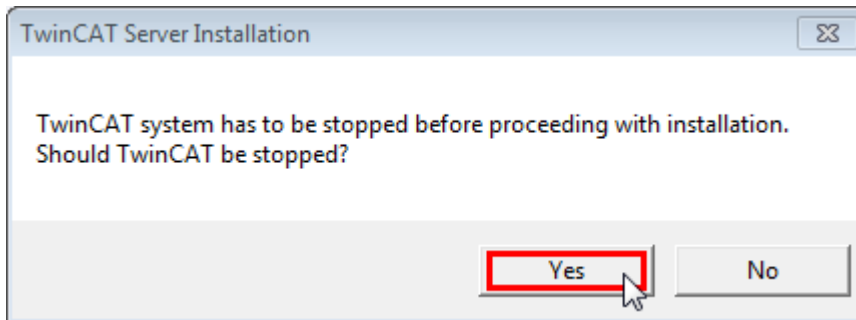

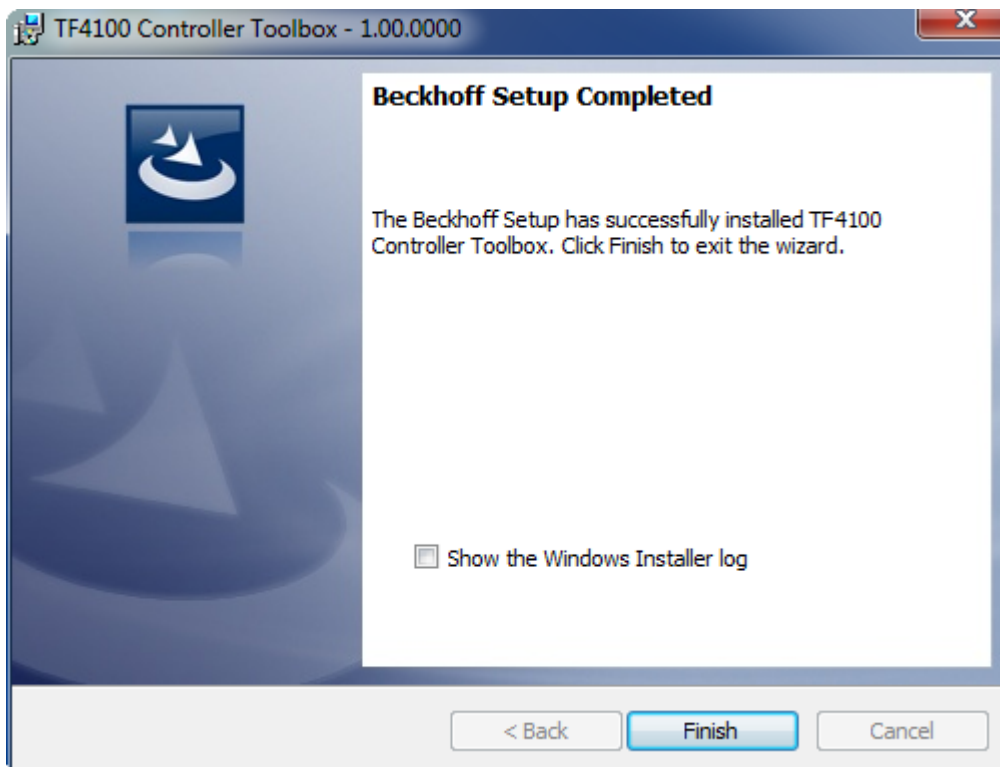5. Select **Next**, then **Install** to start the installation.

⇨ A dialog box informs you that the TwinCAT system must be stopped to proceed with the installation.

6. Confirm the dialog with **Yes**.

7. Select **Finish** to exit the setup.

⇨ The TwinCAT 3 Function has been successfully installed and can be licensed (see Licensing [▶ 14]).

3.4   After the Installation

The next step after successful installation is licensing of the products "TE130x Scope View" and/or "TF3300 Scope Server". By default the product level "Base" is active, which is available without a license. The advanced functionality can be enabled through corresponding licensing, e.g. for the product level "Professional". Further steps then are:

- First steps with TwinCAT 3 Scope [▶ 153]
- Detailed documentation of the individual functions [▶ 21]
- The integration of TwinCAT 3 Scope View Control into your own .NET-based visualization

3.5   Licensing

The TwinCAT 3 Function can be activated as a full version or as a 7-day test version. Both license types can be activated via the TwinCAT 3 development environment (XAE).
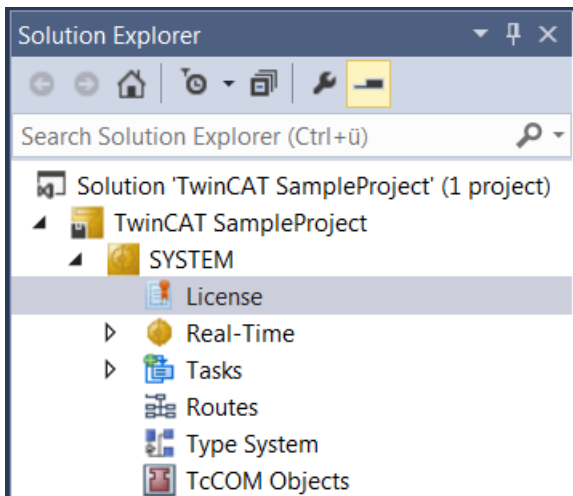
The licensing of a TwinCAT 3 Function is described below. The description is divided into the following sections:

Further information on TwinCAT 3 licensing can be found in the "Licensing" documentation in the Beckhoff Information System (TwinCAT 3 > Licensing).

Licensing a 7-day test version

1. Start the TwinCAT 3 development environment (XAE).
2. Open an existing TwinCAT 3 project or create a new project.
3. If you want to activate the license for a remote device, set the desired target system. To do this, select the target system from the **Choose Target System** drop-down list in the toolbar.

   ⇨ The licensing settings always refer to the selected target system. When the project is activated on the target system, the corresponding TwinCAT 3 licenses are automatically copied to this system.
4. In the **Solution Explorer**, double-click **License** in the **SYSTEM** subtree.

   ⇨ The TwinCAT 3 license manager opens.
5. Open the **Manage Licenses** tab. In the **Add License** column, check the check box for the license you want to add to your project (e.g. "TF6420: TC3 Database Server").

6. Open the **Order Information (Runtime)** tab.

⇨ In the tabular overview of licenses, the previously selected license is displayed with the status "missing"**.**
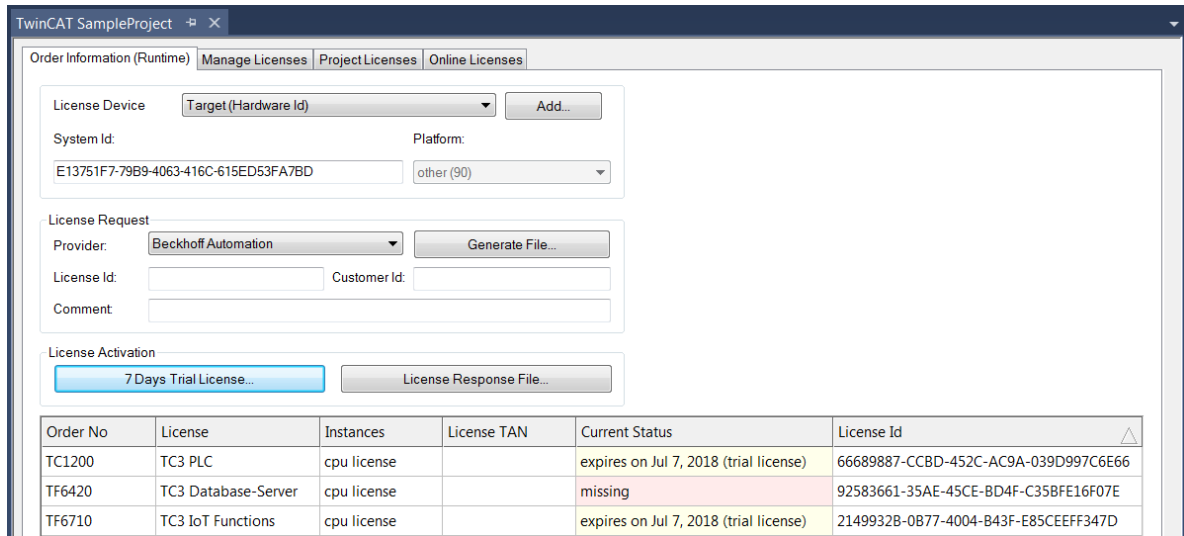
7. Click **7-Day Trial License...** to activate the 7-day trial license.
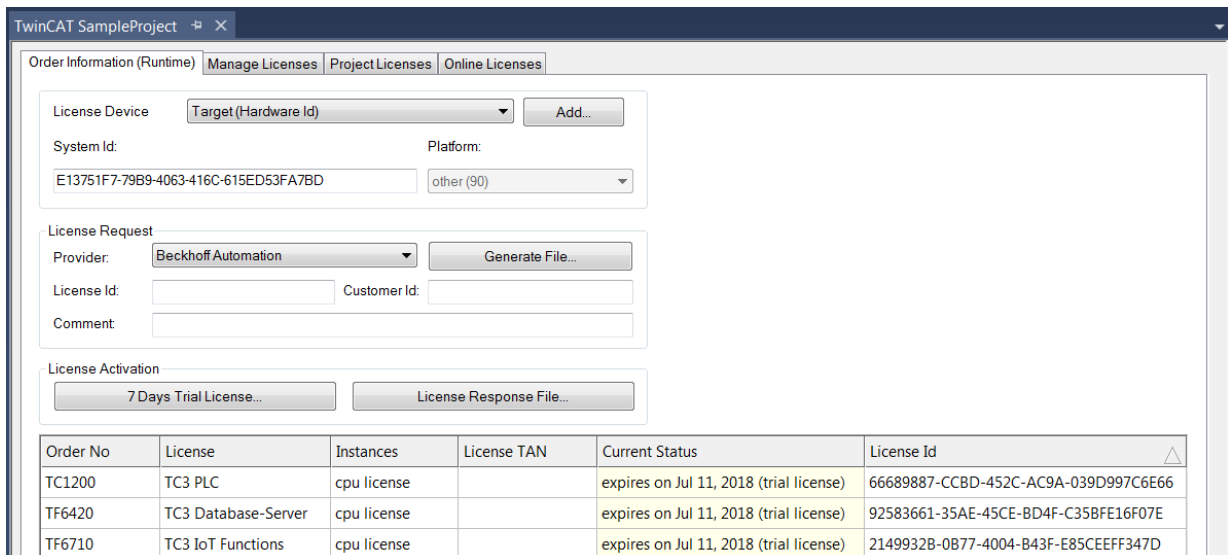
⇨ A dialog box opens, prompting you to enter the security code displayed in the dialog.

8. Enter the code exactly as it appears, confirm it and acknowledge the subsequent dialog indicating successful activation.

⇨ In the tabular overview of licenses, the license status now indicates the expiration date of the license.

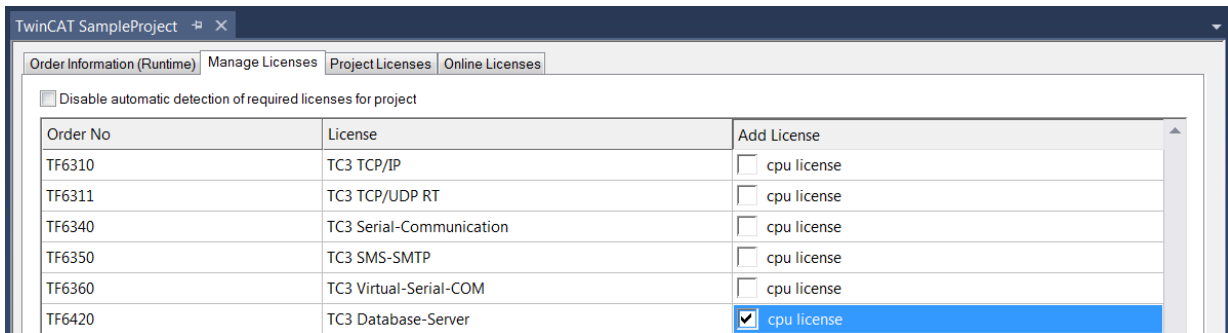9. Restart the TwinCAT system.

⇨ The 7-day trial version is enabled.

Licensing a full version

1. Start the TwinCAT 3 development environment (XAE).

2. Open an existing TwinCAT 3 project or create a new project.

3. If you want to activate the license for a remote device, set the desired target system. To do this, select the target system from the **Choose Target System** drop-down list in the toolbar.

⇨ The licensing settings always refer to the selected target system. When the project is activated on the target system, the corresponding TwinCAT 3 licenses are automatically copied to this system.

4. In the **Solution Explorer**, double-click **License** in the **SYSTEM** subtree.


⇨ The TwinCAT 3 license manager opens.

5. Open the **Manage Licenses** tab. In the **Add License** column, check the check box for the license you want to add to your project (e.g. "TE1300: TC3 Scope View Professional").
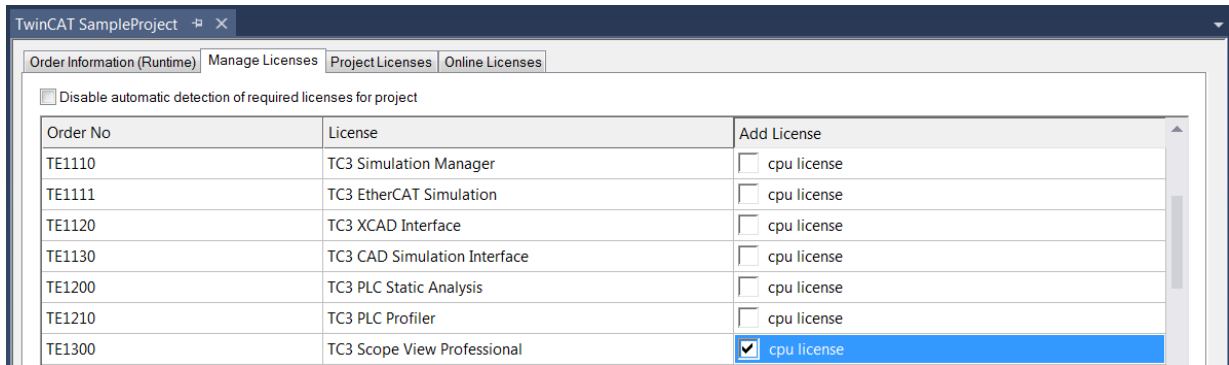

6. Open the **Order Information** tab.

⇨ In the tabular overview of licenses, the previously selected license is displayed with the status "missing"**.**
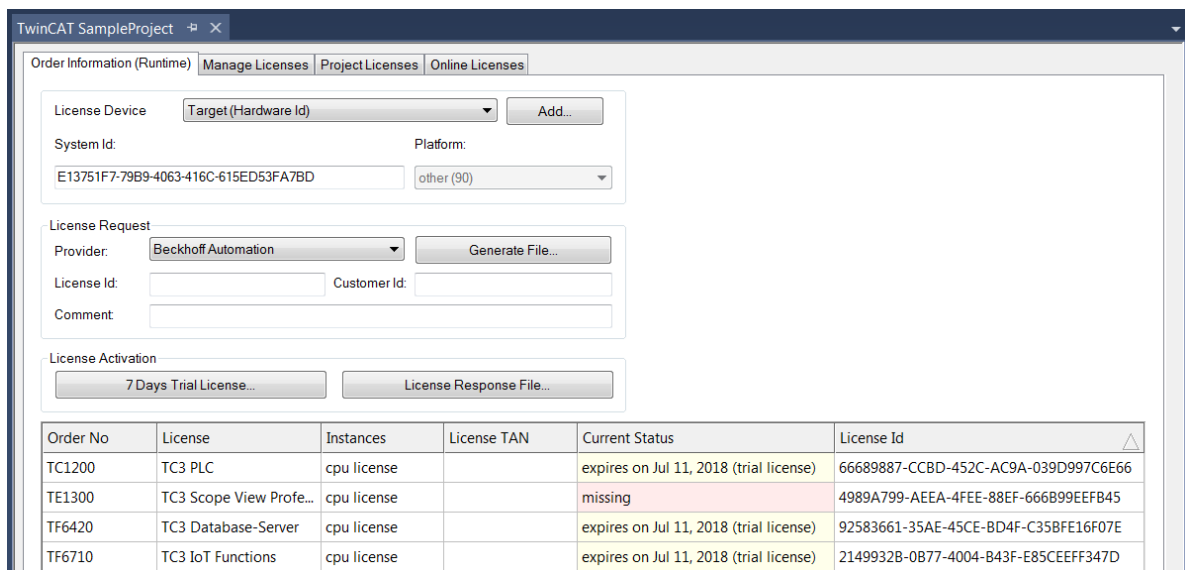
A TwinCAT 3 license is generally linked to two indices describing the platform to be licensed:
System ID: Uniquely identifies the device
Platform level: Defines the performance of the device
The corresponding **System Id** and **Platform** fields cannot be changed.

7. Enter the order number (**License Id**) for the license to be activated and optionally a separate order number (**Customer Id**), plus an optional comment for your own purposes (**Comment**). If you do not know your Beckhoff order number, please contact your Beckhoff sales contact.

Order No	License	Instances	License TAN	Current Status	License Id
TC1200	TC3 PLC	cpu license		expires on Jul 11, 2018 (trial license)	66689887-CCBD-452C-AC9A-039D997C6E66
TE1300	TC3 Scope View Profe...	cpu license		missing	4989A799-AEEA-4FEE-88EF-666B99EEFB45
TF6420	TC3 Database-Server	cpu license		expires on Jul 11, 2018 (trial license)	92583661-35AE-45CE-BD4F-C35BFE16F07E
TF6710	TC3 IoT Functions	cpu license		expires on Jul 11, 2018 (trial license)	2149932B-0B77-4004-B43F-E85CEEFF347D

8. Click the **Generate File**... button to create a License Request File for the listed missing license.

   ⇨ A window opens, in which you can specify where the License Request File is to be stored. (We recommend accepting the default settings.)

9. Select a location and click **Save**.

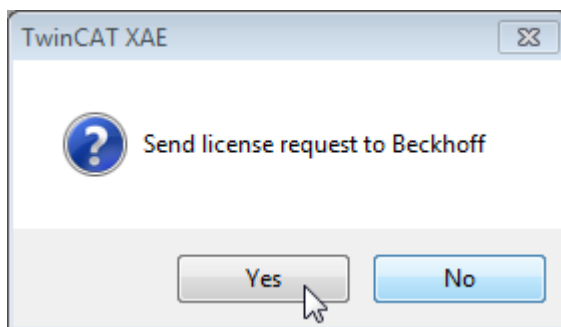   ⇨ A prompt appears asking whether you want to send the License Request File to the Beckhoff license server for verification:

- Click **Yes** to send the License Request File. A prerequisite is that an email program is installed on your computer and that your computer is connected to the internet. When you click **Yes**, the system automatically generates a draft email containing the License Request File with all the necessary information.

- Click **No** if your computer does not have an email program installed on it or is not connected to the internet. Copy the License Request File onto a data storage device (e.g. a USB stick) and send the file from a computer with internet access and an email program to the Beckhoff license server (tclicense@beckhoff.com) by email.

10. Send the License Request File.

   ⇨ The License Request File is sent to the Beckhoff license server. After receiving the email, the server compares your license request with the specified order number and returns a License Response File by email. The Beckhoff license server returns the License Response File to the same email address from which the License Request File was sent. The License Response File differs from the License Request File only by a signature that documents the validity of the license file content. You can view the contents of the License Response File with an editor suitable for XML files (e.g. "XML Notepad"). The contents of the License Response File must not be changed, otherwise the license file becomes invalid.

11. Save the License Response File.

12. To import the license file and activate the license, click **License Response File...** in the **Order Information** tab.

13. Select the License Response File in your file directory and confirm the dialog.
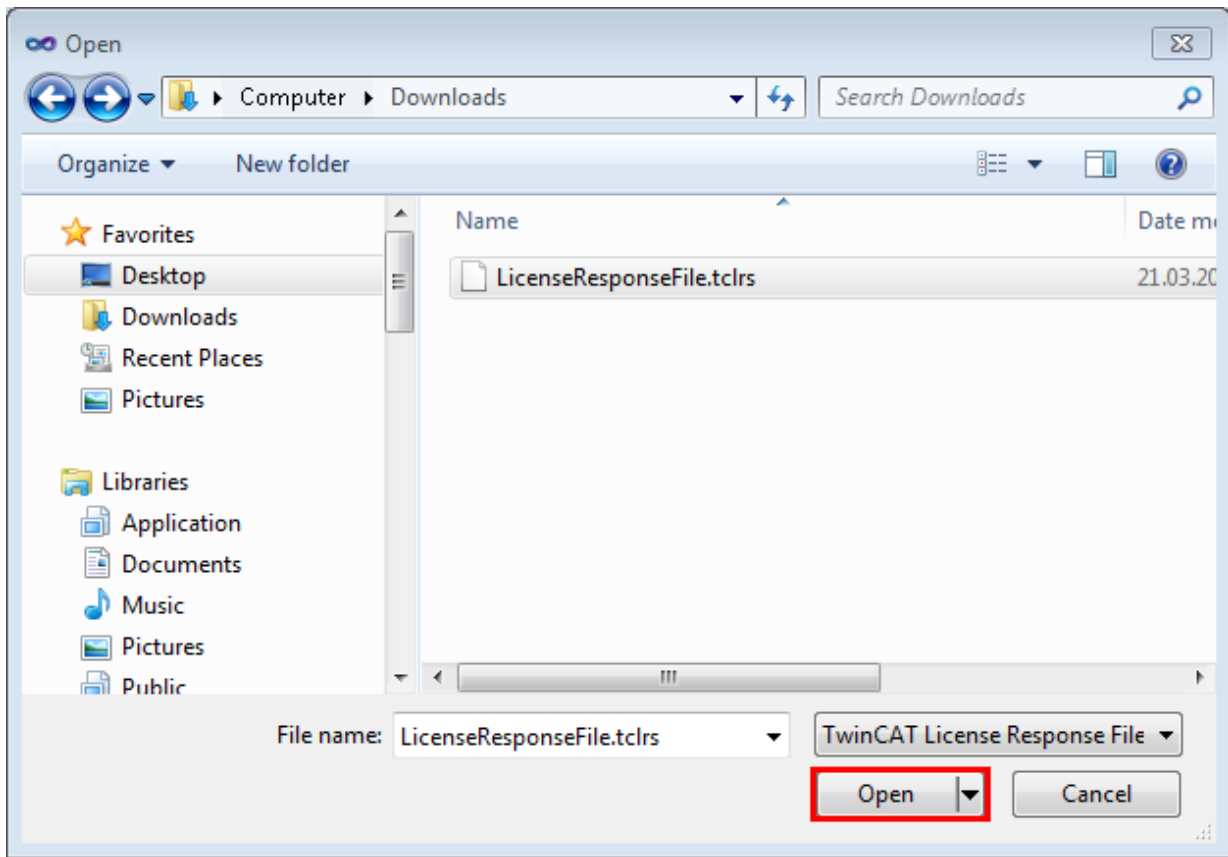
⇨ The License Response File is imported and the license it contains is activated. Existing demo licenses will be removed.

14. Restart the TwinCAT system.

⇨ The license becomes active when TwinCAT is restarted. The product can be used as a full version. During the TwinCAT restart the license file is automatically copied to the directory ...*TwinCAT\3.1\Target \License* on the respective target system.

4   Technical introduction

4.1   Basic concept

The TwinCAT Scope is divided into two main components: the Scope View and the Scope Server. The Scope Server performs the actual data recording. It can connect to control systems via ADS or OPC UA and record data. The configuration necessary for this is done in the Scope View. Apart from the configuration, the recorded data are also graphically displayed there in various types of chart.
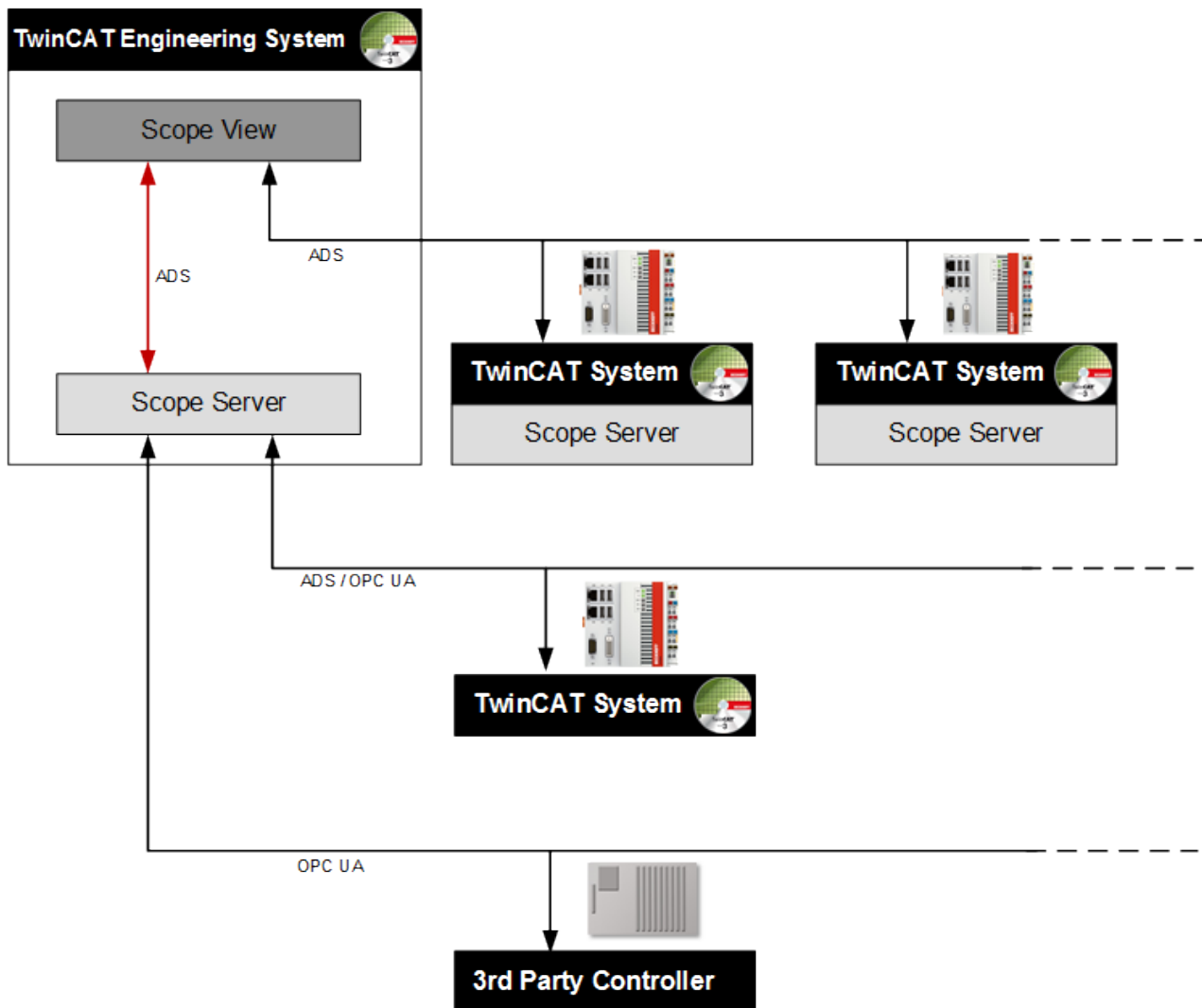
The Scope View normally runs on the engineering systems of the service or commissioning technicians or on machine computers intended to prepare recorded data graphically for the machine personnel. It is therefore intended for commissioning and also for the continuous monitoring of machines and plants.

An ADS connection is necessary in order to be able to start a recording with a Scope View on a Remote Scope Server. A disconnection is also possible from version 3.3.3140.0. This means that the configuration is downloaded to the server and continues to run autonomously there. Alternatively the remote server can be controlled by a PLC function block. If no Scope Server is installed on the control system, the Scope Server on the device with the Scope View –the so-called local server –can also be used. For this there must be a constant connection to the remote device during the recording.

Apart from many properties in the Scope View for the graphic display and analysis of the data streams, the Scope also offers the option of triggered, i.e. event-based operation. Furthermore it is possible to export the recorded data in various file formats to external tools for further processing.

5 Configuration

5.1 Architecture

Not only are signal curves represented in the TwinCAT 3 Scope View; recording configurations are also created. For the creation of these configurations it is important to be familiar with the architecture of the TwinCAT 3 Scope View. This is reflected in the tree structure of the measurement project in the **Solution Explorer.**

|	
● Measurement Scope Project	Main level, at which several Scopes can be added. The Scopes within a project can be controlled independently of one another.
● Standard Scope Project	A Scope always stands for a recording configuration. This means that all elements inserted below it are subject to the same recording settings. If a Scope is selected, the setting options such as recording time and ring buffer are displayed in the Visual Studio **Properties** window. See also: Scope nodes [▶ 23]
● DataPool	All raw acquisitions whose data are to be recorded are located in the DataPool. Whether they are displayed in the View as graphs depends on the configuration of the charts. A distinction is made between raw acquisitions and interpreted acquisitions
● YT Chart	Several YT charts can exist in parallel in a Scope. They are the actual display area in the view and provide the timebase. Each chart has its own toolbar for changing the display. The color and axis settings can be made in the **Properties** window. Clicking on the signal curve in the chart highlights the respective channel in the **Solution Explorer**. See also: YT chart properties [▶ 27]

	● Axis	A chart (YT, XY, array or single bar) can have several axes. An axis provides the range of values for the connected channels. Amongst others, the automatic or free scaling can be set in the **Properties** window. See also: YT axis properties [▶ 31]
	● Channel	A channel represents the style characteristics of a selected variable. In the **Properties** window the color, marks and other parameters can be set. Double-clicking on the channel highlights the respective signal in the chart.
	● Acquisition	The acquisition contains information on the actually selected variables.
	● Marker	Markers are hierarchically assigned to the charts. X and Y markers can be added within a chart. Amongst other things, the current values of the signal/ marker intersection and differences to the other markers are displayed in the marker tool window. Any desired number of X and Y markers can be set. See also: Markers [▶ 70]
	● Dynamic Style	With the help of Dynamic Styles [▶ 73], the style properties such as graph thickness or graph color can be changed at runtime based on the states of the same or other acquisitions.
● XY Chart		Several XY charts can exist in parallel in a Scope. They represent the actual display area in the view. Each chart has its own toolbar for changing the display. The color and scaling settings can be made in the Properties window. Clicking on the signal curve in the chart highlights the respective channel in the **Solution Explorer**. See also: XY chart properties [▶ 37]
	● Shapes	Shapes [▶ 76] are geometric figures that can be placed in the charts. The figures can be individualized accordingly through the specification of coordinates.
● Array Bar Chart		An array bar chart can connect to an array in the controller and display each array element as a bar. Several array bar charts can exist in parallel in a Scope. The settings for the chart can be entered in the **Properties** window.
● Single Bar Chart		The bar chart displays a single variable as a bar diagram. The settings [▶ 56] for this chart type can be made in the Property window.
● Trigger		Triggers are assigned to the Scopes in the tree structure of the Scope View. The trigger action, e.g. a Stop Record, can be set in the Properties window of the Trigger Group. The lower-level triggers can be logically linked to form a trigger condition. The variable selection also takes place here in the **Properties** window. See also: Trigger properties [▶ 64]

The interfaces for the control of the Scope View are divided into several individual windows (Tool windows) and their position and size are freely configurable.
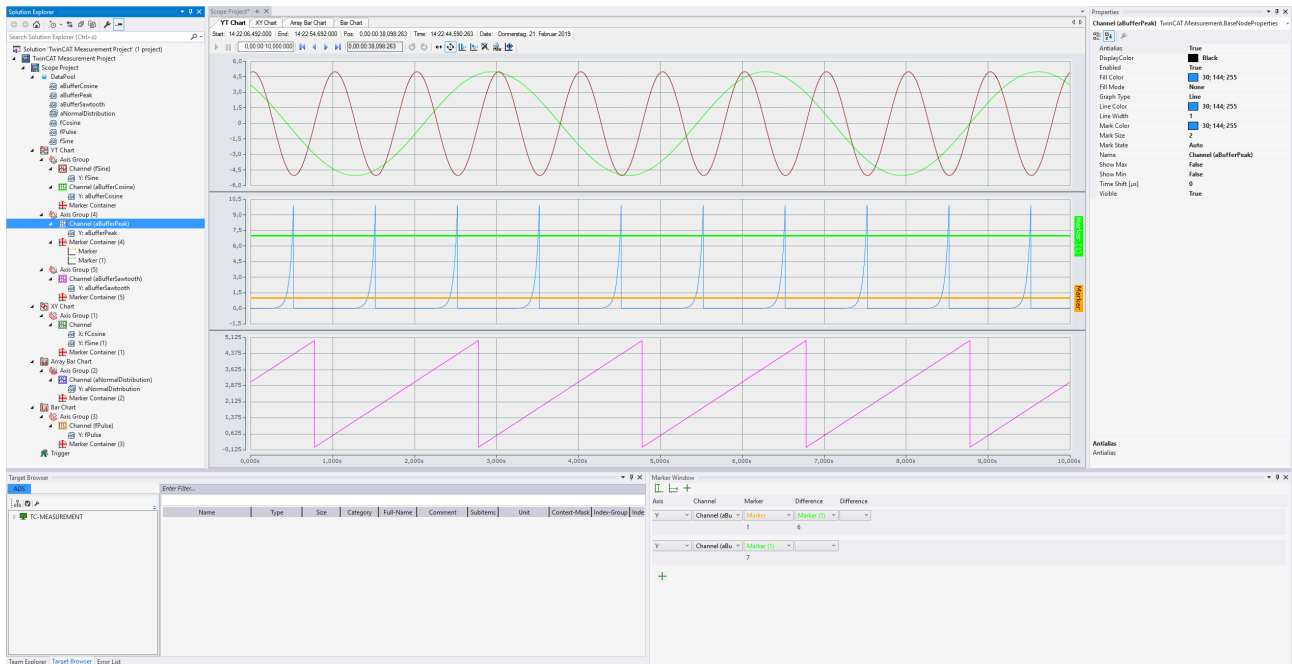
Solution Explorer	Display of the project structure within a solution.
	See also: Architecture [▶ 21]
Markers	Display of the values present at the X/Y-marker.
	See also: Markers [▶ 70]
Target Browser	Channels can be added to a Scope configuration via their symbol names using the Target Browser.
	See also: Target Browser [▶ 104]
Properties	The settings of the element marked in the Solution Explorer are displayed in the Properties and can be edited.
	See also: Scope nodes [▶ 23]
Error list	List of errors, warnings and messages.
	Each scope project lists the generated messages independently here. The messages for the respectively selected Scope can be deleted via the context menu command **Clear Error List**.
ScopeViewControl	Display of the individual charts.
	The charts can be displayed next to each other or in overlapping tabs within the control, exactly like all other windows.

5.1.1 Principal elements

5.1.1.1 Scope nodes

The Scope node in the Solution Explorer is a type of administration shell for the Scope configuration. Several Scope nodes can be created in a measurement project. A Scope contains the charts, axes and channels of a configuration. With selected Scope nodes the higher-level properties are displayed in the Property window. These fulfill the following functions.

BECKHOFF
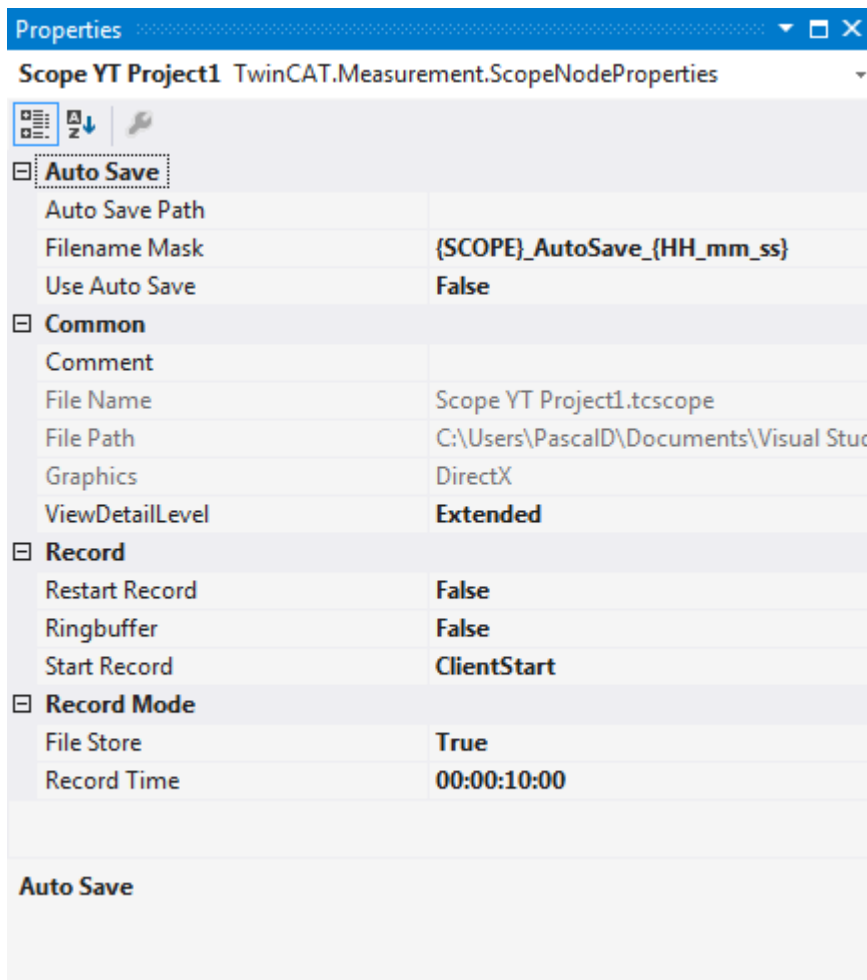
Auto Save

Auto Save Path	Here you can select the path for saving the recording via the file browser.
Filename Mask	Affects the filename of the recording to be saved. In this way it is possible to realize continuous time stamps, for example.
Use Auto Save	Turns on automatic saving for a stopped recording.

Common

Comment	A free comment can be stored here.
File Name	Filename of the current Scope instance.
File Path	Directory in which the Scope instance was saved.
Graphics	Shows the currently used graphic. A distinction is made between GDI+ and DirectX.
ViewDetailLevel	Here you can set the level of detail for showing or hiding various options. For a better overview, the Extended level is particularly recommended for XY plots. The option is therefore only available for XY plots at present. The options "Default" and "Extended" are available for all chart types.

Record

Restart Record	Here you can restart a stopped recording.
Ring buffer	Here you can specify how the server should respond, once the recording time has been reached.
	• If the option is disabled, the recording is stopped when the recording time is reached. The recording can be terminated prematurely via the stop button.
	• If the option is enabled the recording is not stopped, but the oldest data in the memory is overwritten. This increases the start time of the recording. The recording can be stopped by pressing the stop button.
Start Record	Here you can set whether the recording should be started via the recording button (UserStart) or when a trigger event occurs (TriggerStart).

Record Mode

File Store	Here you can specify whether the server should buffer the data in a local file (True) or only in the RAM (False). The choice depends on the quantity of data to be recorded and the recording equipment. The access time is faster, if only the RAM is used. For larger data quantities it is usually necessary to activate the "File Store" option.
Record Time	Here you can specify the total recording time. It is entered in the format Days:Hours:Minutes:Seconds.

Changing the settings of several Scopes at the same time

To change the settings for several Scopes at the same time, please refer to the chapter <u>Multiple selection</u> [▶ 161].

5.1.1.2   Data pool

The data pool is a container for project acquisitions. All acquisitions created for the project are placed in the data pool. Acquisitions that are enabled at the start of a recording are recorded and can be referenced in the interpreter of a channel during both the configuration and the recording in order to display the data of the acquisition.

There are two types of acquisition, the array bar acquisition, which is displayed with a green accent, and other acquisitions, which are displayed with a red accent. The array bar acquisitions can only be referenced in array bar channel interpreters and not in other channel interpreters or trigger set interpreters.

Each project has a data pool that is automatically added during creation.

In order to start a recording there may not be any acquisitions with the same configuration in the data pool. Since several interpreters may point to an acquisition, acquisitions with the same configuration are not necessary.

Acquisition interpreters
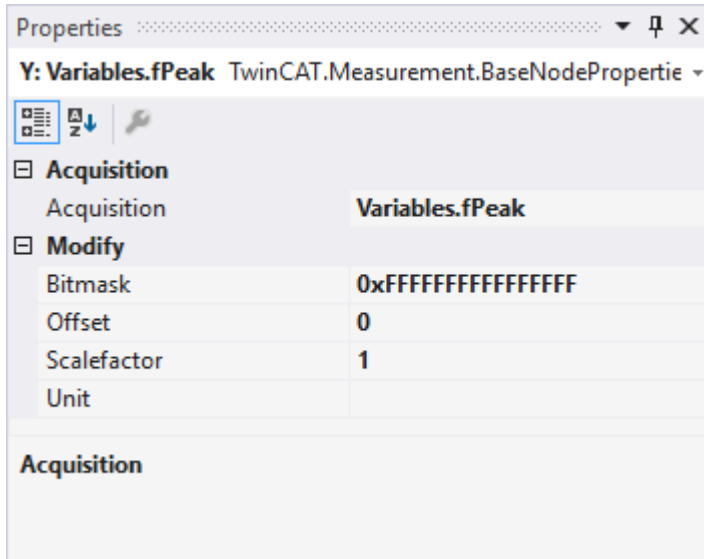
Acquisition interpreters can reference an acquisition of the data pool and forward the data after manipulation.

The following settings can be made:

Acquisition

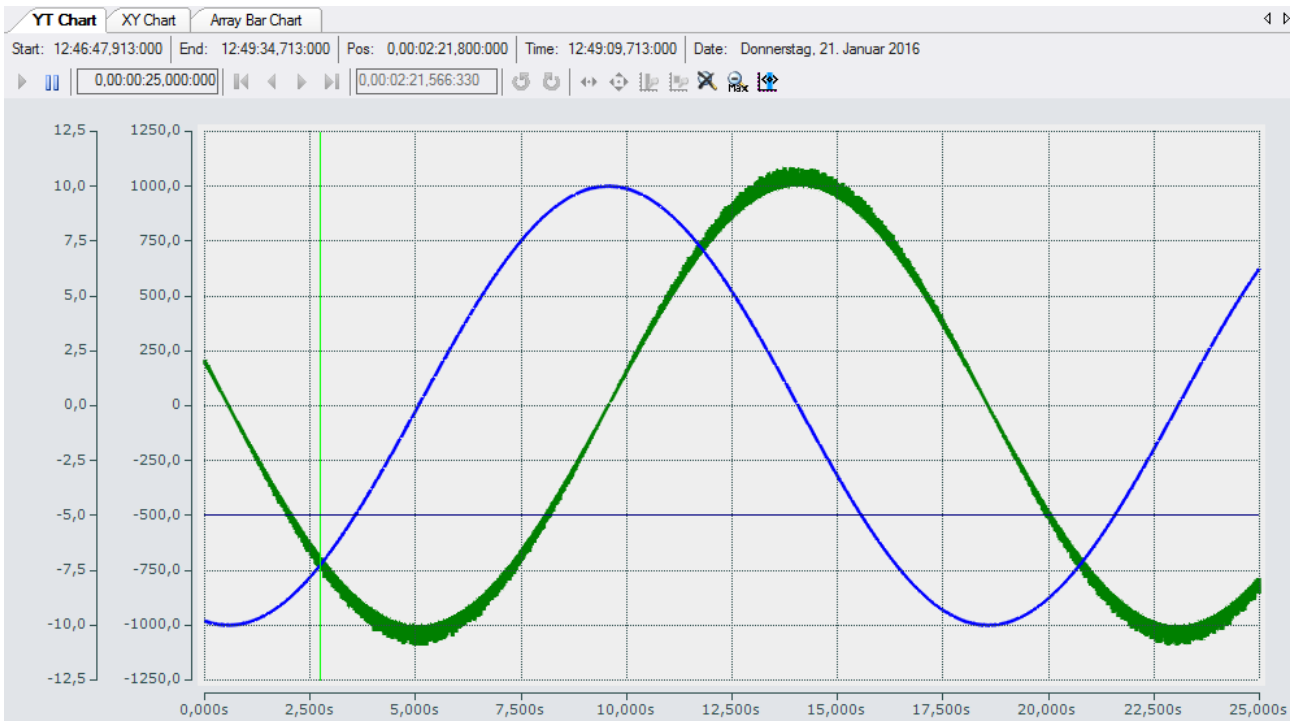| Acquisition	Selection of the acquisition used from the data pool.

Modify

Bitmask	In this field a value can be entered with which the display value is masked, provided it is not of the type floating point. This means that the value of the channel is ANDed with the binary value of the mask. This helps you, for example, to observe individual bits of a status byte.
Offset	A free offset can be added to the function value of the channel in the offset field.
Scalefactor	The display value of a channel can be changed with the scale factor.
Unit	This property enables the physical unit to be configured individually for the X-acquisition and the Y-acquisition of the channel. This can be done with the aid of the "Unit Wizard", which can be opened using the button on the right in the field. A further option is to define the units beforehand as attributes in the PLC code. These are then read out directly and entered in the Unit field. Detailed information on the units and their configuration options can be taken from the chapter [▶ 85].

5.1.1.3   Charts and channels

5.1.1.3.1 YT Chart

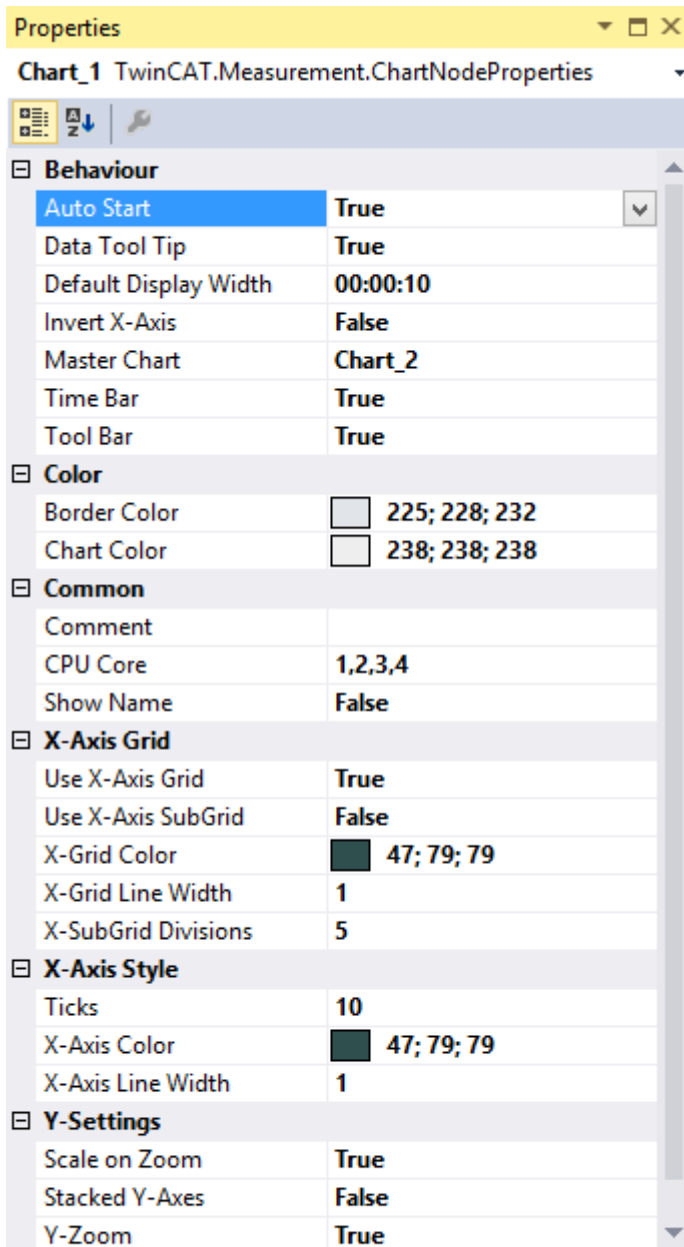All the properties of the individual hierarchy levels of YT Charts are explained below.

The View Detail level, which can be set in the Scope settings, can be used to separate Channel Style and Channel Acquisition at channel level through an additional hierarchy level. Channel Style and Channel Acquisition are described separately below.

YT chart properties

Here you can enter all the settings relating to a YT chart.

BECKHOFF

Properties ▾ □ ✕

Chart_1 TwinCAT.Measurement.ChartNodeProperties ▾

⊟ **Behaviour**	
Auto Start	**True**
Data Tool Tip	**True**
Default Display Width	**00:00:10**
Invert X-Axis	**False**
Master Chart	**Chart_2**
Time Bar	**True**
Tool Bar	**True**
⊟ **Color**	
Border Color	225; 228; 232
Chart Color	238; 238; 238
⊟ **Common**	
Comment	
CPU Core	1,2,3,4
Show Name	**False**
⊟ **X-Axis Grid**	
Use X-Axis Grid	**True**
Use X-Axis SubGrid	**False**
X-Grid Color	47; 79; 79
X-Grid Line Width	1
X-SubGrid Divisions	5
⊟ **X-Axis Style**	
Ticks	10
X-Axis Color	47; 79; 79
X-Axis Line Width	1
⊟ **Y-Settings**	
Scale on Zoom	**True**
Stacked Y-Axes	**False**
Y-Zoom	**True**

Auto Start

Auto Start

Behaviour

Auto Start	If this option is enabled, the chart starts the live display when a new recording is started.
Data Tool Tip	If this option is enabled, a click on a data point results in display of a tool tip with the exact values of the data point on the X- and Y-axis, plus the timestamp and name of the corresponding channel.
Default Display Width	This time specifies the standard width of the associated chart. This value is set, for example, when the rescale button is selected in the chart toolbar.
Invert X-Axis	Switches the signal sequence from the default (left to right) to right to left
Master Chart	In the drop-down list that is available here, a chart that is part of the current configuration can be selected and designated as master chart. All actions that are executed in the master chart are also automatically executed in this chart. A separate stop display option is provided, through which Y-zoom and Y-panning modes are available. An overview chart can also be displayed.

| Time Bar | Specifies whether the time bar is displayed in the chart. |
| Tool Bar | Specifies whether the toolbar is displayed in the chart. |

Color

The background and frame color can be set by means of color dialogs.

Common

Comment	A free comment can be saved here.
CPU Core	Here you can set (for each chart individually) which CPU core should be used for the current chart display. If required, multiple CPU cores can be specified for a chart. This may improve the performance.
Show Name	Specifies whether the name of the chart is shown in the graph.

X-Axis Grid

Use X-Axis Grid	The X-subdivisions in the chart can be switched on or off here.
Use X-Axis SubGrid	Auxiliary lines for the finer X-subdivision of the main grid can be shown or hidden here.
X-Grid Color	Color of the grid
X-Grid Line Width	Line width of the grid in pixels
X-SubGrid Divisions	Number of areas into which the main grid is subdivided by auxiliary lines.

X-Axis Style

Since the X-axis is permanently assigned to a chart, all associated settings can be made here.
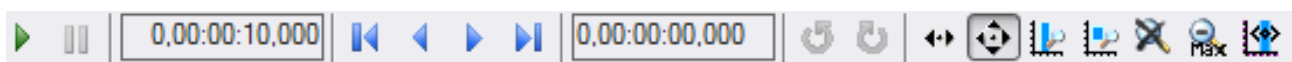
Ticks	Number of subdivisions
X-Axis Color	Color of the X axis
X-Axis LineWidth	Line width of the axis in pixels

Y-Settings

Scale on Zoom	If this option is selected, the chart instructs all axes (X and Y) to perform an auto-scaling after a zoom or panning action. This can be used to expand the selected area.
Stacked Y-Aches	This setting can be used to specify whether the axes of a chart should be displayed side by side, so that the values of the attached channels are shown within the same area, or whether the axes should be above each other, each with its own display area.
Y-Zoom	Here you can set the chart behavior during zooming. If the option is set, you can navigate in the data of a chart in the Y-direction as well. This includes both zooming and panning (shifting the display with the mouse).

To change the settings for several charts at the same time, please refer to the chapter .
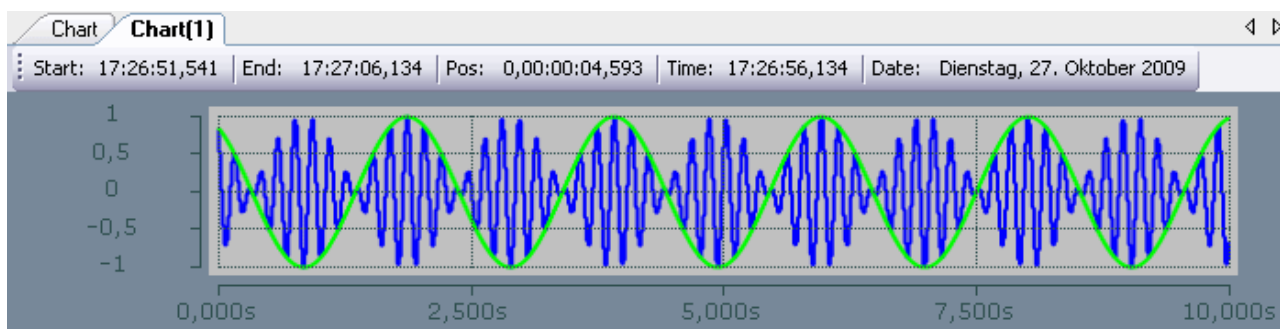
The chart display is operated with the toolbar. The overview shows all buttons and their explanation (from left):

Play	Starts the live display mode. The data currently being accumulated are displayed.
Break	The display switches to the pause mode. You can now navigate in the data already recorded without stopping the recording.
Display-Width	The current display width is displayed here. It can be edited in the format hh:mm:ss,fff. The zoom function works down to the µs range. Alternatively, the display width can be changed by turning the mouse wheel in the selected chart window. The changed value is adopted automatically.
Scroll buttons	The outer scroll buttons move the current display in steps that correspond to the display width. The inner scroll keys move the display only by a tenth of the display width and can be kept pressed to view the data set.
Position	Shows the position. It can be edited in the format hh:mm:ss,fff. The colons are used as separators. If not all units are edited the format is sorted in ascending order, starting with seconds.
Undo/Redo Time/ Position	This option can be used to undo step changes in the display width or the current position, irrespective of how they were made (e.g. zoom, scroll, etc.). The right mouse button is likewise assigned this function. Once undone, values can be repeated with redo.
Panning Horizontal	In the horizontal panning mode the current display can be shifted along the x-axis by clicking and dragging with the mouse.
Panning Free	In the free panning mode the current display can be shifted along the x- and y-axes by clicking and dragging with the mouse.
Zoom Horizontal	A new time range for the display can be selected by stretching a rectangle over the x-axis.
Zoom Free	You can zoom into the current display by stretching a rectangle over the graphic area.
Zoom to Default	Carries out an autoscale on all axes. The x-axis is/axes are set to the default display time.
Zoom Out Max	Scales the x-axis in such a way that all current values in the recording appear in the display.
Overview	Use the Overview option to display a chart within the chart. The signal range currently shown in the main chart is highlighted in the Overview Chart. The Overview Chart also offers an absolute time axis for the whole recording time.

By turning the mouse wheel you can also zoom in or out on the current display. The cursor position of the mouse determines the center.

The current recording times are displayed in the chart toolbar:

Start-Time	The common starting point of the recordings of all connected channels. The start time defines the zero point of the recording.
End-Time	Maximum common time of all connected channels. The end time thus marks the final value of the recording. The difference between the end time and start time is maximally as large as the record time set (see Scope nodes [▶ 23]).
Position	The position time represents the zero point of the current chart, i.e. the time from the start time to the beginning of the display.
Time	Absolute time at the chart origin
Date	Absolute date at the chart origin

YT axis properties

Here you can implement separate settings for each available YT axis.
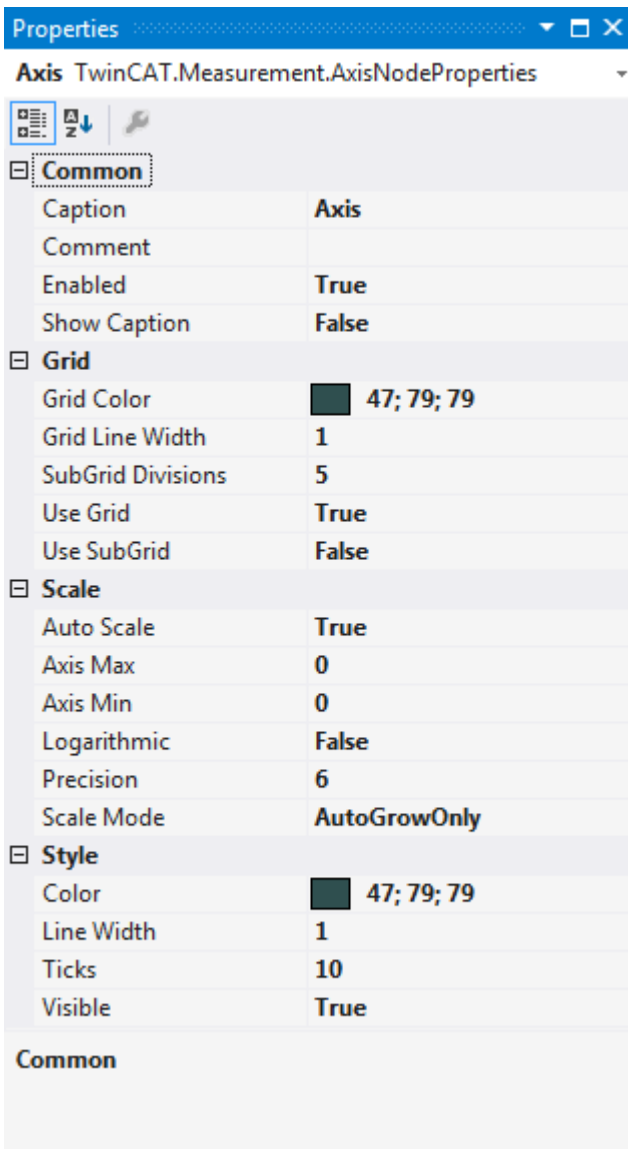
Common

Caption	Here you can enter an axis label.
Comment	A free comment can be saved here.
Enabled	Here you can switch the axis on or off completely. This functionality is also available via the context menu.
Show Caption	Defines whether the name of the chart (Caption) is shown in the graph.

Grid

Grid Color	Color of the grid
Grid Line Width	Line width of the grid in pixels
SubGrid Divisions	Number of areas into which the main grid is subdivided by auxiliary lines. In the case of logarithmic scaling the auxiliary lines for subdivision are shown only in the set number if the range of values per tick is precisely one decade. Otherwise the displayed auxiliary line shows the decades not displayed in the main grid.
Use Grid	The Y-subdivisions of the axis can be shown or hidden here.
Use SubGrid	Auxiliary lines for the finer Y-subdivision of the main grid can be shown or hidden here. Auxiliary lines have no axis labeling.

Scale

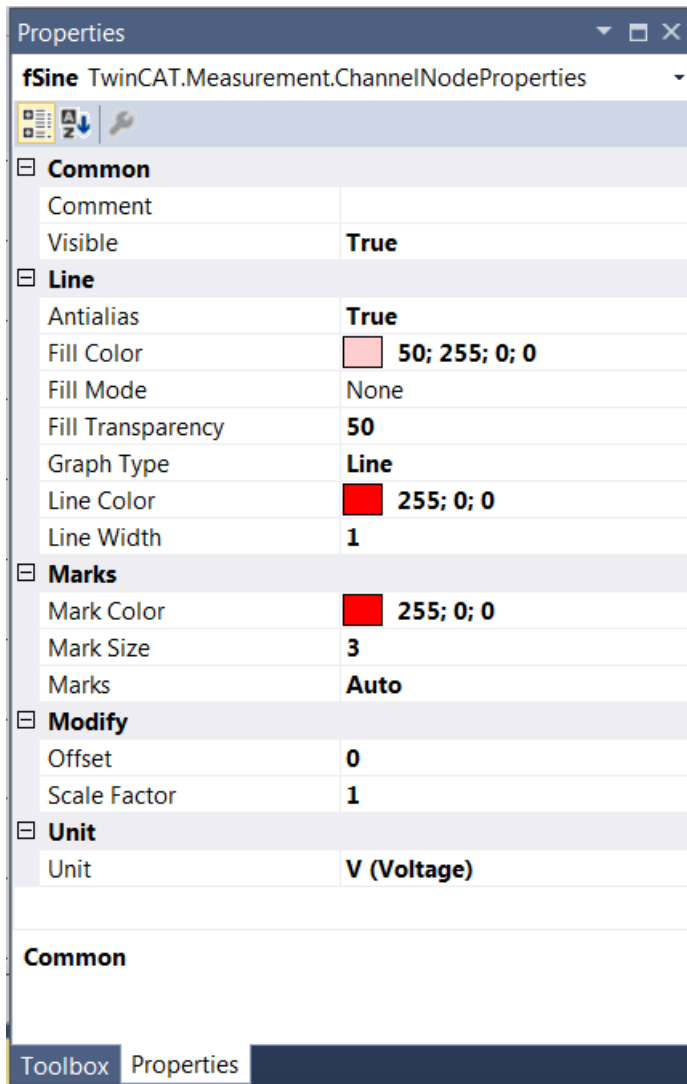| Auto Scale	If Auto Scale is active, each axis is scaled such that the last-known minimum and maximum of all connected channels lie within the display range.
Axis Max	If Auto Scale is inactive, the maximum value can be edited directly.
Axis Min	If Auto Scale is inactive, the minimum value can be edited directly.
Logarithmic	Switches between logarithmic and linear scaling of the axes.
Precision	Here you can specify the number of significant digits for the axis labeling. It corresponds to the number of visible digits on the axis.
Scale Mode	Two auto-scaling modes are available for selection. The default mode is AutoGrowOnly. This means the y-axis always scales to the historic maximum, until a rescale is carried out for the current view. Alternatively, AutoGrowNShrink can be selected. This mode always automatically adjusts the maximum of the y-axis to the maximum in the current view (display width).

Axis Style

Color	Color of the axis
Line Width	Line width in pixels
Ticks	Maximum number of subdivisions. If there is not enough space for the selected number of ticks, these are automatically reduced in the chart. In the case of logarithmic scaling the number of ticks depends on the displayed range of values and can thus deviate from the settings.
Visible	Specifies whether the axis is shown or hidden in the chart.

To change the settings for several axes at the same time, please refer to the chapter Multiple selection [▶ 161].

YT channel style

All style settings belonging to a YT channel can be made here.

Common

Comment	A free comment can be saved here. If applicable the comment will be adopted from the connected ADS symbol.
Visible	Here you can set whether the channel should be displayed in the chart.

Line

Antialias	This option decides how the lines will be drawn. Antialias is "nicer" but requires considerably more computing. This has an effect in particular if there are many channels with large movements.
Fill Color	Fill mode can be used to color an area, e.g. above or below a curve. Here you can select the fill color.
Fill Mode	Fill mode can be used to color an area, e.g. above or below a curve. The default setting is "None". Variables with the data type BOOL are an exception. For BOOL variables the default value is "Horizontal Zero". Further alternatives: Bottom, Top, Center and Source.
Fill Transparency	Here you can set the transparency value of the fill color. The default value is 50.
Line Color	Graph color
Line Width	Line width of the chart. The line width 1 requires least computing.
Type	Changes the value display between line, steps and bars. The default value is line.

Marks

Mark Color	Color of the interpolation point markers
Marks	Selection of the visibility of markings: On (permanently visible) | Auto (dependent on the zoom level) | Off (markings switched off).
Mark Size	Size of the interpolation point markers

Modify

Bit Mask	In this field a value can be entered with which the display value is masked, provided it is not of the type floating point. This means that the value of the channel is ANDed with the binary value of the mask. This helps you, for example, to observe individual bits of a status byte.
Offset	A free offset can be added to the function value of the channel in the offset field.
Scale Factor	The display value of a channel can be changed with the scale factor. This makes sense if angle signals are to be displayed in degrees, rather than radians, for example. With a scaling factor of k = 360 / (2*Pi) = 57.296, therefore, degrees would be displayed instead of radians.
Time Shift	This option enables the graph to be moved along the time axis. This can be useful to compensate a known bus runtime, for example.

Unit

Unit	This property allows the physical unit of the respective channel to be configured. This can be done with the aid of the "Unit Wizard", which can be opened using the button on the right in the field. A further option is to define the units beforehand as attributes in the PLC code. These are then read out directly and entered in the Unit field. Detailed information on the units and their configuration options can be taken from the section .

To change the settings for several channels at the same time, please refer to the chapter .

Configuration of channel style properties with the aid of PLC attributes

All style properties of a channel can be set directly on a variable or a data type in the PLC with the aid of attributes. This makes it possible to always display a certain symbol with the same style properties without having to reconfigure it every time.

A PLC attribute is specified directly via the declaration of a variable or a data type in curly brackets.

```
28      {attribute 'TcScope' := '<LineWidth:3, LineColor:red>'}
29      fTriangular            : LREAL;
```

The following syntax applies to the assignment of the channel style properties:

{attribute 'TcScope' := '<Property1:value, Property2:value,…>'}
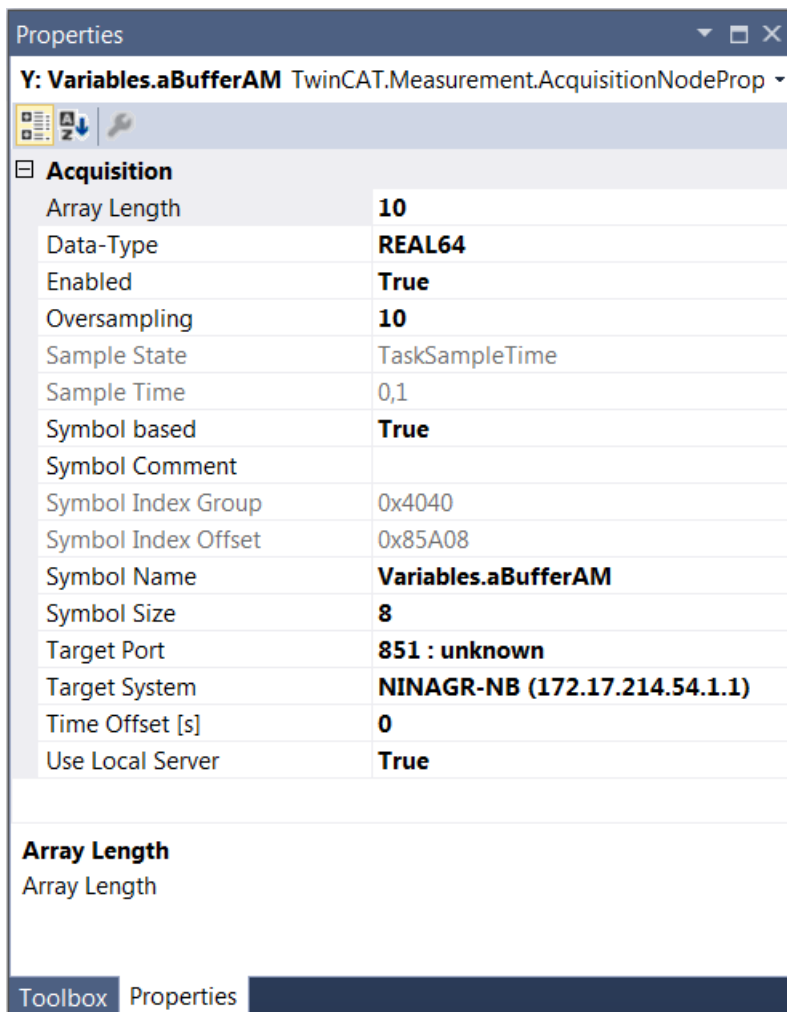
Example:
{attribute 'TcScope' := '<LineWidth:3, LineColor:red>'}

Any number of channel style properties can be specified in an attribute.

A separate syntax applies to the specification of the unit (Unit property; see:).

YT channel acquisition

All acquisition settings belonging to a YT channel can be made here.

Properties	▼ ▢ ✕
Y: Variables.aBufferAM TwinCAT.Measurement.AcquisitionNodeProp ▼	

Acquisition

Array Length	**10**
Data-Type	**REAL64**
Enabled	**True**
Oversampling	**10**
Sample State	TaskSampleTime
Sample Time	0,1
Symbol based	**True**
Symbol Comment	
Symbol Index Group	0x4040
Symbol Index Offset	0x85A08
Symbol Name	**Variables.aBufferAM**
Symbol Size	**8**
Target Port	**851 : unknown**
Target System	**NINAGR-NB (172.17.214.54.1.1)**
Time Offset [s]	**0**
Use Local Server	**True**

Array Length
Array Length

Toolbox | Properties

● **Different properties**

ⓘ The properties of the channels can deviate from the illustration shown here.
Some setting possibilities are visible / editable only for special channels.
Example: The setting **Array Length** is only visible for array symbols.

Acquisition

Array Length	Shows the number of array elements.
Data-Type	Indicates the data type of the selected variable.
Enabled	Here you can decide whether the configured channel should be recorded or not.
Oversampling	This option is only shown for arrays. The array is interpreted as a variable, and the values of the individual array elements are displayed contiguously as oversampling values. This corresponds to the behavior of EtherCAT oversampling terminals. The value that can be entered here describes the number of oversampling values to be displayed. By default, the value corresponds to the array length. See also: Oversampling recordings [▶ 163]
Sample State	Here you can specify whether the data for the selected variables are to be sampled with the TaskSampleTime or the FreeSampleTime.
Sample Time	If the sample state "FreeSampleTime" was selected, you can set the sampling rate here. It is always greater than the TaskSampleTime.
Symbol based	If this option is set, the variables are communicated via the symbol name. If this option is not set, the variables are communicated via the Group and Offset indices, and the input fields for Group and Offset are enabled.
Symbol Comment	Here the actual symbol comment is displayed, if such a comment exists.
Symbol Index Group	Group index of the variable
Symbol Index Offset	Offset index of the variable
Symbol Name	Symbol name of the selected variable
Symbol Size	Shows the size of the variables in bytes.
Target Port	Shows the variable port on the respective TwinCAT system. A drop-down list can be used to select the available ports for the selected target system.
Target System	Indicates the target system selected for a recording. By default this is the target system for the selected variable.
Time Offset [s]	This can be used for one-time manipulation of the original time stamp during recording of the data point. e.g. to compensate path differences of different target systems. The phase differences have to be determined manually by the user.
Use Local Server	If this option is set, the Scope Server installed on the Scope View system is used for the recording. If the option is not set, a connection with the remote server of the target system is established.

To change the settings for several channels at the same time, please refer to the chapter Multiple selection [▶ 161].
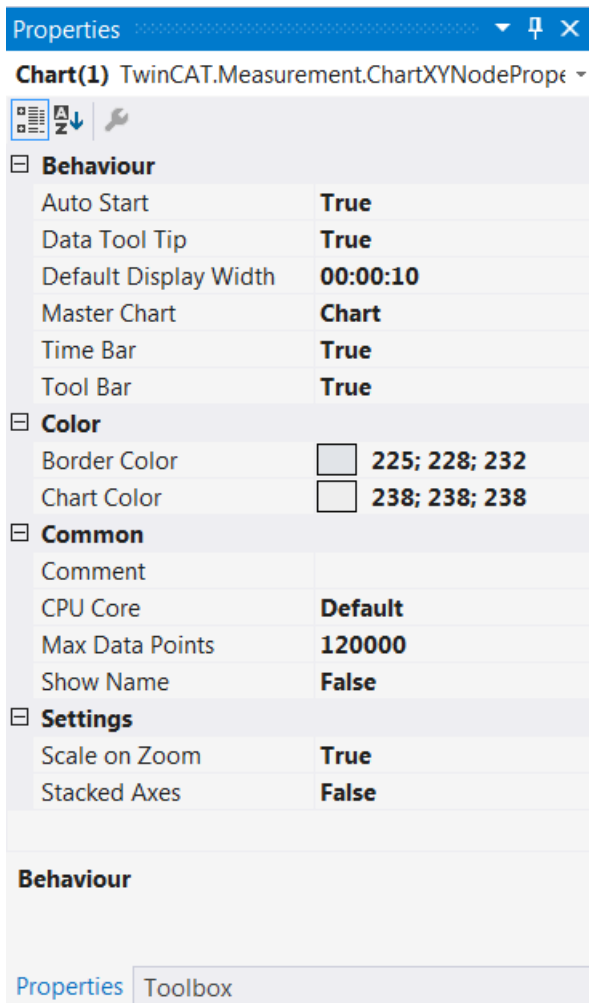
5.1.1.3.2 XY Chart

All the properties of the individual hierarchy levels of XY Charts are explained below.

The View Detail level, which can be set in the Scope settings, can be used to separate Channel Style and Channel Acquisition at channel level through an additional hierarchy level. Channel Style and Channel Acquisition are described separately below.
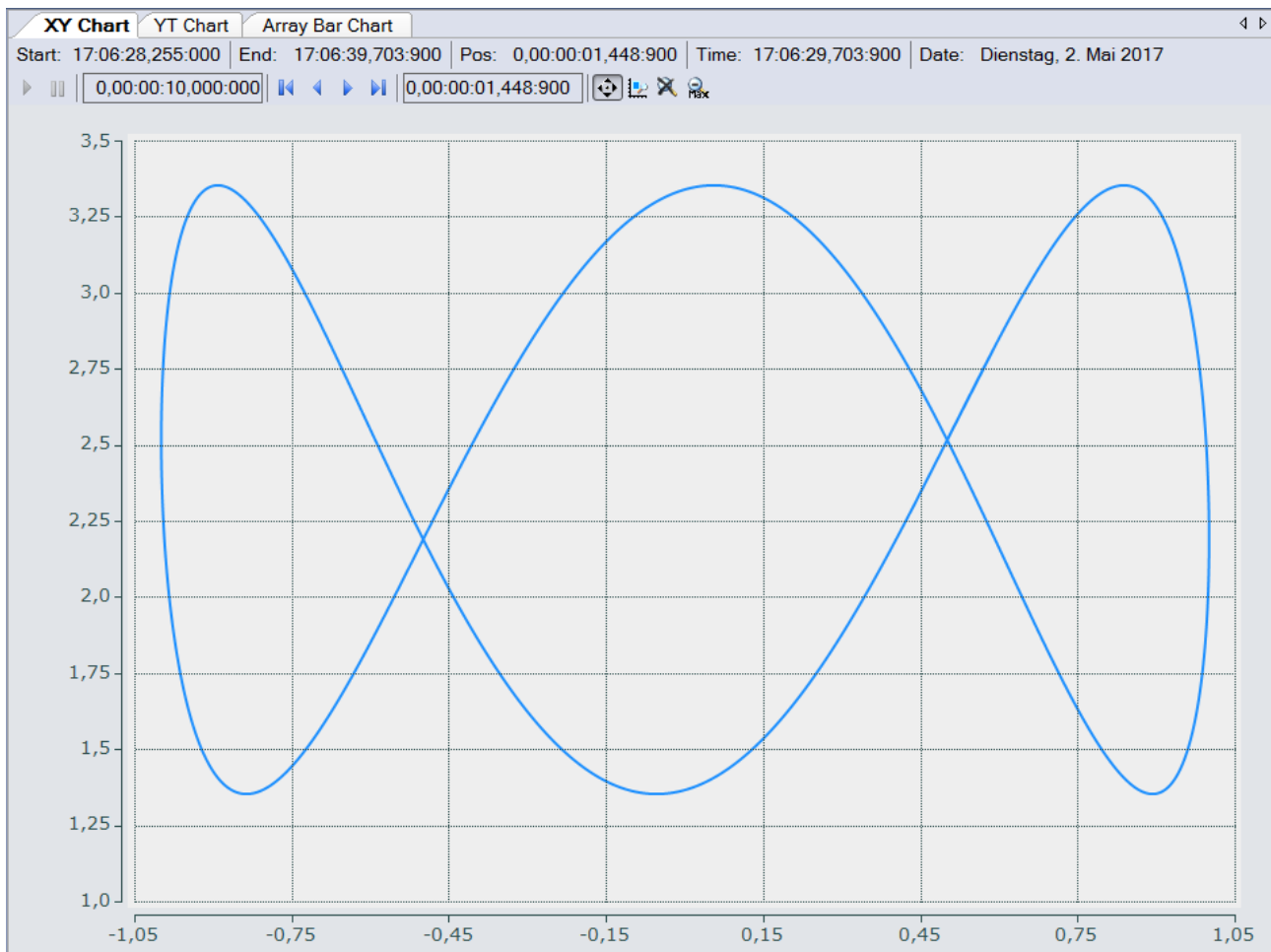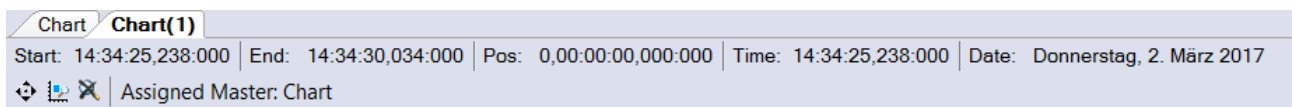
XY chart properties

All settings assigned to the XY chart can be made here.

Properties ▼ 📌 ✕

Chart(1) TwinCAT.Measurement.ChartXYNodePrope ▼

⊟ **Behaviour**
Auto Start	**True**
Data Tool Tip	**True**
Default Display Width	**00:00:10**
Master Chart	**Chart**
Time Bar	**True**
Tool Bar	**True**

⊟ **Color**
| Border Color | ☐ **225; 228; 232** |
| Chart Color | ☐ **238; 238; 238** |

⊟ **Common**
Comment	
CPU Core	**Default**
Max Data Points	**120000**
Show Name	**False**

⊟ **Settings**
| Scale on Zoom | **True** |
| Stacked Axes | **False** |

Behaviour

Properties | Toolbox

Behaviour

Auto Start	If this option is enabled, the chart starts the live display when a new recording is started.
Data Tool Tip	If this option is enabled, a click on a data point results in display of a tool tip with the exact values of the data point on the X- and Y-axis, plus the timestamp and name of the corresponding channel.
Default Display Width	This time indicates the display time for the signal or the signal length in the XY chart. This value is set, for example, when the rescale button is selected in the chart toolbar.
Master Chart	In the drop-down list that is available here, a chart that is part of the current configuration can be selected and designated as master chart. All actions that are executed in the master chart are also automatically executed in this chart. A separate pause option is provided, through which XY-zoom and XY-panning modes are available.

Chart / **Chart(1)**
Start: 14:34:25,238:000 | End: 14:34:30,034:000 | Pos: 0,00:00:00,000:000 | Time: 14:34:25,238:000 | Date: Donnerstag, 2. März 2017
⊕ ⬛ ✕ | Assigned Master: Chart

Time Bar	Specifies whether the time bar is displayed in the chart.
Tool Bar	Specifies whether the toolbar is displayed in the chart.

Color

The background and frame color can be set by means of color dialogs.

Common

Comment	A free comment can be saved here.
CPU Core	Here you can set (for each chart individually) which CPU core should be used for the current chart display. If required, multiple CPU cores can be specified for a chart. This may improve the performance.
Max Data Points	Determines the maximum number of data points that can be displayed at the same time. The default value is 120,000. This means that, at a sampling rate of 1 ms (TaskSampleTime), the maximum recording time for which values can be displayed simultaneously is 2 minutes.
Show Name	Specifies whether the name of the chart is shown in the graph.

Settings

Scale on Zoom	If this option is selected, the chart instructs all axes (X and Y) to perform autoscaling after a zoom or panning action. This can be used to expand the selected area.
Stacked Axes	This setting can be used to specify whether the axes of a chart are shown side by side or above each other. In the side by side display, the values of the appended channels are shown within the same area. If they are shown above each other, the channels are displayed in their own areas.

To change the settings for several charts at the same time, please refer to the chapter Multiple selection [▶ 161].

The chart display is operated with the toolbar. The overview shows all buttons and their explanation (from left):

Play	Starts the live display mode. The data currently being accumulated are displayed.
Break	The display switches to the pause mode. You can now navigate in the data already recorded without stopping the recording.
Display-Width	The current display width is displayed here. It can be edited in the format hh:mm:ss,fff. Alternatively, the display width can be changed by turning the mouse wheel in the selected chart window. The changed value is adopted automatically.
Scroll buttons	The outer scroll buttons move the current display in steps that correspond to the display width. The inner scroll keys move the display only by a tenth of the display width and can be kept pressed to view the data set.
Position	Shows the position. It can be edited in the format hh:mm:ss,fff. The colons are used as separators. If not all units are edited the format is sorted in ascending order, starting with seconds.
Panning Free	In the free panning mode the current display can be shifted along the X- and Y-axes by clicking and dragging with the mouse.
Zoom Free	You can zoom into the current display by stretching a rectangle over the graphic area.
Zoom to Default	Carries out an autoscale on all axes. The x-axis is/axes are set to the default display time.
Zoom Out Max	Scales the X and Y-axis in such a way that all current values in the recording appear in the display. Up to the maximum number of data points that can be shown. The maximum number is 60,000 data points.

By turning the mouse wheel you can also zoom in or out on the current display. The cursor position defines the center.
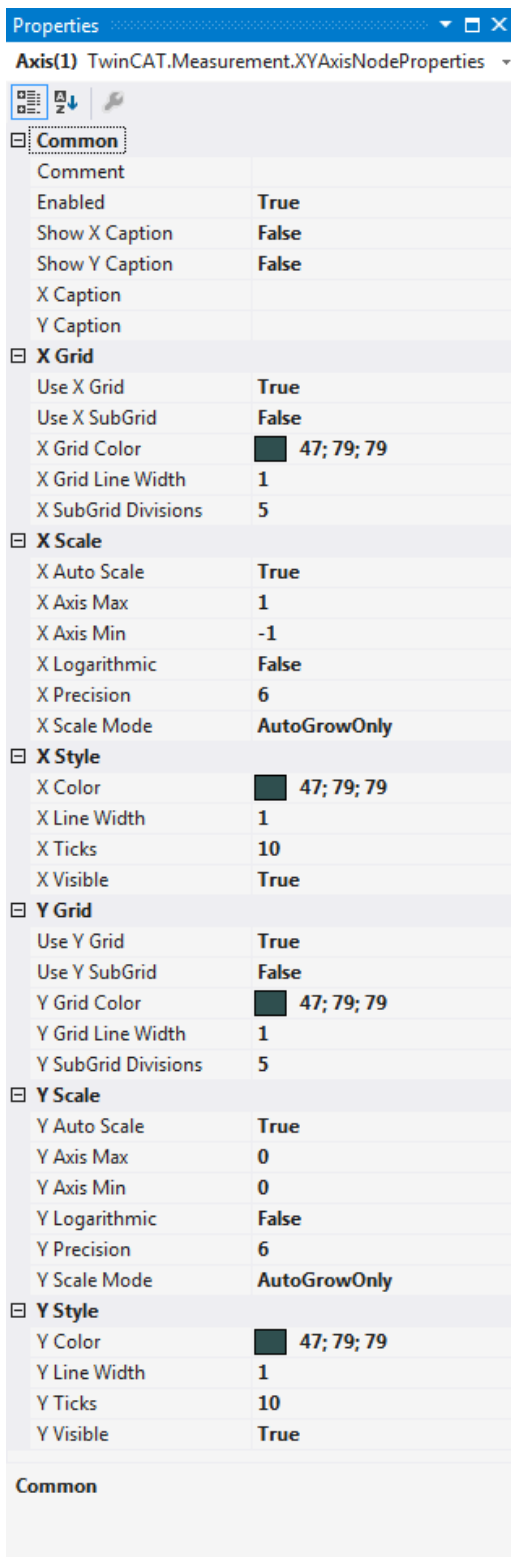
The current recording times are displayed in the chart toolbar:
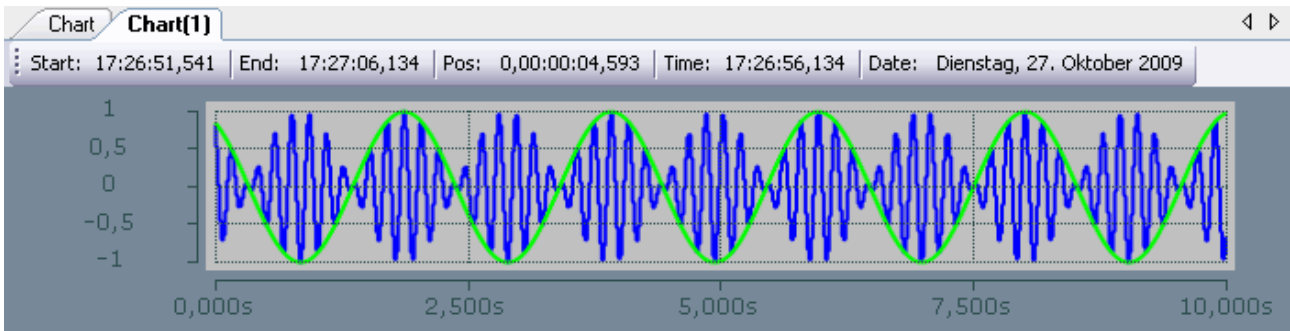
BECKHOFF

Start-Time	The common starting point of the recordings of all connected channels. The start time defines thus the zero point of the recording.
End-Time	Maximum common time of all connected channels. The end time marks the final value of the recording. The difference between the end time and start time is maximally as large as the record time set (see Scope nodes [▶ 23]).
Position	The position time represents the zero point of the current chart, i.e. the time from the start time to the beginning of the display.
Time	Absolute time at the chart origin.
Date	Absolute date at the chart origin.

XY axis properties

Here you can set separate settings for each available XY axis.

Common

Comment	A free comment can be saved here.
Enabled	Here you can switch the axis on or off completely. This functionality is also available via the context menu.
Show X Caption	Specifies whether the name of the x-axis is shown in the chart.
Show Y Caption	Specifies whether the name of the y-axis is shown in the chart.
X Caption	Here you can enter an x-axis label.
Y Caption	Here you can enter an y-axis label.

X Grid

Use X Grid	The X-subdivisions of the axis can be shown or hidden here.
Use X SubGrid	Auxiliary lines for the finer X-subdivision of the main grid can be shown or hidden here. Auxiliary lines have no axis labeling.
X Grid Color	Color of the grid
X Grid Line Width	Line width of the grid in pixels
X SubGrid Divisions	Number of areas into which the main grid is subdivided by auxiliary lines. In the case of logarithmic scaling the auxiliary lines for subdivision are shown only in the set number if the range of values per tick is precisely one decade. Otherwise the displayed auxiliary line shows the decades not displayed in the main grid.

X Scale

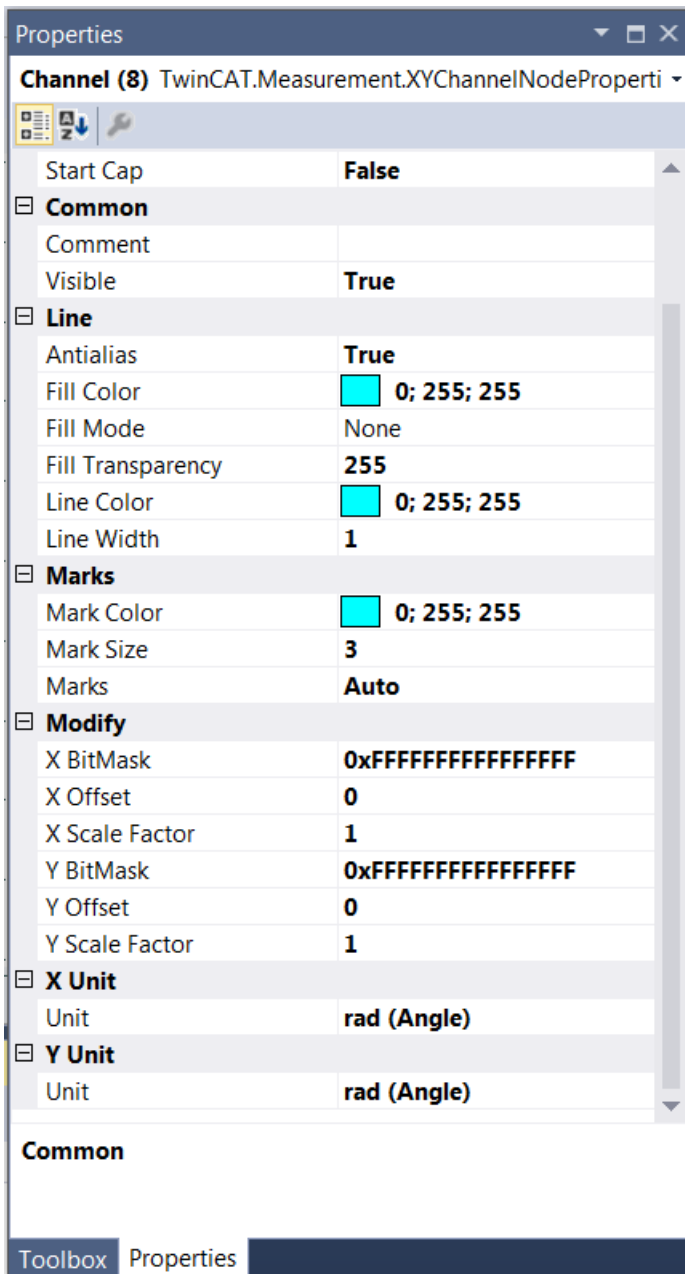| X Auto Scale	If Auto Scale is active, each axis is scaled such that the last-known minimum and maximum of all connected channels lie within the display range.
X Axis Max	If Auto Scale is inactive, the maximum value can be edited directly.
X Axis Min	If Auto Scale is inactive, the minimum value can be edited directly.
X Logarithmic	Switches between logarithmic and linear scaling of the axes.
X Precision	Here you can specify the number of significant digits for the axis labeling. It corresponds to the number of visible digits on the axis.
X Scale Mode	Two auto-scaling modes are available for selection. The default mode is AutoGrowOnly. This means the x-axis always scales to the historic maximum, until a rescale is carried out for the current view. Alternatively, AutoGrowNShrink can be selected. This mode always automatically adjusts the maximum of the x-axis to the maximum in the current view (display width).

X Style

X Color	Color of the axis
X Line Width	Line width in pixels
X Ticks	Maximum number of subdivisions. If there is not enough space for the selected number of ticks, these are automatically reduced in the chart. In the case of logarithmic scaling the number of ticks depends on the displayed range of values and can thus deviate from the settings.
X Visible	Specifies whether the axis is shown or hidden in the chart.

Y Grid

Use Y Grid	The Y-subdivisions of the axis can be shown or hidden here.
Use Y SubGrid	Auxiliary lines for the finer Y-subdivision of the main grid can be shown or hidden here. Auxiliary lines have no axis labeling.
Y Grid Color	Color of the grid
Y Grid Line Width	Line width of the grid in pixels
Y SubGrid Divisions	Number of areas into which the main grid is subdivided by auxiliary lines. In the case of logarithmic scaling the auxiliary lines for subdivision are shown only in the set number if the range of values per tick is precisely one decade. Otherwise the displayed auxiliary line shows the decades not displayed in the main grid.

Y Scale

Y Auto Scale	If Auto Scale is active, each axis is scaled such that the last-known minimum and maximum of all connected channels lie within the display range.
Y Axis Max	If Auto Scale is inactive, the maximum value can be edited directly.
Y Axis Min	If Auto Scale is inactive, the minimum value can be edited directly.
Y Logarithmic	Switches between logarithmic and linear scaling of the axes.
Y Precision	Here you can specify the number of significant digits for the axis labeling. It corresponds to the number of visible digits on the axis.
Y Scale Mode	Two auto-scaling modes are available for selection. The default mode is AutoGrowOnly. This means the y-axis always scales to the historic maximum, until a rescale is carried out for the current view. Alternatively, AutoGrowNShrink can be selected. This mode always automatically adjusts the maximum of the y-axis to the maximum in the current view (display width).

Y Style

Y Color	Color of the axis
Y Line Width	Line width in pixels
Y Ticks	Maximum number of subdivisions. If there is not enough space for the selected number of ticks, these are automatically reduced in the chart. In the case of logarithmic scaling the number of ticks depends on the displayed range of values and can thus deviate from the settings.
Y Visible	Specifies whether the axis is shown or hidden in the chart.

To change the settings for several axes at the same time, please refer to the chapter <u>Multiple selection</u> [▶ 161].

XT channel style

All style settings belonging to a XY channel can be made here.

Properties	▼ ▢ ✕
Channel (8) TwinCAT.Measurement.XYChannelNodeProperti ▼	

Start Cap	**False**
⊟ **Common**	
Comment	
Visible	**True**
⊟ **Line**	
Antialias	**True**
Fill Color	▢ **0; 255; 255**
Fill Mode	None
Fill Transparency	**255**
Line Color	▢ **0; 255; 255**
Line Width	**1**
⊟ **Marks**	
Mark Color	▢ **0; 255; 255**
Mark Size	**3**
Marks	**Auto**
⊟ **Modify**	
X BitMask	**0xFFFFFFFFFFFFFFFF**
X Offset	**0**
X Scale Factor	**1**
Y BitMask	**0xFFFFFFFFFFFFFFFF**
Y Offset	**0**
Y Scale Factor	**1**
⊟ **X Unit**	
Unit	**rad (Angle)**
⊟ **Y Unit**	
Unit	**rad (Angle)**

Common

Toolbox | Properties

Cap

Cap Color	Specifies the colour of the active cap.
Cap Size	Specifies the size of the cap.
End Cap	Activates the end cap for the channel.
Start Cap	Activates the start cap for the channel.

Common

Comment	A free comment can be saved here. If applicable the comment will be adopted from the connected ADS symbol.
Visible	Here you can set whether the channel should be displayed in the chart.

Line

Antialias	This option decides how the lines will be drawn. Antialias is "nicer" but requires considerably more computing. This has an effect in particular if there are many channels with large movements.
Fill Color	Fill mode can be used to color an area, e.g. above or below a curve. Here you can select the fill color.
Fill Mode	Fill mode can be used to color an area, e.g. above or below a curve. The default setting is "None". Alternatives are: Horizontal Zero, Bottom, Top, Center and Source.
Fill Transparency	Here you can set the transparency value of the fill color. The default value is 50.
Line Color	Graph color
Line Width	Line width of the chart. The line width 1 requires least computing.

Marks

Mark Color	Color of the interpolation point markers
Marks	Selection of the visibility of markings: On (permanently visible) | Auto (dependent on the zoom level) | Off (markings switched off).
Mark Size	Size of the interpolation point markers

Modify

X Bit Mask	In this field a value can be entered with which the display value is masked, provided it is not of the type floating point. This means that the value of the channel is ANDed with the binary value of the mask. This helps you, for example, to observe individual bits of a status byte.
X Offset	A free offset can be added to the function value of the channel in the offset field.
X Scale Factor	The display value of a channel can be changed with the scale factor.
Y Bit Mask	In this field a value can be entered with which the display value is masked, provided it is not of the type floating point. This means that the value of the channel is ANDed with the binary value of the mask. This helps you, for example, to observe individual bits of a status byte.
Y Offset	A free offset can be added to the function value of the channel in the offset field.
Y Scale Factor	The display value of a channel can be changed with the scale factor.

X Unit and Y Unit

Unit	This property enables the physical unit to be configured individually for the X-acquisition and the Y-acquisition of the channel. This can be done with the aid of the "Unit Wizard", which can be opened using the button on the right in the field. A further option is to define the units beforehand as attributes in the PLC code. These are then read out directly and entered in the Unit field. Detailed information on the units and their configuration options can be taken from the section Units [▶ 85].

To change the settings for several channels at the same time, please refer to the chapter .

Configuration of channel style properties with the aid of PLC attributes

All style properties of a channel can be set directly on a variable or a data type in the PLC with the aid of attributes. This makes it possible to always display a certain symbol with the same style properties without having to reconfigure it every time.

A PLC attribute is specified directly via the declaration of a variable or a data type in curly brackets.

```
28    {attribute 'TcScope' := '<LineWidth:3, LineColor:red>'}
29    fTriangular          : LREAL;
```

The following syntax applies to the assignment of the channel style properties:

{attribute 'TcScope' := '<Property1:value, Property2:value,…>'}

Example:
{attribute 'TcScope' := '<LineWidth:3, LineColor:red>'}

Any number of channel style properties can be specified in an attribute.

Note here that the X-acquisition and Y-acquisition sometimes have common channel style properties. For example, the properties that concern the line or the marks. If two symbols are added to an XY channel in which such properties are stored as PLC attributes, the properties of the symbol added last are always set. A separate syntax applies to the specification of the unit (Unit property; see: Configuration of units with the aid of PLC attributes [▶ 100]).

XY channel acquisition

All acquisition settings belonging to a XY channel can be made here.

Properties	▼ ☐ ✕
X: **Variables.fSine** TwinCAT.Measurement.AcquisitionNodeProperties ▼	

⊟ **Acquisition**	
Data-Type	**REAL64**
Enabled	**True**
Sample State	**TaskSampleTime**
Sample Time	1
Symbol based	**True**
Symbol Comment	
Symbol Index Group	0x4040
Symbol Index Offset	0x859E8
Symbol Name	**Variables.fSine**
Symbol Size	**8**
Target Port	**851 : unknown**
Target System	**NINAGR-NB (172.17.214.54.1.1)**
Time Offset [s]	**0**
Use Local Server	**True**

Data-Type
Data-Type

Toolbox	Properties	

● **Different properties**

ⓘ The properties of the channels can deviate from the illustration shown here.
Some setting options are only visible/editable for special channels.
Example: The setting **Array Length** is only visible for array symbols.

Acquisition

Data-Type	Indicates the data type of the selected variable.
Enabled	Here you can decide whether the configured channel should be recorded or not.
Oversampling	This option is only shown for arrays. The array is interpreted as a variable, and the values of the individual array elements are displayed contiguously as oversampling values. This corresponds to the behavior of EtherCAT oversampling terminals. The value that can be entered here describes the number of oversampling values to be displayed. By default, the value corresponds to the array length. Comparisons: <u>Oversampling recordings [▶ 163]</u>.
Sample State	Here you can specify whether the data for the selected variables are to be sampled with the TaskSampleTime or the FreeSampleTime.
Sample Time	If the sample state "FreeSampleTime" was selected, you can set the sampling rate here. It is always greater than the TaskSampleTime.
Symbol based	If this option is set, the variables are communicated via the symbol name. If this option is not set, the variables are communicated via the Group and Offset indices, and the input fields for Group and Offset are enabled.
Symbol Comment	Here the actual symbol comment is displayed, if such a comment exists.
Symbol Index Group	Group index of the variable
Symbol Index Offset	Offset index of the variable
Symbol Name	Symbol name of the selected variable.
Symbol Size	Shows the size of the variables in bytes.
Target Port	Shows the variable port on the respective TwinCAT system. A drop-down list can be used to select the available ports for the selected target system.
Target System	Indicates the target system selected for a recording. By default this is the target system for the selected variable.
Time Offset [s]	This can be used for one-time manipulation of the original timestamp during recording of the data point. e.g. to compensate path differences of different target systems. The phase differences have to be determined manually by the user.
Use Local Server	If this option is set, the Scope Server installed on the Scope View system is used for the recording. If the option is not set, a connection with the remote server of the target system is established.

To change the settings for several channels at the same time, please refer to the chapter <u>Multiple selection</u> <u>[▶ 161]</u>.

5.1.1.3.3 Array Bar Chart

The properties of the individual hierarchy levels of array bar charts are explained below.

The View Detail level, which can be set in the Scope settings, can be used to separate Channel Style and Channel Acquisition at channel level through an additional hierarchy level. Channel Style and Channel Acquisition are described separately below.

Bar chart properties

Here you can enter all the settings relating to a array bar chart.

Behaviour

Auto Start	. If this option is enabled, the chart starts the live display when a new recording is started.
Data Tool Tip	If this option is enabled, a click on a data point results in display of a tool tip with the exact values of the data point on the X- and Y-axis, plus the timestamp and name of the corresponding channel.
Default Display Width	This time refers to the display times of the minimum and maximum values in the array bar chart. The extreme values are always recorded. The freely adjustable display time makes it possible to display an extended history of the extreme values.
Master Chart	In the drop-down list that is available here, a chart that is part of the current configuration can be selected and designated as master chart. All actions that are executed in the master chart are also automatically executed in this chart. A separate pause option is provided, through which XY-zoom and XY-panning modes are available.

Chart Chart(1)

Start: 14:34:25,238:000 | End: 14:34:30,034:000 | Pos: 0,00:00:00,000:000 | Time: 14:34:25,238:000 | Date: Donnerstag, 2. März 2017

Assigned Master: Chart

Time Bar	Specifies whether the time bar is displayed in the chart.
Tool Bar	Specifies whether the toolbar is displayed in the chart.

Color

The background and frame color can be set by means of color dialogs.

Common

Comment	A free comment can be saved here.
CPU Core	Here you can set (for each chart individually) which CPU core should be used for the current chart display. If required, multiple CPU cores can be specified for a chart. This may improve the performance.
Show Name	Specifies whether the name of the chart is shown in the graph.

Settings

Stacked Axes	This setting can be used to specify whether the axes of a chart are shown side by side or above each other. In the side by side display, the values of the appended channels are shown within the same area. If they are shown above each other, the channels are displayed in their own areas.

To change the settings for several charts at the same time, please refer to the chapter <u>Multiple selection</u> [▶ <u>161</u>].

The chart display is operated with the toolbar. The overview shows all buttons and their explanation (from left):

Start: 10:25:05,417:000 | End: 10:25:18,417:000 | Pos: 0,00:00:02,800:000 | Time: 10:25:08,217:000 | Date: Freitag, 22. Januar 2016

0,00:00:10,000:000   0,00:00:00,000:000

Play	Starts the live display mode. The data currently being accumulated are displayed.
Pause	The display switches to the pause mode. You can now navigate in the data already recorded without stopping the recording.
Display-Width	This time refers to the display times of the minimum and maximum values in the array bar chart. The extreme values are always recorded. The freely adjustable display time makes it possible to display an extended history of the extreme values.
Scroll buttons	The outer scroll buttons move the current display in steps that correspond to the display width. The inner scroll keys move the display only by a tenth of the display width and can be kept pressed to view the data set.
Position	The displayed position can be edited in the Position field in the format hh:mm:ss,fff. The colons serve here as separators. If not all units are edited the format is sorted in ascending order, starting with seconds.
Undo/Redo Time/Position	This option can be used to undo step changes in the display width or the current position, irrespective of how they were made (e.g. zoom, scroll, etc.). The right mouse button is likewise assigned this function. Once undone, values can be repeated with redo.
Panning Horizontal	In the horizontal panning mode the current display can be shifted along the x-axis by clicking and dragging with the mouse.
Panning Free	In the free panning mode the current display can be shifted along the x- and y-axes by clicking and dragging with the mouse.
Zoom Horizontal	A new time range for the display can be selected by stretching a rectangle over the x-axis.
Zoom Free	You can zoom into the current display by stretching a rectangle over the graphic area.
Zoom to Default	If autoscale is set, this option triggers a zoom to the maximum number of array elements. If autoscale is disabled, the zoom automatically focuses on the defined limits.
Zoom Out Max	Scales the x-axis such that all array elements are displayed.

Bar axis properties

All the settings for the axes of array bar charts are described here.

Common

Comment	A free comment can be saved here.
Enabled	Here you can switch the axis on or off completely. This functionality is also available via the context menu.
Show X Caption	Specifies whether the name of the x-axis is shown in the chart.
Show Y Caption	Specifies whether the name of the y-axis is shown in the chart.
X Caption	Here you can enter an x-axis label.
Y Caption	Here you can enter an y-axis label.

X Grid

Use X Grid	The X-subdivisions of the axis can be shown or hidden here.
Use X SubGrid	Auxiliary lines for the finer X-subdivision of the main grid can be shown or hidden here. Auxiliary lines have no axis labeling.
X Grid Color	Color of the grid
X Grid Line Width	Line width of the grid in pixels
X SubGrid Divisions	Number of areas into which the main grid is subdivided by auxiliary lines. In the case of logarithmic scaling the auxiliary lines for subdivision are shown only in the set number if the range of values per tick is precisely one decade. Otherwise the displayed auxiliary line shows the decades not displayed in the main grid.

X Scale

X Auto Scale	If Auto Scale is active, each axis is scaled such that the last-known minimum and maximum of all connected channels lie within the display range.
X Axis Max	If Auto Scale is inactive, the maximum value can be edited directly.
X Axis Min	If Auto Scale is inactive, the minimum value can be edited directly.
X Logarithmic	Switches between logarithmic and linear scaling of the axes.
X Precision	Here you can specify the number of significant digits for the axis labeling. It corresponds to the number of visible digits on the axis.

X Style

X Color	Color of the axis
X Line Width	Line width in pixels
X Ticks	Maximum number of subdivisions. If there is not enough space for the selected number of ticks, these are automatically reduced in the chart. In the case of logarithmic scaling the number of ticks depends on the displayed range of values and can thus deviate from the settings.
X Visible	Specifies whether the axis is shown or hidden in the chart.

Y Grid

Use Y Grid	The Y-subdivisions of the axis can be shown or hidden here.
Use Y SubGrid	Auxiliary lines for the finer Y-subdivision of the main grid can be shown or hidden here. Auxiliary lines have no axis labeling.
Y Grid Color	Color of the grid
Y Grid Line Width	Line width of the grid in pixels
Y SubGrid Divisions	Number of areas into which the main grid is subdivided by auxiliary lines. In the case of logarithmic scaling the auxiliary lines for subdivision are shown only in the set number if the range of values per tick is precisely one decade. Otherwise the displayed auxiliary line shows the decades not displayed in the main grid.

Y Scale

Y Auto Scale	If Auto Scale is active, each axis is scaled such that the last-known minimum and maximum of all connected channels lie within the display range.
Y Axis Max	If Auto Scale is inactive, the maximum value can be edited directly.
Y Axis Min	If Auto Scale is inactive, the minimum value can be edited directly.
Y Logarithmic	Switches between logarithmic and linear scaling of the axes.
Y Precision	Here you can specify the number of significant digits for the axis labeling. It corresponds to the number of visible digits on the axis.

Y Style

Y Color	Color of the axis
Y Line Width	Line width in pixels
Y Ticks	Maximum number of subdivisions. If there is not enough space for the selected number of ticks, these are automatically reduced in the chart. In the case of logarithmic scaling the number of ticks depends on the displayed range of values and can thus deviate from the settings.
Y Visible	Specifies whether the axis is shown or hidden in the chart.

To change the settings for several axes at the same time, please refer to the chapter Multiple selection [▶ 161].

Bar channel style
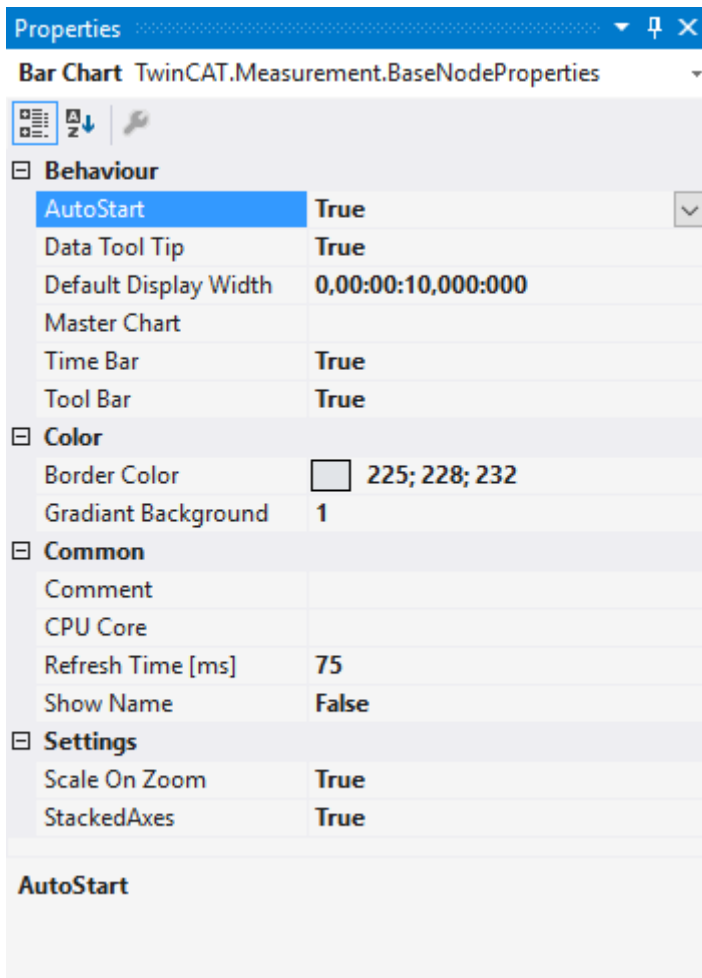
All Style properties of array bar chart channels are explained here.

Common

Comment	A free comment can be saved here. If applicable the comment will be adopted from the connected ADS symbol.
Visible	Here you can set whether the channel should be displayed in the chart.

Line

Antialias	This option decides how the lines will be drawn. Antialias is "nicer" but requires considerably more computing. This has an effect in particular if there are many channels with large movements.
Graph Type	Here you can switch between different display types. The default value for an array bar chart is Bar. Other options are: Line and Stair.
Line Color	Graph color
Line Width	Line width of the chart. The line width 1 requires least computing.
Show Max	If the option is set to TRUE, the maximum values for the set display width are shown.
Show Min	If the option is set to TRUE, the minimum values for the set display width are shown.

Marks

Mark Color	Color of the interpolation point markers
Marks	Selecting the visibility of markings → On (permanently visible) | Off (markings switched off).
Mark Size	Size of the interpolation point markers

Modify

BitMask	In this field a value can be entered with which the display value is masked, provided it is not of the type floating point. This means that the value of the channel is ANDed with the binary value of the mask. This helps you, for example, to observe individual bits of a status byte.
X Offset	A free offset can be added to the function value of the channel in the offset field.
X Scale Factor	The display value of a channel can be changed with the scale factor.
Y Offset	A free offset can be added to the function value of the channel in the offset field.
Y Scale Factor	The display value of a channel can be changed with the scale factor.

Unit

Unit	This property allows the physical unit of the respective channel to be configured. This can be done with the aid of the "Unit Wizard", which can be opened using the button on the right in the field. A further option is to define the units beforehand as attributes in the PLC code. These are then read out directly and entered in the Unit field. Detailed information on the units and their configuration options can be taken from the section Units [▶ 85].

To change the settings for several channels at the same time, please refer to the chapter Multiple selection [▶ 161].

Configuration of channel style properties with the aid of PLC attributes

All style properties of a channel can be set directly on a variable or a data type in the PLC with the aid of attributes. This makes it possible to always display a certain symbol with the same style properties without having to reconfigure it every time.

A PLC attribute is specified directly via the declaration of a variable or a data type in curly brackets.

```
28    {attribute 'TcScope' := '<LineWidth:3, LineColor:red>'}
29    fTriangular             : LREAL;
```

The following syntax applies to the assignment of the channel style properties:

{attribute 'TcScope' := '<Property1:value, Property2:value,…>'}

Example:
{attribute 'TcScope' := '<LineWidth:3, LineColor:red>'}

Any number of channel style properties can be specified in an attribute.

A separate syntax applies to the specification of the unit (Unit property; see: Configuration of units with the aid of PLC attributes [▶ 100]).

Bar channel acquisition

This section explains all settings for the channel acquisition of array bar chart channels.

Properties	▼ ☐ ✕
Y: Variables.aBufferAM TwinCAT.Measurement.AcquisitionNodeProp ▾	

Acquisition	
Array Length	**10**
Data-Type	**REAL64**
Enabled	**True**
Sample State	**FreeSample**
Sample Time	**100**
Symbol based	**True**
Symbol Comment	
Symbol Index Group	0x4040
Symbol Index Offset	0x85A08
Symbol Name	**Variables.aBufferAM**
Symbol Size	**80**
Target Port	**851 : unknown**
Target System	**NINAGR-NB (172.17.214.54.1.1)**
Time Offset [s]	**0**
Use Local Server	**True**

Acquisition

Toolbox | Properties

Acquisition

Array Length	Indicates the length of selected arrays, which is also the maximum displayed on the x-axis.
Data-Type	Indicates the data type of the selected variable.
Enabled	Here you can decide whether the configured channel should be recorded or not.
Sample State	Here you can specify whether the data for the selected variables are to be sampled with the TaskSampleTime or the FreeSampleTime. The default value for array bar charts is "FreeSample", since it makes no sense to scan a bar display so quickly that it is not possible for the human eye to observe it.
Sample Time	If the sample state "FreeSampleTime" was selected, you can set the sampling rate here. It is always greater than the "TaskSampleTime". The default setting for array bar charts is 100 ms. This is a sampling rate for bar charts that is comfortable for the human eye.
Symbol based	If this option is set, the variables are communicated via the symbol name. If this option is not set, the variables are communicated via the Group and Offset indices, and the input fields for Group and Offset are enabled.
Symbol Comment	Here the actual symbol comment is displayed, if such a comment exists.
Symbol Index Group	Group index of the variable
Symbol Index Offset	Offset index of the variable
Symbol Name	Symbol name of the selected variable
Symbol Size	Shows the size of the variables in bytes.
Target Port	Shows the variable port on the respective TwinCAT system. A drop-down box can be used to select the available ports for the selected target system.
Target System	Indicates the target system selected for a recording. By default this is the target system for the selected variable.
Time Offset [s]	This can be used for one-time manipulation of the original time stamp during recording of the data point. e.g. to compensate path differences of different target systems. The phase differences have to be determined manually by the user.
Use Local Server	If this option is set, the Scope Server installed on the Scope View system is used for the recording. If the option is not set, the system tries to connect to the remote server of the target system.

To change the settings for several channels at the same time, please refer to the chapter <u>Multiple selection</u> [▶ 161].

5.1.1.3.4 Single bar chart

All the properties of the individual hierarchy levels of single bar charts are explained below.
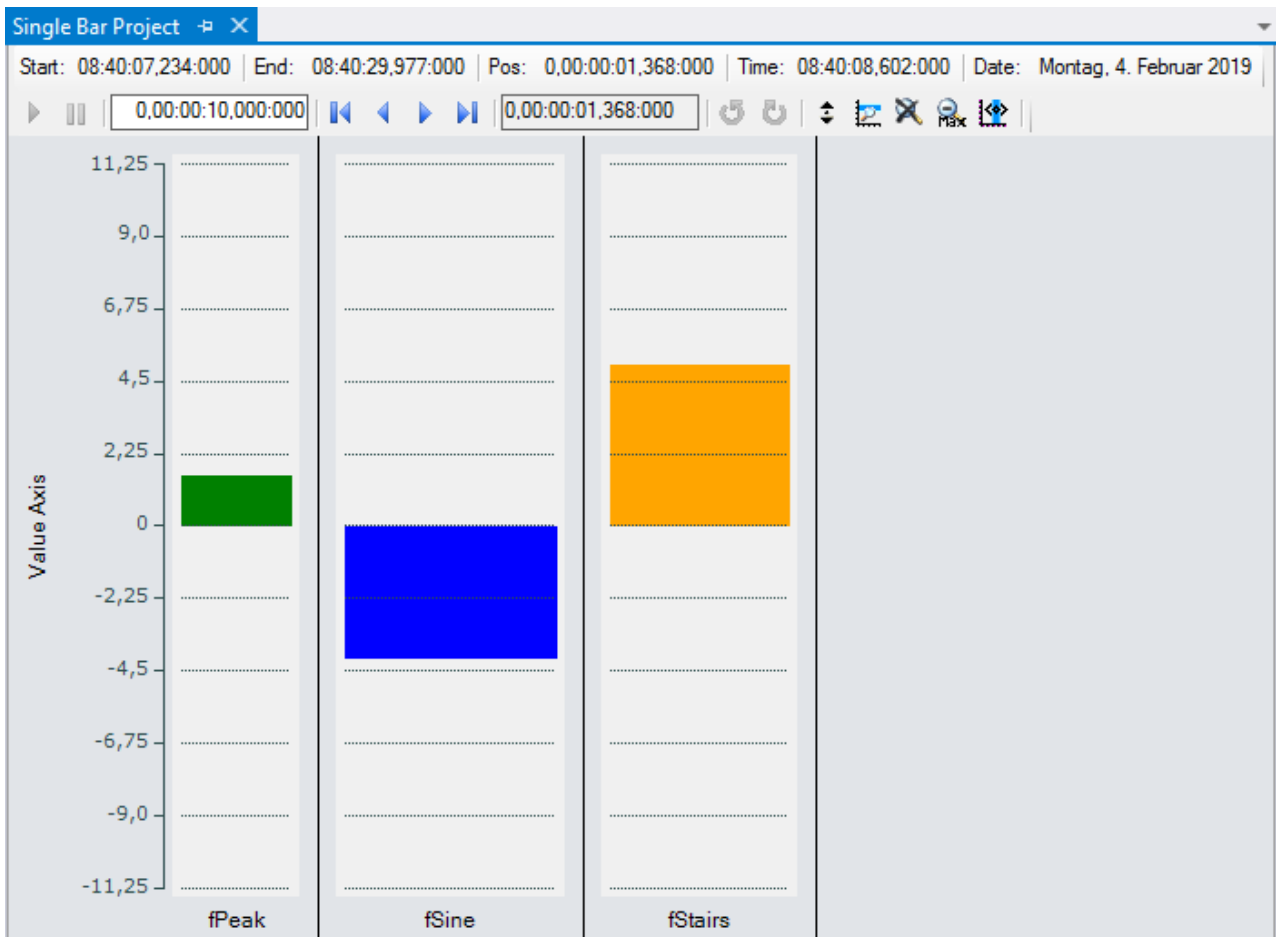
Single bar chart properties

All settings assigned to the single bar chart can be made here.

BECKHOFF

Behaviour

AutoStart	If this option is enabled, the chart starts the live display when a new recording is started.
Data Tool Tip	If this option is activated, a tool tip showing the exact values of a data point is displayed when clicking on the data point. The time stamp and the name of corresponding channel are also shown.
Default Display Width	This time specifies the standard width of the associated chart. This value determines the period between the displayed minimum and maximum values.
Master Chart	In the drop-down list that is available here, a chart that is part of the current configuration can be selected and designated as master chart. All actions that are executed in the master chart are also automatically executed in this chart. A separate pause option is provided, through which Y-zoom and Y-panning modes are available. An overview chart can also be displayed.
Time Bar	Specifies whether the time bar (information bar with time values) is displayed in the chart.
Tool Bar	Specifies whether the toolbar is displayed in the chart.

Color

Border Color	Defines the background color of the chart.
Gradient Background	Provides for a color gradient in the background color.

Version: 1.9 TC3 Scope View

Common

Comment	A free comment can be saved here.
CPU Core	Here you can set (for each chart individually) which CPU core should be used for the current chart display. If required, multiple CPU cores can be specified for a chart. This may improve the performance.
Refresh Time	Defines a rate at which the chart in the user interface is refreshed.
Show Name	Specifies whether the name of the chart is shown in the graph.

Settings

Scale on Zoom	If this option is selected, the chart instructs the Y-axes to perform autoscaling after a zoom or panning action. The selected area can thus be widened.
Stacked Axes	With this setting you can select whether the various axis groups are arranged on a horizontal line or underneath one another.

To change the settings for several charts at the same time, please refer to the chapter Multiple selection [▶ 161].

The chart display is operated with the toolbar. The overview shows all buttons and their explanation (from left):

Play	Starts the live display mode. The data currently being accumulated are displayed.
Break	The display switches to the pause mode. You can now navigate in the data already recorded without stopping the recording.
Display-Width	The current display width is displayed here. It can be edited in the format hh:mm:ss,fff. The zoom function works down to the µs range.
Scroll buttons	The outer scroll buttons move the current display in steps that correspond to the display width. The inner scroll keys move the display only by a tenth of the display width and can be kept pressed to view the data set.
Position	Shows the position. It can be edited in the format hh:mm:ss,fff. The colons are used as separators. If not all units are edited the format is sorted in ascending order, starting with seconds.
Undo/Redo Time/ Position	This option can be used to undo step changes in the display width or the current position, irrespective of how they were made (e.g. zoom, scroll, etc.). The right mouse button is likewise assigned this function. Once undone, values can be repeated with redo.
Panning Vertical	In the vertical panning mode the current display can be shifted along the Y-axis by clicking and dragging.
Zoom Vertical	A new value range for the Y-axis can be selected by stretching a rectangle over the axis.
Zoom to Default	Carries out an autoscale on all axes. The display range is reset to the default display time.
Zoom Out Max	Sets the display range to the maximum recording length.
Overview	Use the Overview option to display a chart within the chart. The signal range currently shown in the main chart is highlighted in the Overview Chart. In addition the overview chart offers an absolute time axis for the entire recording duration.

The current recording times are displayed in the chart toolbar:

Start-Time	The common starting point of the recordings of all connected channels. The start time defines the zero point of the recording.
End-Time	Maximum common time of all connected channels. The end time thus marks the final value of the recording. The difference between the end time and start time is maximally as large as the record time set (see Scope nodes [▶ 23]).
Position	The position time represents the zero point of the current chart, i.e. the time from the start time to the beginning of the display.
Time	Absolute time at the chart origin
Date	Absolute date at the chart origin

Single bar axis properties

Settings can be made separately for each available single bar axis here.

Properties ▾ ⊟ ✕

Axis Group TwinCAT.Measurement.BaseNodeProperties ▾

⊟ **Common**
Canvas Color	☐ 238; 238; 238
Comment	
Enabled	**True**

⊟ **Y-Axis**
Color	■ **DarkSlateGray**
Inverted	**False**
Linewidth	**1**
Logarithmic	**False**
Manual Max	**0,5**
Manual Min	**-0,5**
Precision	**6**
Scale Mode	**AutoGrowOnly**
Show Title	**False**
Title	**Value Axis**
Visible	**True**

⊟ **Y-Grid**
Grid Color	■ **DarkSlateGray**
Grid Linewidth	**1**
GridDivisions	**10**
Show Grid	**True**
Show Subgrid	**False**
Subgrid Divisions	**5**

Color

Common

Canvas Color	Defines the color of the chart.
Comment	A free comment can be saved here.
Enabled	Here you can switch the axis on or off completely. This functionality is also available via the context menu.

Y-axis

Color	Color of the axis
Linewidth	Line width in pixels
Logarithmic	Switches between logarithmic and linear scaling of the axes.
Manual Max	If the Scale Mode is set to MinMax, the maximum value can be edited directly.
Manual Min	If the Scale Mode is set to MinMax, the minimum value can be edited directly.
Precision	Here you can specify the number of significant digits for the axis labeling. It corresponds to the number of visible digits on the axis.
Scale Mode	Two auto-scaling modes are available for selection. The default mode is AutoGrowOnly. This means the y-axis always scales to the historic maximum, until a rescale is carried out for the current view. Alternatively, AutoGrowNShrink can be selected. This mode always automatically adjusts the maximum of the y-axis to the maximum in the current view (display width).
Show Title	Specifies whether the name of the axis is shown in the chart.
Title	Defines the name of the axis.
Visible	Specifies whether the axis is shown or hidden in the chart.

Y-Grid

Grid Color	Color of the grid.
Grid Linewidth	Line width of the grid in pixels
Grid Divisions	Maximum number of subdivisions. If there is not enough space for the selected number of ticks, these are automatically reduced in the chart. In the case of logarithmic scaling the number of ticks depends on the displayed range of values and can thus deviate from the settings.
Show Grid	The subdivision of the axis can be shown or hidden here.
Show Subgrid	Auxiliary lines for the finer Y-subdivision of the main grid can be shown or hidden here. These auxiliary lines have no axis labeling.
Subgrid Divisions	Number of areas into which the main grid is subdivided by auxiliary lines. In the case of logarithmic scaling the auxiliary lines for subdivision are shown only in the set number if the range of values per tick is precisely one decade. Otherwise the displayed auxiliary line shows the decades not displayed in the main grid.

To change the settings for several axes at the same time, please refer to the chapter <u>Multiple selection</u> [▶ 161].

Single Bar Channel

All settings belonging to a single bar channel can be made here.

Common

Enabled	No data are acquired for the channel if Enabled is set to false.
Visible	Here you can set whether the channel should be displayed in the chart.

Line

Antialias	This option decides how the lines will be drawn. The computing time is longer with antialias. This has an effect in particular if there are many channels with large movements.
Graph Type	Here you can switch between different display types. The default value with a single bar chart is bar. Other options are: Line and Stair.
Line Color	Graph Colour.

Marks

Mark Color	Color of the interpolation point marking.
Mark Size	Size of the interpolation point marking.
Mark State	Selecting the visibility of markings → On (permanently visible)	Off (markings switched off).

Min/Max

Show Max	If the option is set to TRUE, the maximum values for the set display width are shown.
Show Min	If the option is set to TRUE, the minimum value for the set display width is shown.

Position

Position	Indicates the position of the channel on the axis.

To change the settings for several channels at the same time, please refer to the chapter <u>Multiple selection</u> [▶ 161].

Configuration of channel style properties with the aid of PLC attributes

All style properties of a channel can be set directly on a variable or a data type in the PLC with the aid of attributes. This makes it possible to always display a certain symbol with the same style properties without having to reconfigure it every time.

A PLC attribute is specified directly via the declaration of a variable or a data type in curly brackets.

```
28      {attribute 'TcScope' := '<LineWidth:3, LineColor:red>'}
29      fTriangular             : LREAL;
```

The following syntax applies to the assignment of the channel style properties:

{attribute 'TcScope' := '<Property1:value, Property2:value,…>'}

Example:
{attribute 'TcScope' := '<LineWidth:3, LineColor:red>'}

Any number of channel style properties can be specified in an attribute.

A separate syntax applies to the specification of the unit (Unit property; see: <u>Configuration of units with the aid of PLC attributes</u> [▶ 100]).

Single bar channel acquisition

All the settings for the channel acquisition interpreter of single bar charts are explained here.

Acquisition

Acquisition	Selection of the acquisition used from the data pool.

Modify

Bitmask	In this field a value can be entered with which the display value is masked, provided it is not of the type floating point. This means that the value of the channel is ANDed with the binary value of the mask. This helps you, for example, to observe individual bits of a status byte.
Offset	A free offset can be added to the function value of the channel in the offset field.
Scalefactor	The display value of a channel can be changed with the scale factor.
Unit	This property enables the physical unit to be configured individually for the X-acquisition and the Y-acquisition of the channel. This can be done with the aid of the "Unit Wizard", which can be opened using the button on the right in the field. A further option is to define the units beforehand as attributes in the PLC code. These are then read out directly and entered in the Unit field. Detailed information on the units and their configuration options can be taken from the chapter [▶ 85].

5.1.1.4   Trigger

Various trigger functions can be added to the Scope configuration.

In order to facilitate combination of individual trigger conditions and actions, these are grouped into trigger groups.

> **ℹ** **Properties of the trigger group**
>
> The properties of the trigger group can deviate from the illustration shown here, because some setting options are visible and editable only with special trigger actions. For example, the **Trigger Position** setting is visible only with the trigger actions "Stop Display" and "Restart Display".

Trigger groups and trigger sets can be added via the trigger context menu.
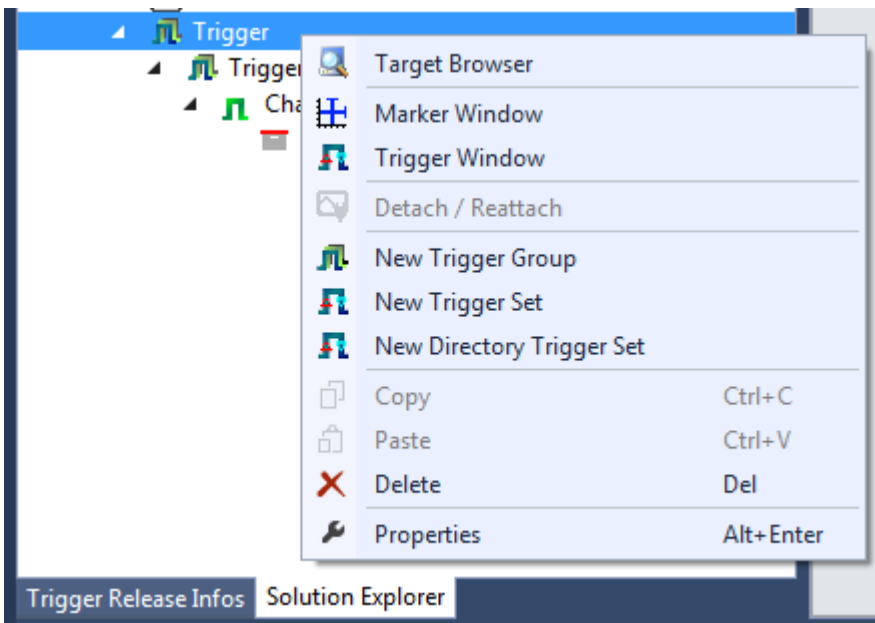
Trigger group
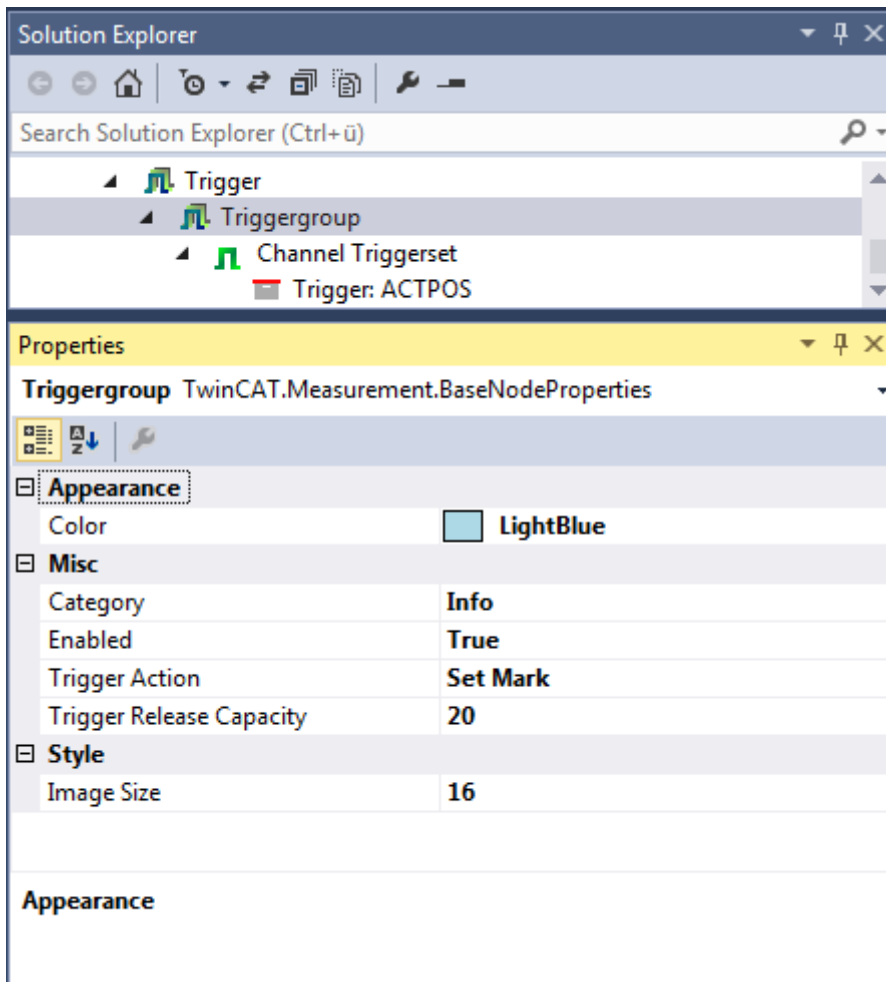
The trigger group provides settings for the trigger actions and for the general properties.

Appearance

Color	Color of the trigger

Misc

Trigger Action	**Set Mark:** If this option is selected, the position at which the trigger set condition occurs is marked by a horizontal line.
	Start Record: If this option is selected, the Start Record condition should be set to Trigger Start in the Scope settings. Otherwise the Scope would start recording as usual. With setting this action, therefore, a question box appears that carries out this setting if necessary. When the Record button is pressed in the menu bar, the scope connects to the participating servers as usual and starts to monitor the connected channels, without starting the actual recording. The recording begins with the trigger time of the last trigger set condition.
	Stop Record: If the stop action is selected, the Scope should be operated in the ring buffer mode. With setting this action, therefore, a question box appears that carries out this setting if necessary. The recording is then stopped when the Trigger Set condition occurs. In addition, the options Pre-Trigger and Post-Trigger appear, which can be used to indicate whether a fixed time range should appear in the recording before and/or after the trigger event. Is Pre-Trigger is selected, trigger events that occur before this time has elapsed are ignored. If Post-Trigger is active, the recording continues for the specified time after the trigger event.
	Start Subsave: This trigger event starts a background recording with the current configuration. This subsave always runs in the ring buffer mode (independent of the settings in the Scope settings). It is possible to start up to five subsaves simultaneously (via one or more trigger events). The maximum memory capacity of the subsave can be specified in the Record Time field. This recording duration may also be longer than the recording duration of the basic configuration (in this case ring buffer operation should also be selected).
	Stop Subsave: After the StopSubsave trigger event, the oldest subsave is stopped and transferred to the view in the background. From here it is saved in a .svd file via the specified path. The generated filename consists of the name of the Scope in the basic configuration and an ID composed of the date and time, in alphanumeric order.
	Stop Display: The Stop Display action stops all the charts of the Scope, which are in live mode at the trigger time, after the Trigger Set condition has occurred and the entire display width is full. If the trigger condition is met again, the display jumps to this new event. Pause can be actuated in the chart in order to prevent this retriggering.
	Restart Display: All charts paused due to a StopDispay trigger are resumed on fulfillment of the trigger condition.
Enable	Here you can switch the trigger group on or off.
Trigger Release Capacity	The Trigger Release Capacity indicates how many trigger markers are to be stored and displayed simultaneously. These are always the last recorded markers. The default value is 20, so that the last 20 recorded trigger markers are visible in the overview chart and can be displayed in the chart.
Category	This option is available only if no directory trigger sets have been created underneath the trigger group. If a category is defined, an image is drawn above the marking of the trigger release info. Various preset images can be shown, or a user-defined image if "CustomPicture" has been selected. If "Text" is selected, no image is shown above the trigger marking, but rather the specified text.
Content Value	This option is available only if "CustomPicture" or "Text" is selected in "Category". In the case of "CustomPicture" the path to a user-defined image or a folder can be set with this option. If a folder has been specified, the latest image saved in the folder is used. In the case of "Text", a text can be specified here that is to be displayed above the trigger marking.
Post-Trigger	This option is only available with the "Stop Record" and "Stop Subsave" trigger actions. Post-Trigger causes the recording to continue for a certain time after the Trigger Set condition has occurred. This time can be entered here.
Pre-Trigger	This option is only available with the "Stop Record" and "Stop Subsave" trigger actions. Pre-Trigger causes the Trigger Set conditions to be ignored from the start of the recording to the expiration of the Pre-Trigger time. The corresponding time can be set here.
Use Post-Trigger	This option is only available with the "Stop Record" and "Stop Subsave" trigger actions. The Post-Trigger can be switched on or off here.

Use Pre-Trigger	This option is only available with the "Stop Record" and "Stop Subsave" trigger actions. The Pre-Trigger can be switched on or off here.
Record Time	This option is only available with the trigger action "Start Subsave". The recording time for the subsaves can be entered here. This recording duration may also be longer than the recording duration of the basic configuration (in this case ring buffer operation should also be selected).
Save Path	This option is only available with the "Stop Subsave" trigger action. The path to the file in which the subsave is to be saved can be entered here.
Clear Chart	This option is only available with the "Stop Display" and "Restart Display" trigger actions. If this option is set, the chart range visible on the display is deleted when the Trigger Set condition occurs again. It can be seen how the chart is continued until the next trigger event.
Trigger Position	This option is only available with the "Stop Display" and "Restart Display" trigger actions. The trigger position can be used to select the percentage of the display (display width) before or after the trigger event (for example: positive 0 pass of the green signal at 10 % of 2.5 s = 0.25 s).

Auto delete (for Stop SubSave only)

Auto delete	Defines the mode in which the auto delete function is to run. If set to "Disabled", no files will be deleted. If set to "Capacity", the oldest file created will be deleted if more than the number of files specified in "Delete more than" have been saved. If set to "OlderThan", all files older than the time specified in "Delete older than" will be deleted. The option "CapacityOrOlderThan" is a combination of "Capacity" and "OlderThan", i.e. files are deleted if the conditions for one of the two options are satisfied.
Delete more than	This option defines when and with how many created files the oldest is to be deleted. The option is only active as long as "Capacity" or "CapacityOrOlderThan" is selected in "Auto Delete".
Delete older than	This option defines the number of seconds after which a file is to be deleted with auto delete. The option is only active as long as "OlderThan" or "CapacityOrOlderThan" is selected in "Auto Delete".

Channel Trigger Set

Each channel trigger set can be regarded as a self-contained trigger condition. The action report of the trigger group defines what happens when a trigger group is triggered.
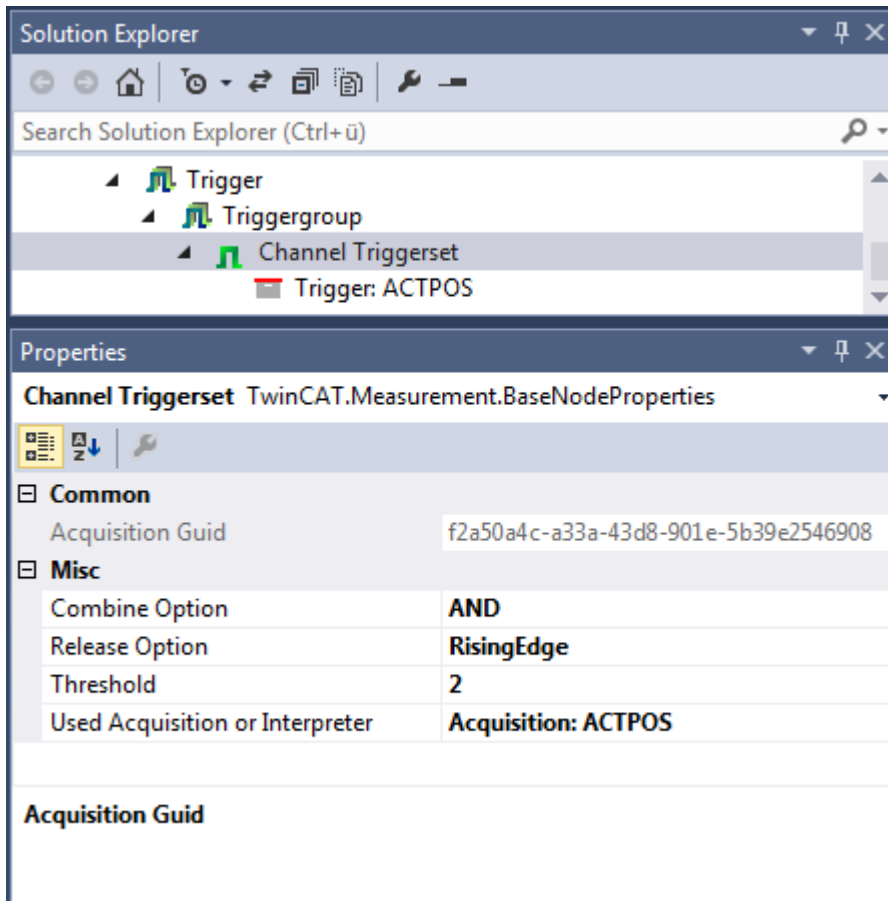
The channel trigger set offers the following setting options:

Misc

Combine Options	By selecting a logic operation these individual conditions can be placed in relation to one another. Note here that AND operations are handled before OR operations.
Release Options	The release condition can be either a rising or falling edge.
Threshold	Corresponding limit value for the rising or falling edge.
Used Acquisition or Interpreter	For each set, an acquisition or an interpreter can be assigned whose state is to be checked for the trigger conditions. (If an acquisition is selected, a new interpreter is created under the channel trigger set.

When a Trigger Set is triggered, the icon in the **Solution Explorer** changes from to . All Trigger Sets remain set until the entire trigger group has tripped.

Directory Trigger Set

A directory trigger set monitors a directory. If an image is added to this folder or optionally also to the subfolders, this image is displayed above a trigger marking. The image is then displayed above the trigger marking. The position of the trigger info is based on the name of the file. The name must be formatted as TwinCAT in order to be displayed correctly.

There are three possibilities to format file names:

"TC_" plus a time stamp in the Windows file time format

"PLC_" plus a time stamp in the PLC format

"DC_" plus a time stamp in the DC format

It is also possible to drag images with a suitably formatted name onto a chart and to display them as triggers. This is also possible in the reply.
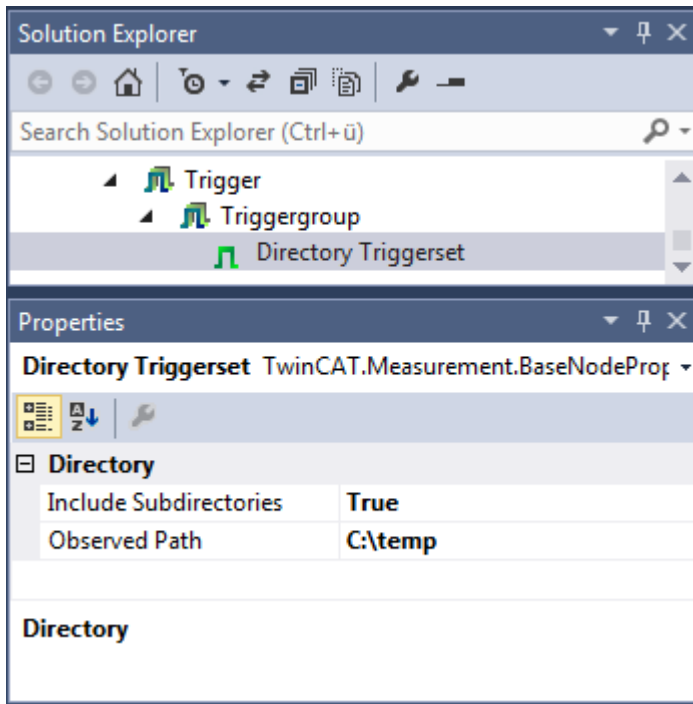
The directory trigger set offers the following setting options:

Directory

Include Subdirectories	If activated, the subfolders of the "Observed Path" will also be included in the monitoring.
Observed Path	Path to a directory that is to be monitored.

Manual triggering of trigger set

Triggers can be manually triggered for test purposes via the Trigger Set context menu command **Manual Trigger Hit**.

Trigger Window

The Trigger window can be opened via the Trigger context menu. It displays the triggers with the times at which they were triggered.

5.1.1.5 Markers

A marker is a chart element that can be docked to an axis. The marker consists of a line that can be found in the chart and a text field that appears alongside the chart and displays the name of the marker.

On clicking on the marker, a tool tip appears that displays the position next to the name.

If the marker was added to a time axis (YT chart) or an index axis (array bar chart and single bar chart), the values of the channel appear in the tool tip.

The values of the different markers can be viewed in the marker window and compared with other markers.
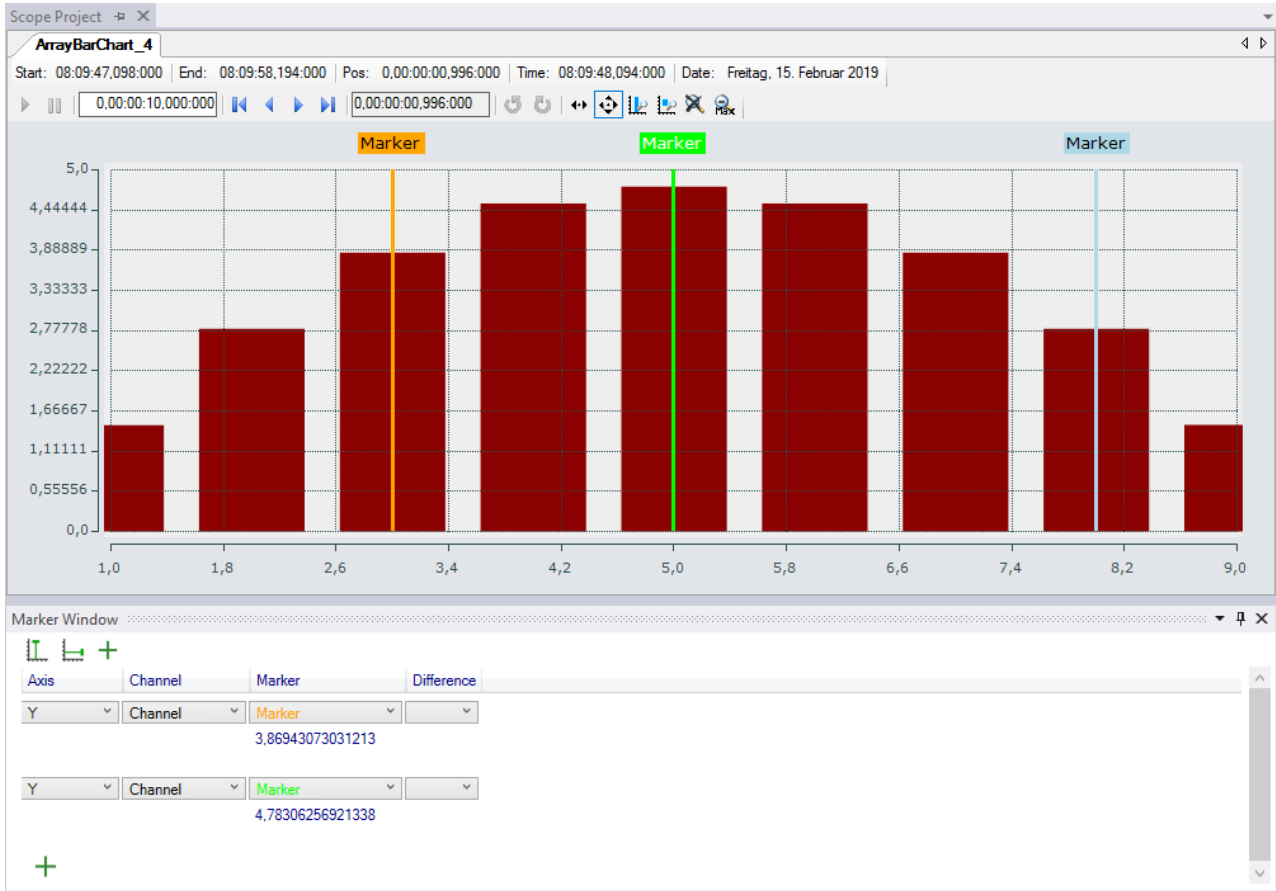
A marker is added via the context menu of a marker container:
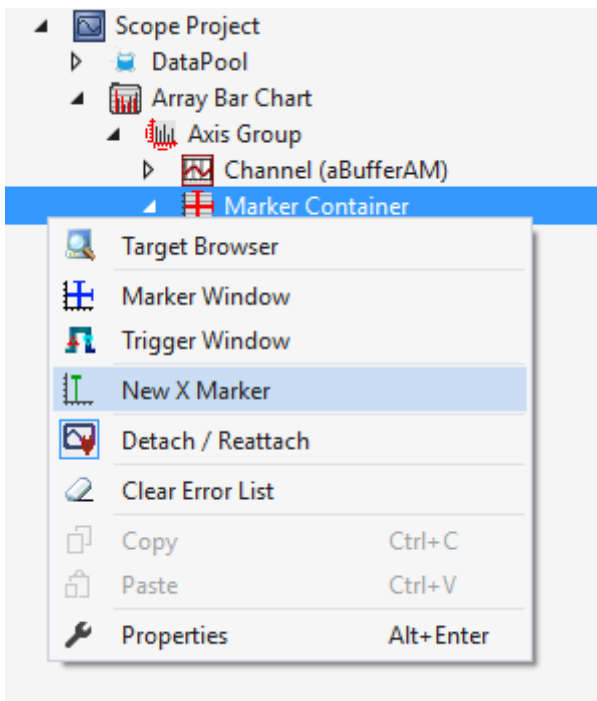
All the properties of a marker are explained here.
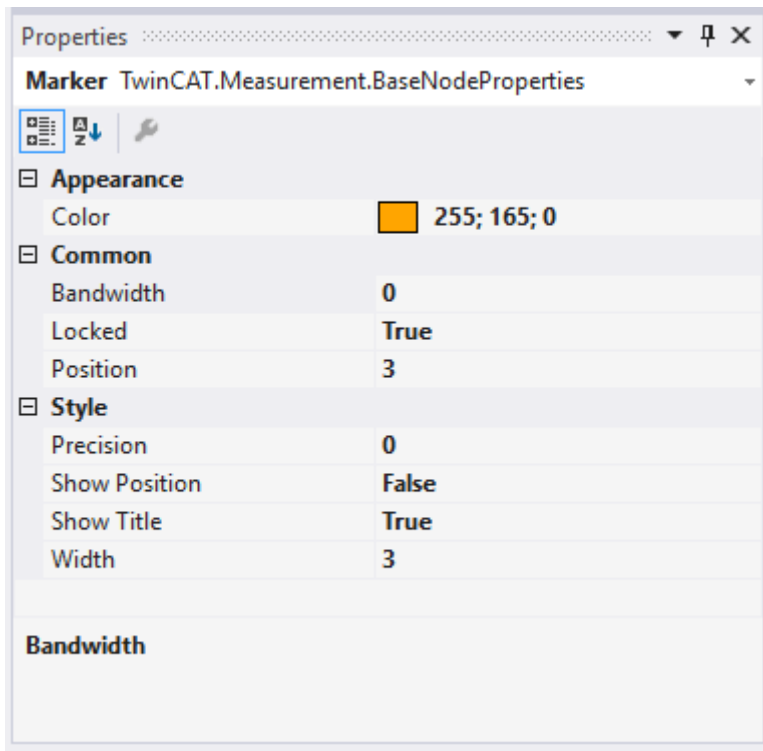
Appearance

Color	Color of a marker.

Common

Bandwidth	Width of the rectangle behind a marker. Specification indicates the width from the marker to the edge of the rectangle. The value of the specification is identical to the input of the position.
Locked	If Locked is set to false the marker can be moved freely within the chart with the mouse.
Position	Position of a cursor. With value and index axes the absolute value of the axis is entered here and with time axes the percentage value.

Style

Precision	Precision of the display values.
Show Position	If this is selected the position is also displayed in the marker label.
Show Title	If Show Title is selected the title will be displayed.
Width	Indicates the width of the marker line in pixels.

The values of the markers can be viewed in the marker window and compared with other markers. A marker window always refers to the axis group selected in the Solution Explorer.
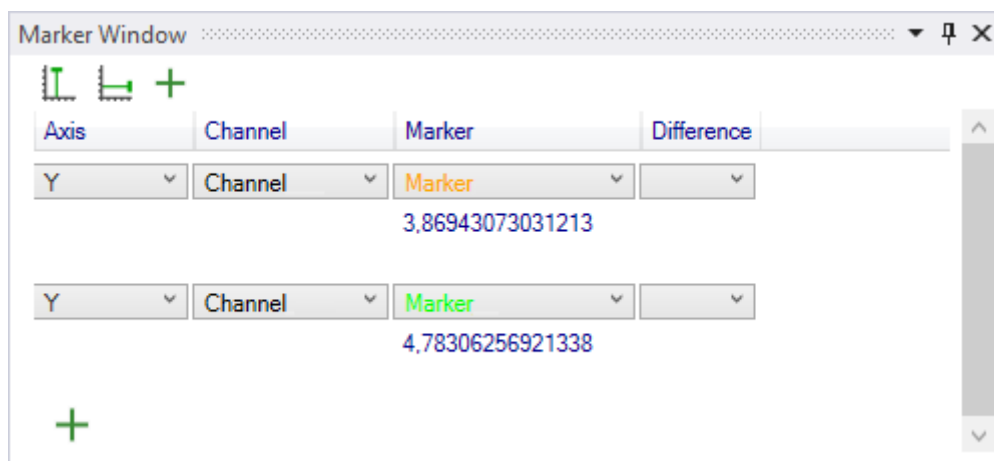
There are three buttons in the top line of the window. With the first one a new X-marker can be created in the current axis group and with the second one a new Y-marker is generated.

A new row can be created in the table with the plus sign.

A row is used to analyze a channel on an axis. Channels and axes can be set in the first two columns. In the third column the point at which the values are read out is selected by choosing the marker. In order to compare this value with values at other positions, various numbers of "difference markers" can be selected in the "Difference" column. If a marker is selected there, a new difference column is automatically generated. The differences always refer to the reference marker in the third column.

5.1.1.6   Dynamic Style

With the aid of dynamic styles the appearance of a channel can be adapted section-wise using an additional data series. Hence sections of the graph can be displayed in a different color, dashed, invisible or thicker.

Any number of dynamic styles can be appended to a graph. Each dynamic style makes a change to a property of the graph (e.g. changing the color to red). Under each dynamic style there must be at least one condition that defines whether this action is to be executed with the current data value. Several conditions are also possible, which are then "ANDed" or "ORed".

A dynamic style is added via the context menu of the channel.

Solution 'TwinCAT Measurement Project24' (1 project)
  ▲ TwinCAT Measurement Project24
    ▲ Scope Project
      ▷ DataPool
      ▲ YT Chart
        ▲ Axis Group

🔍	Target Browser	
⊞	Marker Window	
🎢	Trigger Window	
	Detach / Reattach	
✓	Visible	
✓	Enabled	
🗚	New Dynamic Style	
🗗	Copy	Ctrl+C
🗐	Paste	Ctrl+V
✗	Delete	Del
🔧	Properties	Alt+Enter
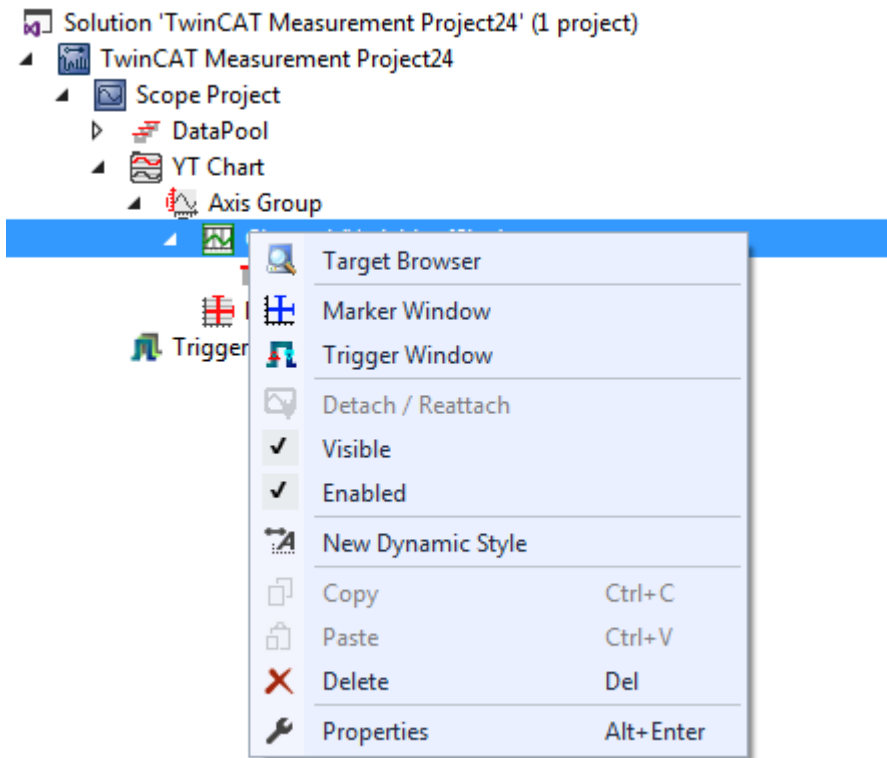
If a channel contains at least one dynamic style, then a new entry "Acquisition" is displayed in the "Dynamic Style" group in the Properties window of the channel. The data series for all dynamic styles can be defined here. This must be recorded by the same target system and sampled with the same sampling rate as the other data series of the channel.

Properties ▾ 📌 ✕

Channel (Variables.fSine) TwinCAT.Measurement.BaseNodeProperties ▾

🔳 🗚 🔧

⊟ Common	
Enabled	**True**
Visible	**True**
⊟ Dynamic Style	
Acquisition	**Acquisition: Variables.fPeak**
⊟ Line	
Antialias	**True**
Fill Color	🟩 **Green**
Fill Mode	**None**
Graph Type	**Extended**
Line Color	🟩 **Green**
Line Width	1
⊟ Marks	
MarkColor	🟩 **Green**
MarkSize	2
MarkState	**Auto**
⊟ Min/Max	
Show Max	**False**
Show Min	**False**
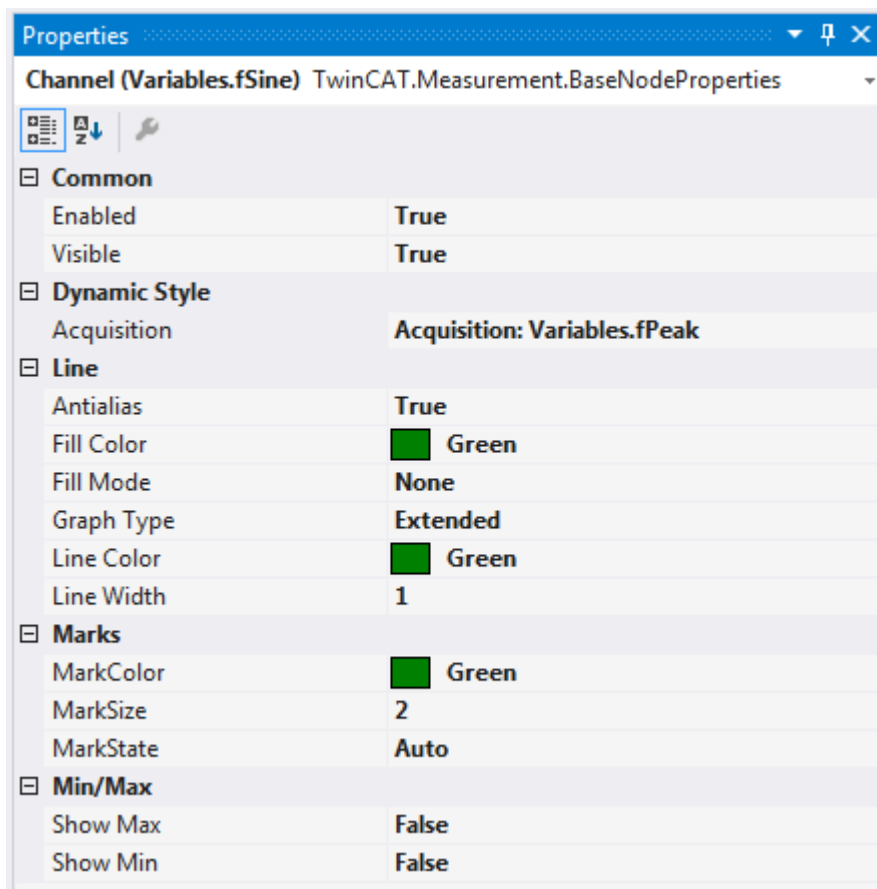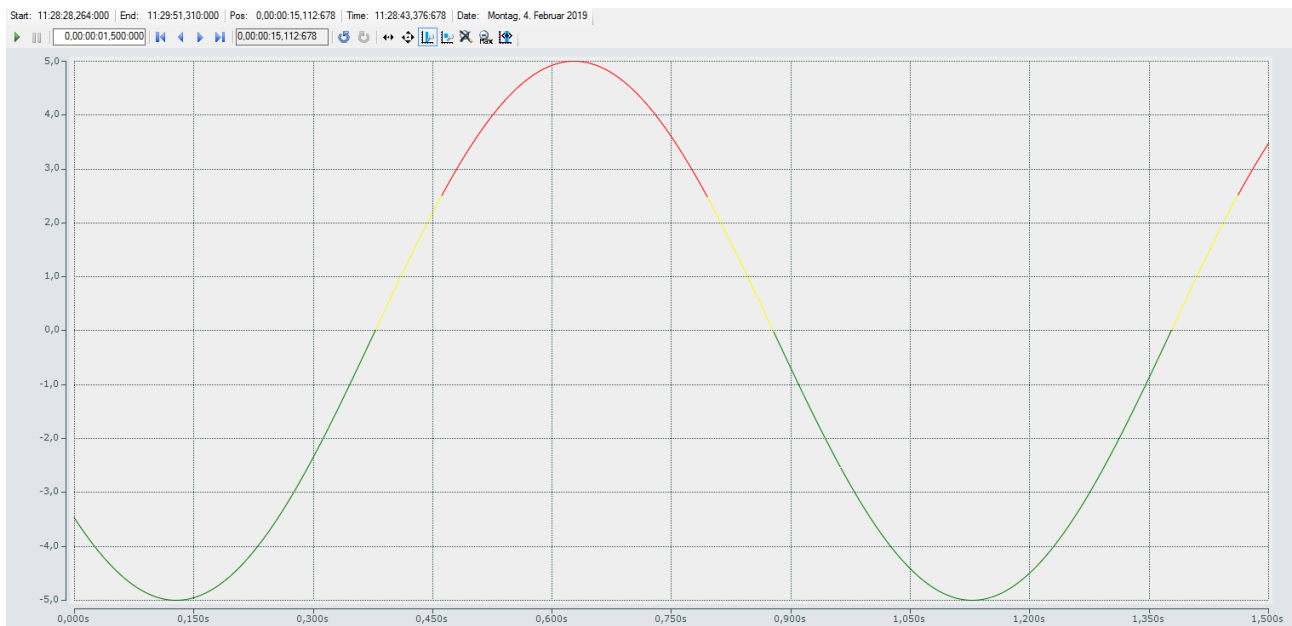
Dynamic styles can be used on channels in XY charts as well as on channels in YT charts. Hence there are a great many application possibilities. Also in the motion area for the illustration of contours or simple alarm states.

5.1.1.6.1 Dynamic style properties

All settings assigned to the dynamic style can be made here.
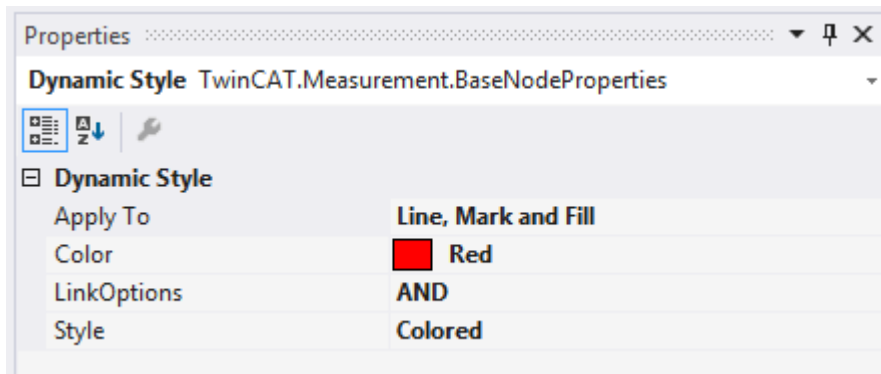
Apply To	Defines which part(s) of the graph will be changed by the dynamic style. (Line, marker and/or fill area)
Color	Defines the color that will be used when changing color. (Visible only if Style is set to "Colored").
LinkOption	Specifies how the different conditions underneath the dynamic style are to be linked ("and" or "or")
Style	Specifies how the dynamic style should influence the graphs: • Normal → no change • Bold → bold • Dashed → dashed • Colored → other color • Invisible → invisible

5.1.1.6.2 Dynamic Style Condition

There are two different conditions in TwinCAT 3 Scope. One is the Threshold condition, which checks whether a value lies above or below a limit. The other is the Area condition, which checks whether a value lies within a range. The conditions can be added via the context menu of the dynamic styles:
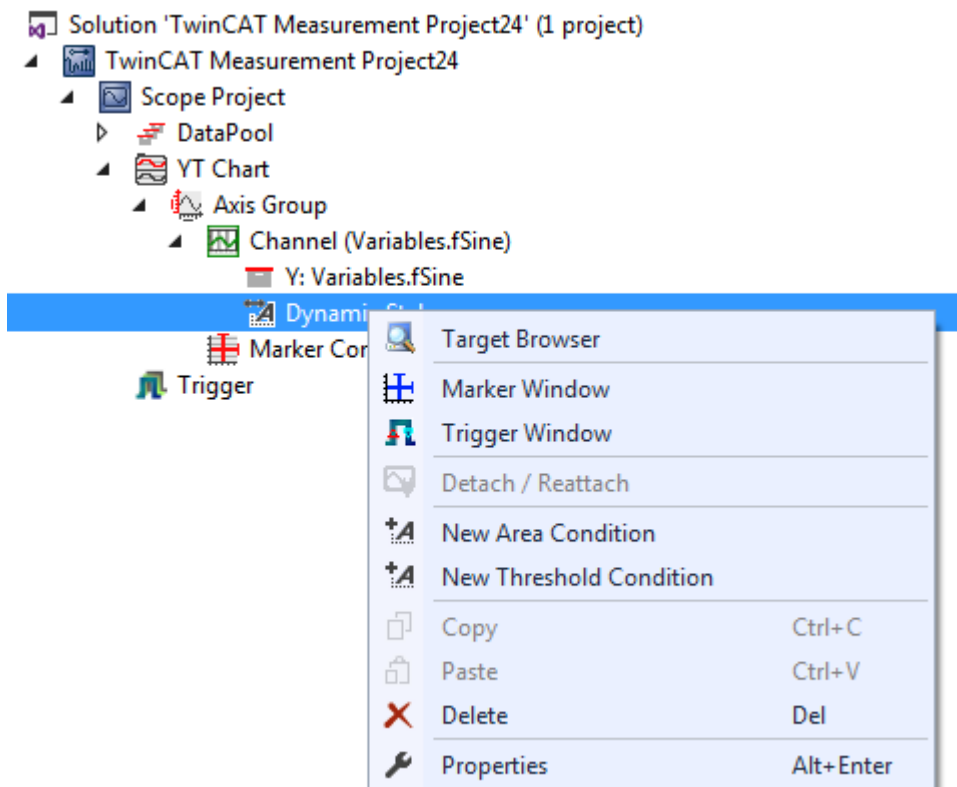
Threshold condition

A Threshold condition has the following properties:

Enabled	Activates this condition
Threshold	Sets the limit value for switchover
Operator	Specifies the comparison operator between limit value and comparison value (smaller than, smaller than/equal to, greater than/equal to, or greater than)

Area condition

An Area condition has the following properties:

Enabled	Activates this condition
Lower Limit	The lower limit of the area to be checked
Lower Operator	Comparison operator for the lower limit (greater than or greater than/equal to)
Upper Limit	The upper limit of the area to be checked
Upper Operator	Comparison operator for the upper limit (smaller or smaller than/equal to)

5.1.1.7 Shapes

The shapes form a new subgroup of the graph element. They are created under the axis group in an XY chart.

There are two types of shape:

- Angular shapes (defined by coordinates)
- Round shapes (defined by a center point and two radii)
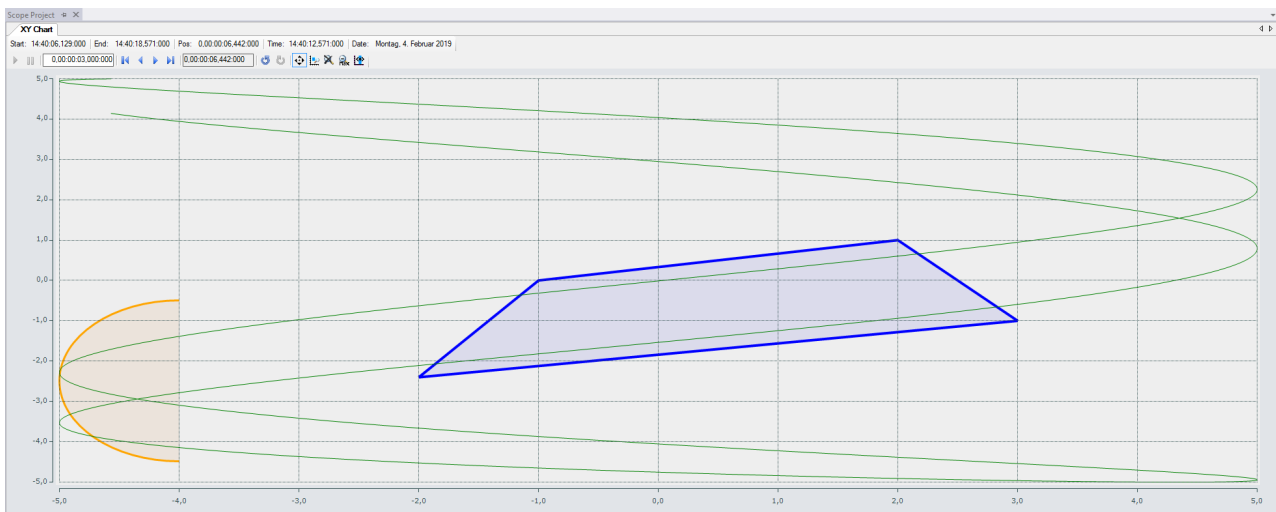
A shape is added via the context menu of an axis group:
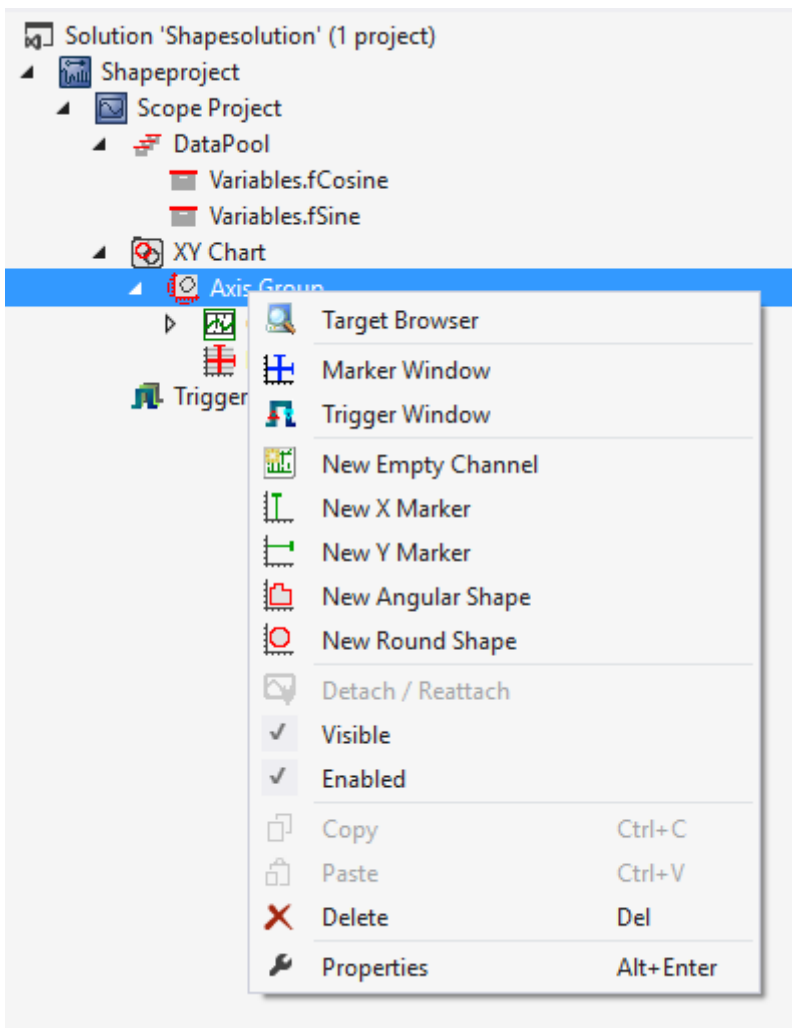
5.1.1.7.1 Angular shape properties

All the properties of an angular shape are explained here.
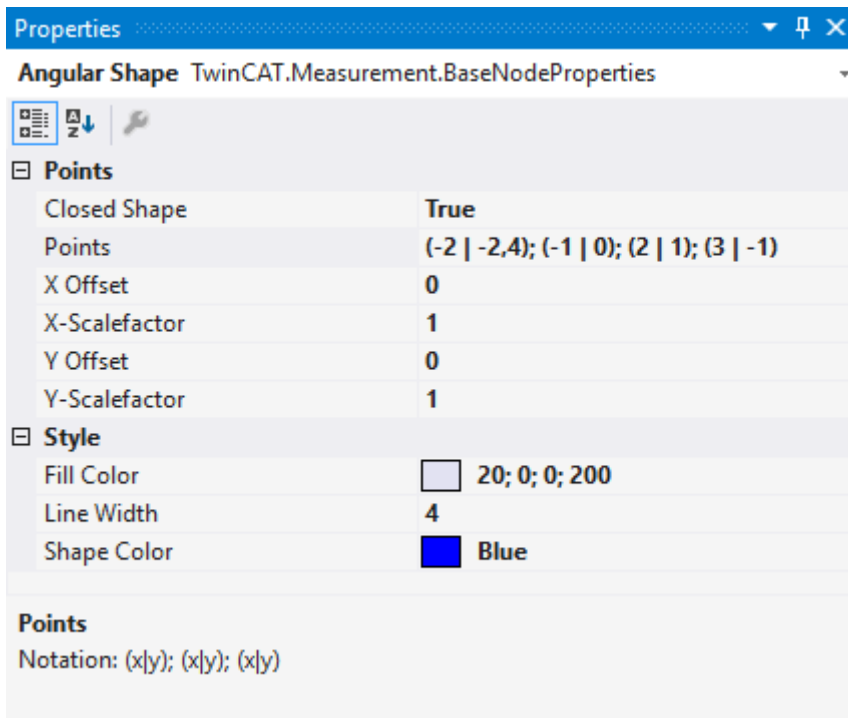
Points

Closed Shape	If Closed Shape is selected, the last point is connected to the first point again.
Points	List of all points. (Notation: (x|y);(x|y))
X Offset	In the offset field a free offset can be added to the coordinates of the shape in the X-direction.
X-Scalefactor	The points of the shape can be changed in the X-direction with the scale factor.
Y Offset	In the offset field a free offset can be added to the coordinates of the shape in the Y-direction.
Y-Scalefactor	The points of the shape can be changed in the Y-direction with the scale factor.

Style

Fill Color	Color of the area content.
Line Width	Width of the edges in pixels.
Shape Color	Color of the edge.

5.1.1.7.2   Round shape properties

All the properties of a round shape are explained here.
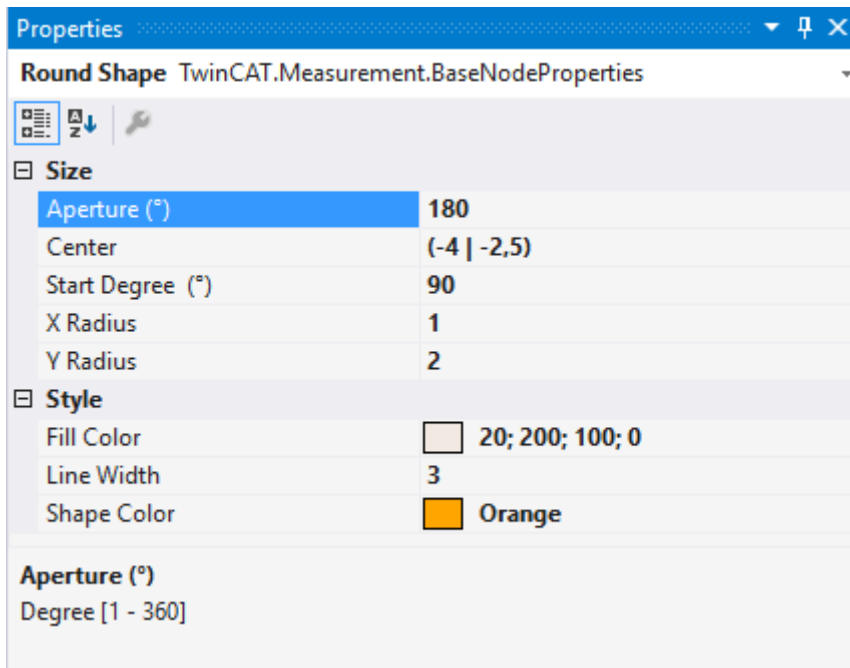
Size

Aperture	The aperture specifies how wide the circle will be drawn.
Center	Center point of the circle.
Start Degree	Starting angle from which the circle will be drawn.
X Radius	Radius of the circle in the X-direction.
Y Radius	Radius of the circle in the Y-direction.

Style

Fill Color	Color of the area content.
Line Width	Width of the edges in pixels.
Shape Color	Color of the edge.

5.1.2   Communication

In principle the TE1300 and TF3300 documentation refers to the TwinCAT-specific communication via ADS. The TwinCAT3 Scope itself is also capable of receiving data via other communication channels and displaying them. For example via OPC UA. In this section the differences relating to the communication channels are illustrated and explained.

5.1.2.1   OPC UA

In addition to the standard communication protocol ADS, the TwinCAT 3 Scope View offers the option of transmitting measured data via the standardized communication channel OPC UA. This is made possible by an OPC UA Client implemented in the TwinCAT 3 Scope Server and has the advantage that data can also be recorded from third-party suppliers' systems and analyzed (see illustration).
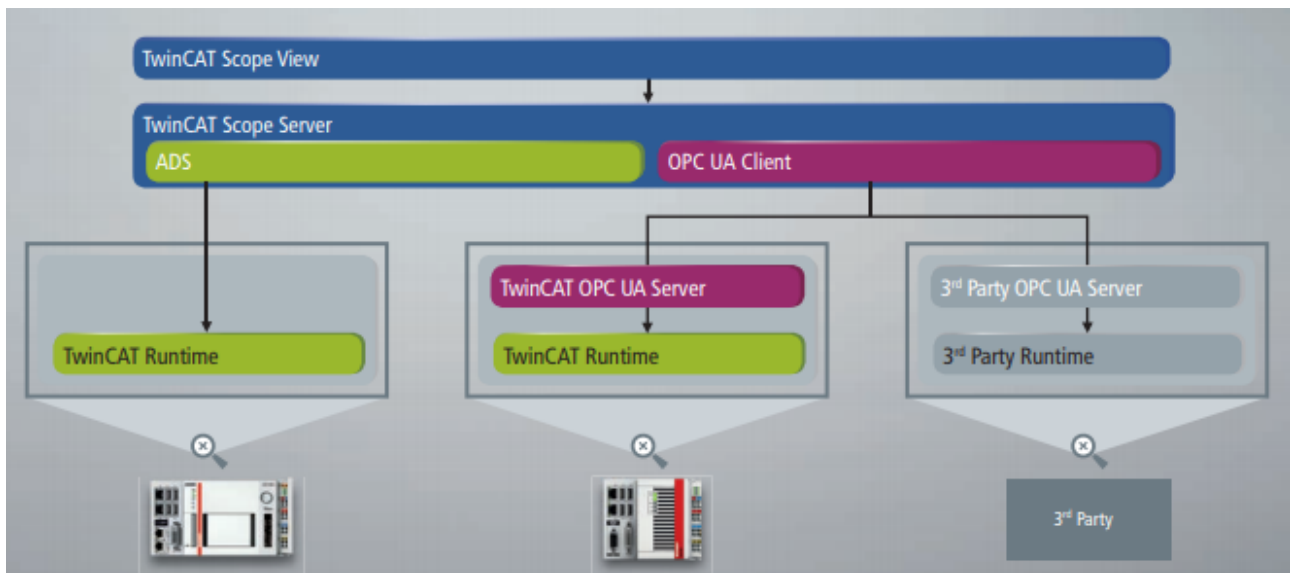
In the following, the special features of communication via OPC UA are explained in comparison with ADS communication.

5.1.2.1.1   Addition of OPC UA symbols

In order to record symbols via OPC UA with the TwinCAT 3 Scope View, they must first be added to the Scope project with the aid of the Target Browser, as is also the case with ADS.

In the **Target Browser** on the **OPC UA** tab you can add any number of target systems on which an OPC UA Server runs. The associated symbols are then listed in a hierarchical tree structure in the right-hand area of the Target Browser and can be added to the Scope project by drag-and-drop. (See also Extension - OPC UA [▶ 107])

5.1.2.1.2   Acquisition of OPC UA symbols

The acquisition of OPC UA symbols differs in part from the ADS acquisition. Since the properties of the groups "Common", "Symbol" and "Target" are identical to the ADS acquisition, only the specific OPC UA properties are explained in the following.

Properties
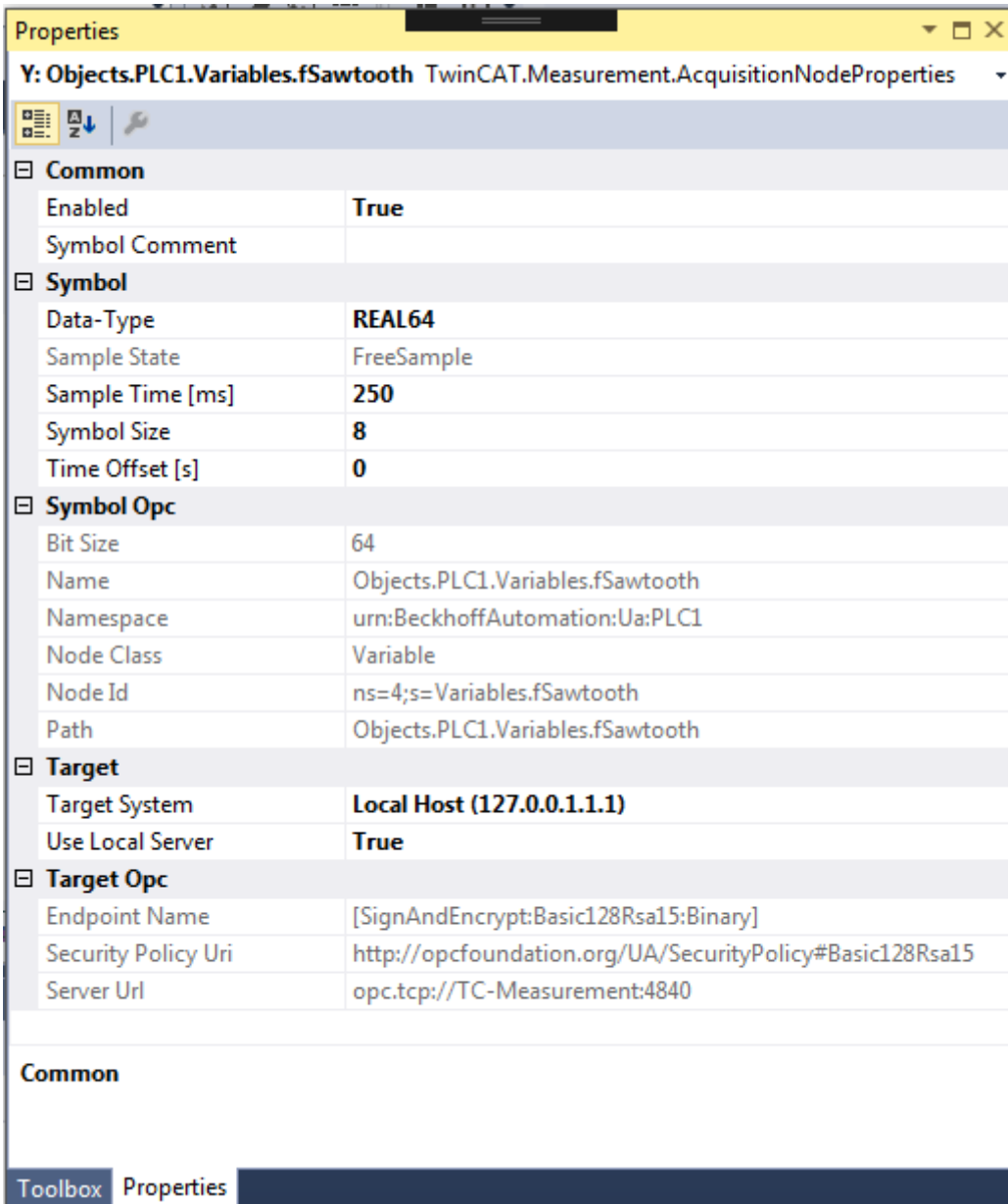
Symbol Opc

Bit Size	Size of the symbol in bits
Name	Name of the symbol
Namespace	Namespace of the symbol
Node Class	Name of the node to which the symbol is assigned in the hierarchy
Path	Path to the symbol

Target Opc

Endpoint Name	Name of the selected OPC UA end point. Contains information about the encryption method and signing.
Security Policy Uri	URI of the selected encryption method
Server URL	URL of the OPC UA Server from which the symbol is to be recorded.

See also: Properties

5.1.2.1.3 Recording of OPC UA symbols

i The function of recording symbols via OPC UA is available only with a Scope View Professional license.

In principle the recording of signals via OPC UA can be started as usual via the **Start Record** button.

However, you should note the following special features:

- **Sample Time:** When recording OPC UA symbols there is no task sample time as with ADS symbols. The sample state is thus always set to "Free Sample". Therefore, set the desired sample time before starting the recording. Select only sampling rates that are also supported by the respective OPC UA Server. Otherwise an error message will appear when starting the recording and the sample time will be changed automatically to the sample time returned by the OPC UA Server.

- **Oversampling:** For arrays it is also possible to use the oversampling function with OPC UA. However, since there is no task sample time and the oversampling actually refers to the task sample time, the interpretation does not correspond to the conventional oversampling with ADS. In the interpretation of the measured data this must be taken into account in particular with free sample times, which deviate widely from the actual task sample time.

- **Encrypted and/or authenticated connections:** If the measured data are recorded by means of an encrypted and/or authenticated OPC UA connection, some further steps have to be performed following the first actuation of the **Start Record** button. More precise information on this can be found in the sections Encrypted connections [▶ 82] and Authenticated access [▶ 84].

- **Determinism:** The intervals between the data points that are transmitted via OPC UA correspond approximately to the set sampling rate. However, it is not necessarily to be expected that all data points are really equidistant as in the case of ADS. There may very well be deviations of 1-2 ms.

- **Missing data:** If "gaps" should occur in the data when recording data via OPC UA (the interval between the data points is not equal to the set sample time), this may be due to the respective OPC UA Server and/or the quality of the connection. In this case, check the Scope Server log if necessary.

5.1.2.1.4 Encrypted connections

OPC UA offers the possibility to implement a secure communication channel via encrypted connections. Even before adding the symbol you can select the encryption method via the respective end point during the creation of the OPC UA Server in the **Target Browser** (see also: Adding an OPC UA Server [▶ 108]).
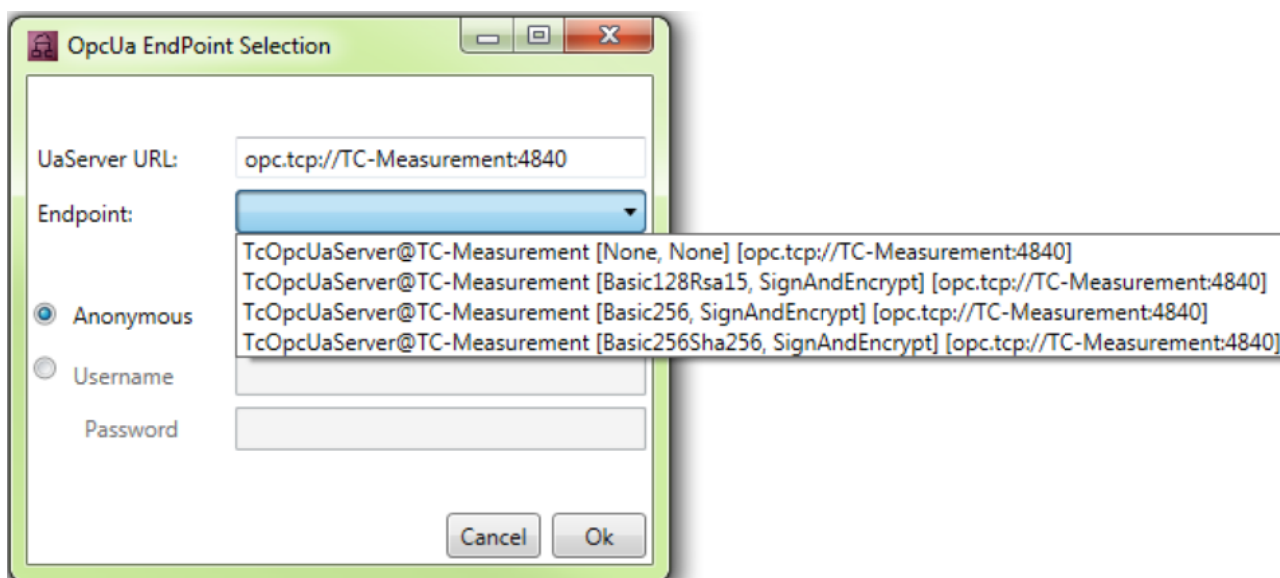
If you add the symbol to the Scope project, the selected encryption method is automatically adopted (see also: Addition of OPC UA symbols [▶ 80]).

As soon as you click on **Start Record** to start the recording, a dialog appears requesting you to adjust the Scope Server configuration.
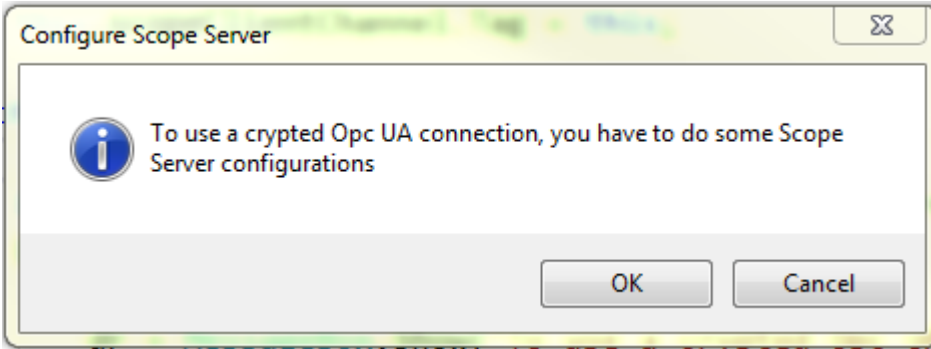
In order to make the necessary configurations on the Scope Server so that the recording can be started, confirm the dialog. The **Scope Server – Opc Ua Configuration** window opens in which you are requested to trust the certificate of the OPC UA Server, with which the communication is to take place, on the client side (Scope Server) (**Trust** button).
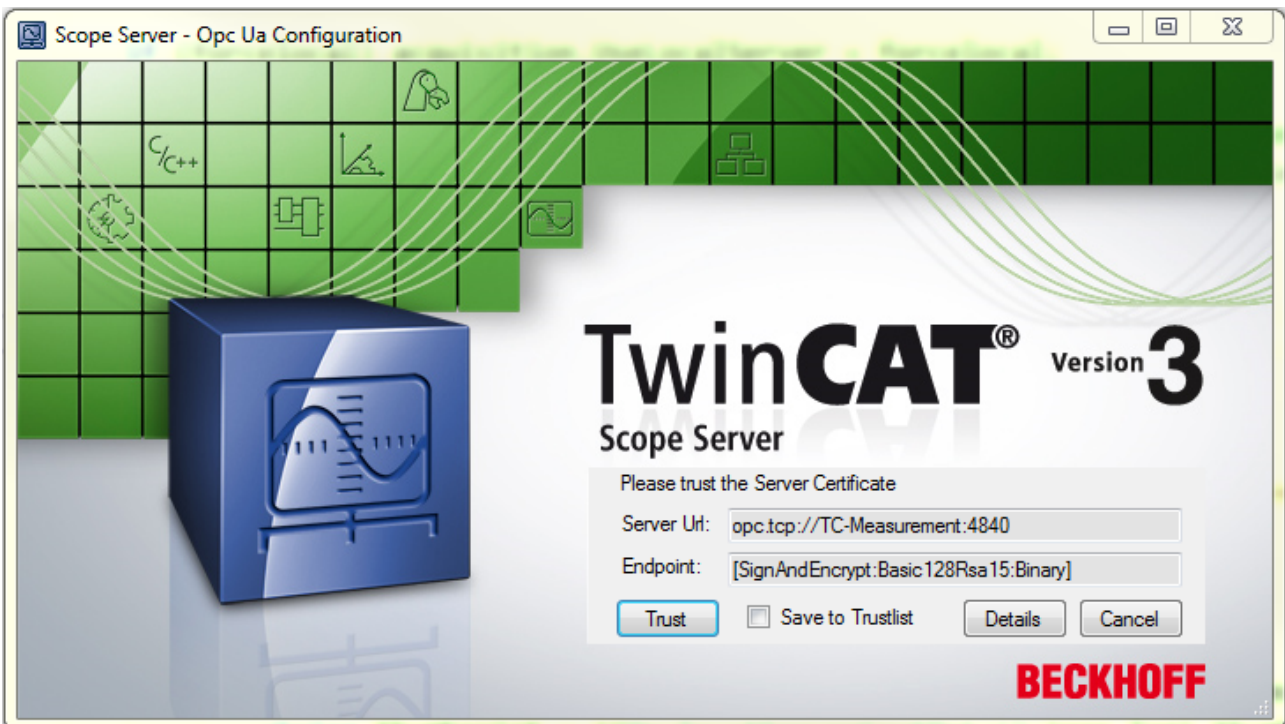
You can trust the certificate in a single case or add it to the list of trusted certificates via the check box **Save to Trustlist**.

In order to check the server certificate, the server URL of the OPC UA Server and the selected end point have to be entered in the window. You can open the detailed information about the certificate via the **Details** button.
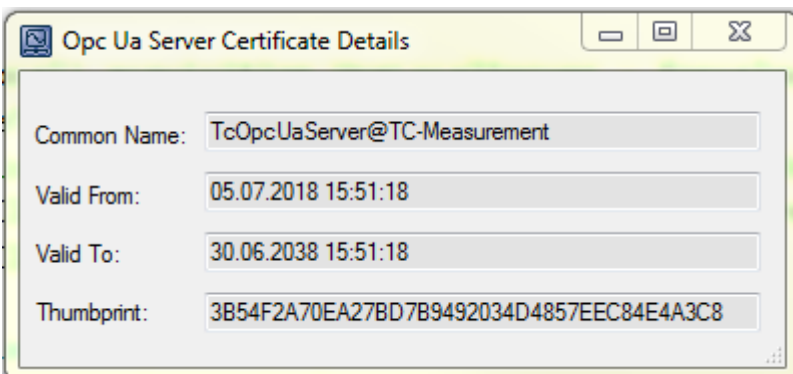
In addition to the "Common name" and the period of validity of the certificate ("Valid from/to"), the "Thumbprint" is of particular relevance. On the basis of the thumbprint you can determine whether the certificate is really that of the OPC UA Server to which you wish to connect.

During the first attempt to connect to an OPC UA Server it is additionally necessary to trust the certificate of the client (Scope Server) on the server side. The following error message informs you: „Connection to OPC UA Server failed. Please trust the client certificate at OPC UA Server." In order to trust the certificate of the client on the server, copy the respective certificate in the certificate directory of the OPC UA Server from the "rejected" folder to the "trusted" folder.

If the certificates have been exchanged on both sides and you have trusted both, you can start the recording.

5.1.2.1.5 Authenticated access

OPC UA offers the option of an authenticated access. To do this the user must log in with his user data (user name and password) in order to be able to communicate data via OPC UA. This is necessary, for example, if various user accounts have been set up on the target device on which the OPC UA Server is running and these accounts are locked via an authentication, or if various authorizations are to be implemented for different data.

In a similar way to the encrypted connections, you define the identity type (anonymous/authenticated) before adding the symbol when creating the OPC UA Server in the **Target Browser**. For an authenticated access, activate the "Username" option instead of the "Anonymous" option and enter a user name and a password (see also: Adding an OPC UA Server [▶ 108]).
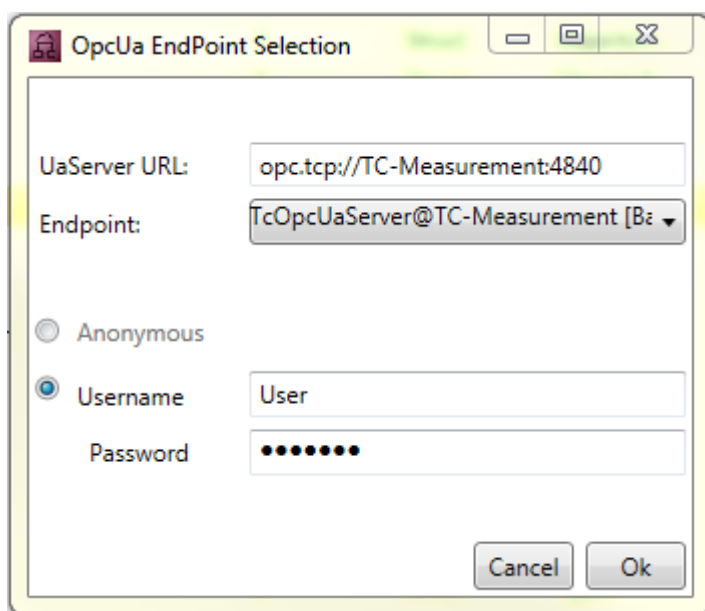
When you add the symbol to the Scope project, the identity type and the user name are transferred with it (see also: Addition of OPC UA symbols [▶ 80]). Before starting the recording the password is requested again for security reasons.

As soon as you click on **Start Record** to start the recording, the window **Scope Server – Opc Ua Configuration** opens, in which you are requested to enter the password. The user name that was entered in the **Target Browser** appears automatically.
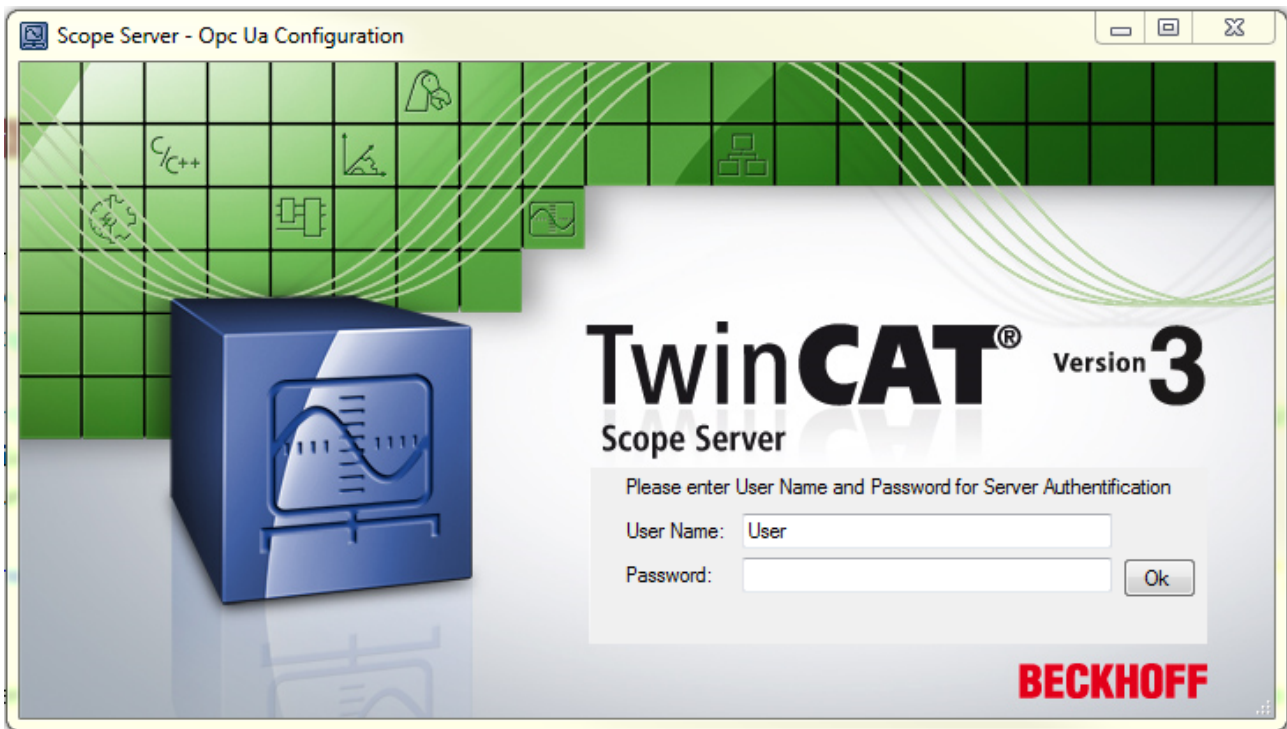
The recording is started after confirming the entry via **OK** and correct authentication.

If an incorrect password is entered, the connection attempt is regarded as having failed ("Wrong credentials. Start Record not possible") and you are given the option to enter the password again.

5.1.3 Units

Measured values can be recorded and displayed as a graph with the aid of the TwinCAT 3 Scope. To make interpretation of the recorded data easier, the TwinCAT 3 Scope offers the option of setting a physical unit for each individual channel. Firstly, this enables the unit in which the measured values were recorded to be made known. Secondly, the data can be converted into other units so that the user has the option to view the data, meaningfully scaled, in a unit that he selects.

5.1.3.1 Introduction

Base units

The units in TC3 Scope are based on the SI system of units. Both the seven standard SI units and all units that can be derived from the product of any powers of these seven standard SI units are designated base units. These base units are coded in TC3 Scope via unique Enum values. To do this, four bits are interpreted as a signed value for each power value of the base units and combined to form a 32-bit integer (Int32), so that any derived units (limited only by the size of the data type) can be formed. Each digit of the Enum value represents one of the base units so that a unit is formed from the following equation in the case of an Enum value of "0xabcdefgh":

$E = Length^a * Mass^b * Time^c * Current^d * Temperature^e * Amount\ of\ Substance^f * Luminous\ Intensity^g * Angle^h$

The following table shows the basic values for the calculation of the derived units.

Measured variable	Name of the unit	Symbol of the unit	Enum value
Dimensionless values	Unit of One	1	0x00000000
Length	Meter	m	0x00000001
Mass	Kilogram	kg	0x00000010
Time	Second	s	0x00000100
Current	Ampere	A	0x00001000
Temperature	Degree Celsius	°C	0x00010000
Amount of Substance	Mole	mol	0x00100000
Luminous Intensity	Candela	cd	0x01000000
Angle	Radian	rad	0x10000000

In addition to the seven standard SI units, two further units are listed here that are relevant for practical use. One of them is the "Unit of One" for dimensionless variables, which also serves as the default value if no other unit has been set. On the other hand, the angle is also considered, since there are several units such as the angular velocity or the solid angle that are composed of the powers of the angle, even though the angle is actually dimensionless. A further peculiarity is mass with its SI unit of kilogram, because unlike the other SI units it already contains a prefix in its basic form. Therefore, in order to be able to carry out scaling with the aid of prefixes, gram is assumed as the unit of mass in TC3 Scope View instead of kilogram. For the derived units, however, the mass percentage is nevertheless represented as kilograms, as the definition would otherwise no longer fit.

In order to form units derived from these basic values, the powers of the standard SI units must be multiplied accordingly. For example, if a force is to be illustrated in Newton, the rule is: $N = m^1 * kg^1 * s^{-2}$, resulting in an Enum value of 0x00000E11. E represents the power of "-2" here, which results from the display as two's complement in the hexadecimal system. All the derived base units already predefined in TC3 Scope are listed below.

Measured variable	Name of the unit	Symbol of the unit	Enum value
Absorbed Dose	Gray	Gy	0x00000E02
Acceleration	Meter per square second	$m * s^{-2}$	0x00000E01
Angular Acceleration	Radian per square second	$rad * s^{-2}$	0x10000E00
Angular Velocity	Radian per second	$rad * s^{-1}$	0x10000F00
Area	Square meter	m^2	0x00000002
Capacity	Farad	F	0x000024FE,
Catalytic Activity	Katal	kat	0x00100F000
Charge	Coulomb	C	0x00001100
Conductance	Siemens	S	0x000023FE
Energy	Joule	J	0x00000E12
Force	Newton	N	0x00000E11
Frequency	Hertz	Hz	0x00000F00
Illuminance	Lux	lx	0x0100000E
Inductance	Henry	H	0x0000EE12
Luminous Flux	Lumen	lm	0x21000000
Magnetic Flux	Weber	Wb	0x0000FE12
Magnetic Flux Density	Tesla	T	0x0000FE10
Power	Watt	W	0x00000D12
Pressure	Pascal	Pa	0x00000E1F
Resistance	Ohm	Ω	0x0000ED12
Solid Angle	Steradian	sr	0x20000000
Velocity	Meter per second	$m * s^{-1}$	0x00000F01
Voltage	Volt	V	0x0000FD12
Volume	Cubic meter	m^3	0x00000003

Any other base units can be formed by means of the Enum values (see also Configuration of units with the aid of PLC attributes [▶ 100]).

However, the specification of the base unit is insufficient in order to represent a unit completely and to scale the measured values in a suitable manner afterwards. Therefore, further parameters need to be specified: a scaling factor, an offset and a prefix. The scaling factor and the offset are used to convert the raw values into the appropriate unit. The prefix offers the option to scale a unit by the power of ten. The following prefixes are available. For example, if "milli" is selected as the prefix, the respective unit is divided by 10^{-3} so that automatic adjustment of the scaling takes place.

Name of the prefix	Symbol of the prefix	Multiplication factor	Enum value
Yocto	y	10^{-24}	-24
Zepto	z	10^{-21}	-21
Atto	a	10^{-18}	-18
Femto	f	10^{-15}	-15
Pico	p	10^{-12}	-12
Nano	n	10^{-9}	-9
Micro	µ	10^{-6}	-6
Milli	m	10^{-3}	-3
Centi	c	10^{-2}	-2
Deci	d	10^{-1}	-1
None	-	10^{0}	0
Deca	da	10^{1}	1
Hecto	h	10^{2}	2
Kilo	k	10^{3}	3
Mega	M	10^{6}	6
Giga	G	10^{9}	9
Tera	T	10^{12}	12
Peta	P	10^{15}	15
Exa	E	10^{18}	18
Zetta	Z	10^{21}	21
Yotta	Y	10^{24}	24

Transformations

In the measurement technology application area, it is frequently the case that an SI unit has to be converted into another SI unit. This is the case, for example, when an acceleration is measured with the aid of a sensor and a fieldbus terminal. Voltages or currents are initially measured, which should give an indication of the acceleration. For this reason, the TC3 Scope also provides for the transformation from one SI unit into another SI unit. Such a transformation is defined by a source variable (source unit), a target variable (target unit), a scaling factor for the conversion and a name.

Example:

- Source variable: Voltage
- Target variable: Acceleration
- Scaling factor: 0.101937
- Name: Accelerometer

User units

In addition to the SI system of units there are further systems of units that are also used in practice (CGS system of units, Anglo-American system of measurement, etc.). In order to be able to represent such units too, the TC3 Scope provides for so-called user units into which the base units can be converted. A user unit is defined through its associated basic variable, the name and the symbol of the user unit, a scaling factor and an offset for conversion as well as a prefix. For the user unit gravitation (g), for example, this results in the following values:

- Basic variable: Acceleration
- Name: Gravity
- Symbol: g
- Scaling factor: 1 / 9.81
- Offset: 0
- Prefix: none

Resulting unit

A unit in the TC3 Scope can thus be composed of the three elements base unit, transformation and user unit, depending on what is required in the specific application case. However, it is always the resulting unit that is displayed. If a voltage is the basic variable, which is then converted via a transformation into an acceleration in m/s^2 and from there into g with the aid of a user unit, the measured data and offset with the scaling factors and offsets and the result is the acceleration in g.

The following illustration shows how the resulting unit is displayed in the Properties window:
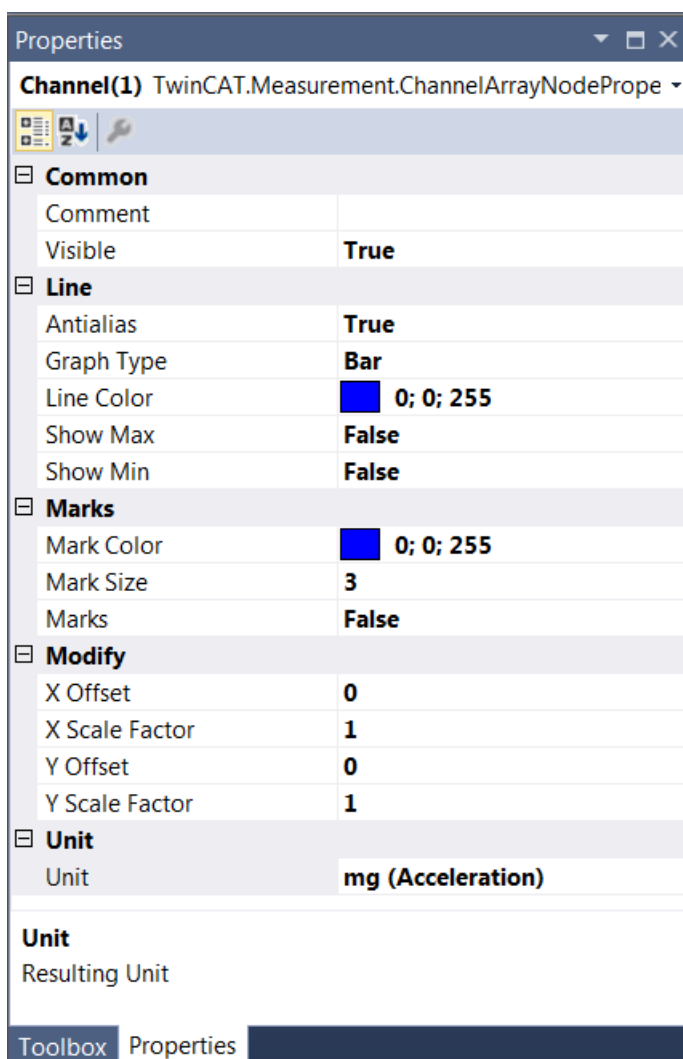
If existent, the symbol for the prefix will be displayed first. After that comes the symbol for the unit and subsequently the measured variable in brackets. Apart from the display in the **Properties** window, the units also appear in the tooltips that appear when clicking on a data point (see illustration) and are displayed in the cursor window.
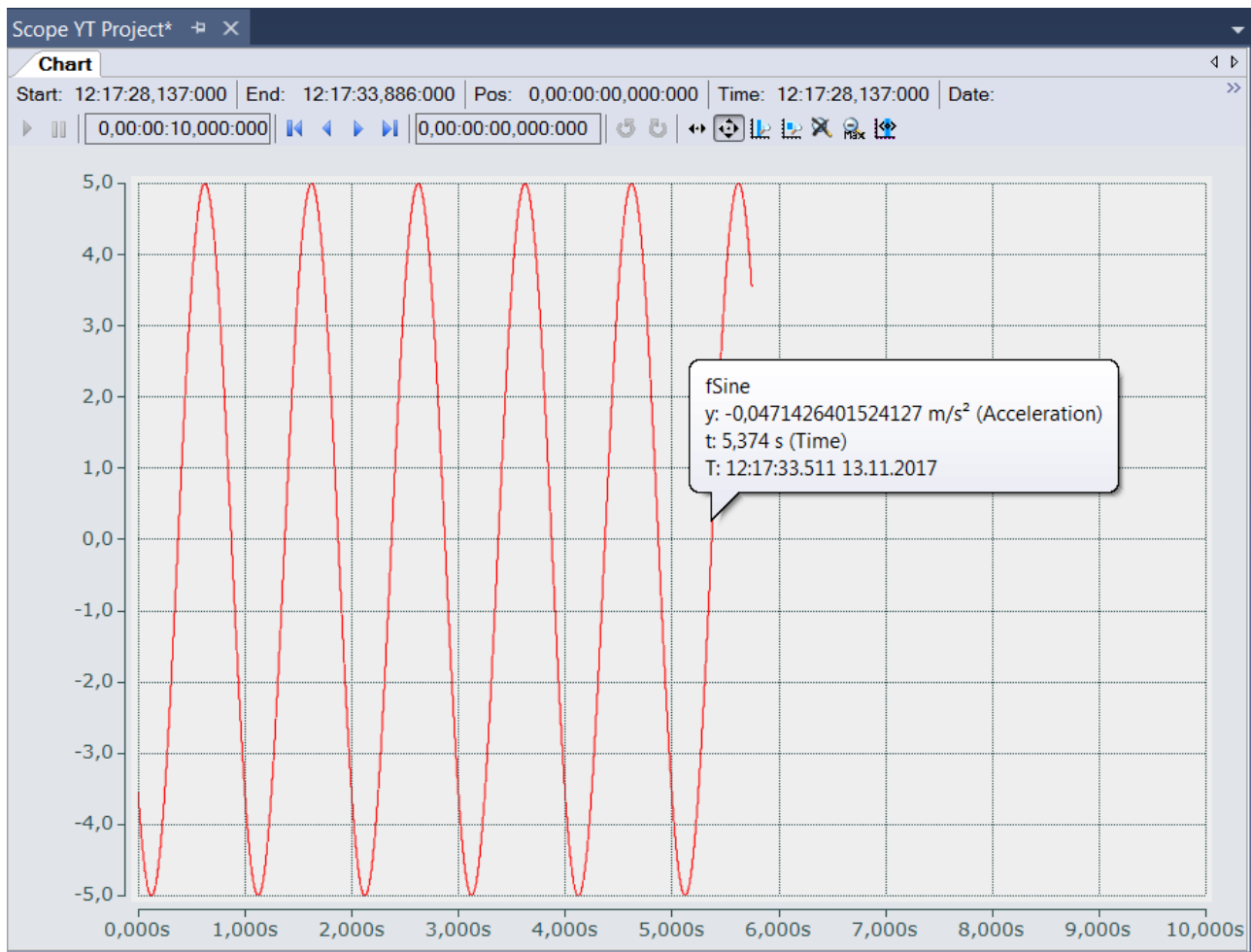
5.1.3.2   Configuring units with the aid of the Unit Wizard

The units can be set for each individual channel in the TC3 Scope. They are displayed in the **Properties** window for the channel style and can also be configured there with the aid of the Unit Wizard.

To open the Unit Wizard, click on the button with the three points on the right of the field.
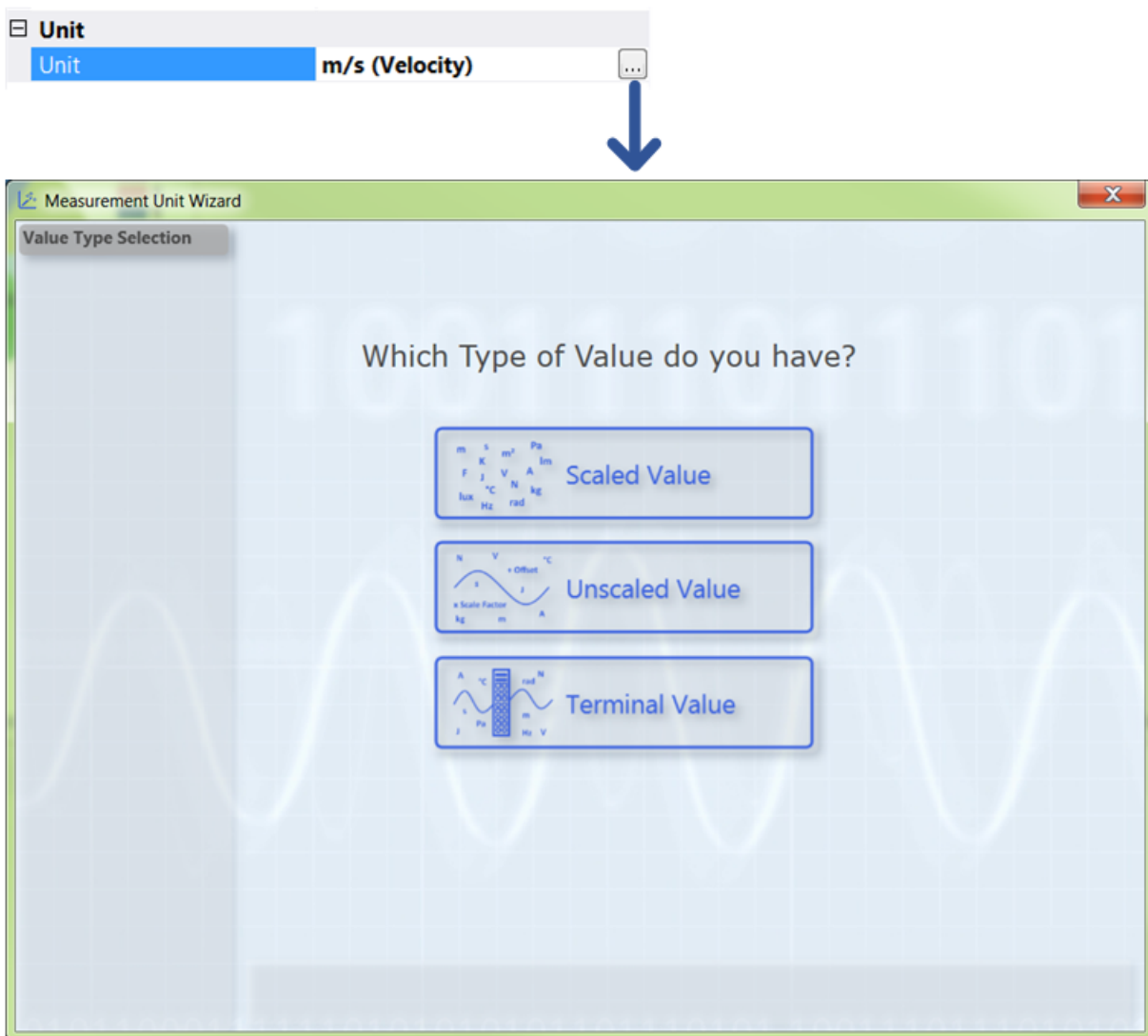
On the first page of the wizard there are three selection options between which you can choose, depending on the type of values you have measured and recorded with the TC3 Scope.

- Scaled Value**:** If you have already scaled your values as desired and only wish to enter the appropriate unit, select the option "Scaled Value" [▶ 90].
- Unscaled Value**:** If you have not yet scaled your values and wish to scale them yourself with the aid of a scaling factor and an offset in addition to selecting the unit, select the option "Unscaled Value" [▶ 91].
- Terminal Value**:** If you have measured your values using a Beckhoff fieldbus terminal and now wish to scale the values automatically through the setting of the hardware employed and to generate a unit, select the option "Terminal Value" [▶ 95]. (However, this option is available only with the Scope View Professional License).

Scaled Value

If you have selected the option "Scaled Value", the following page opens:
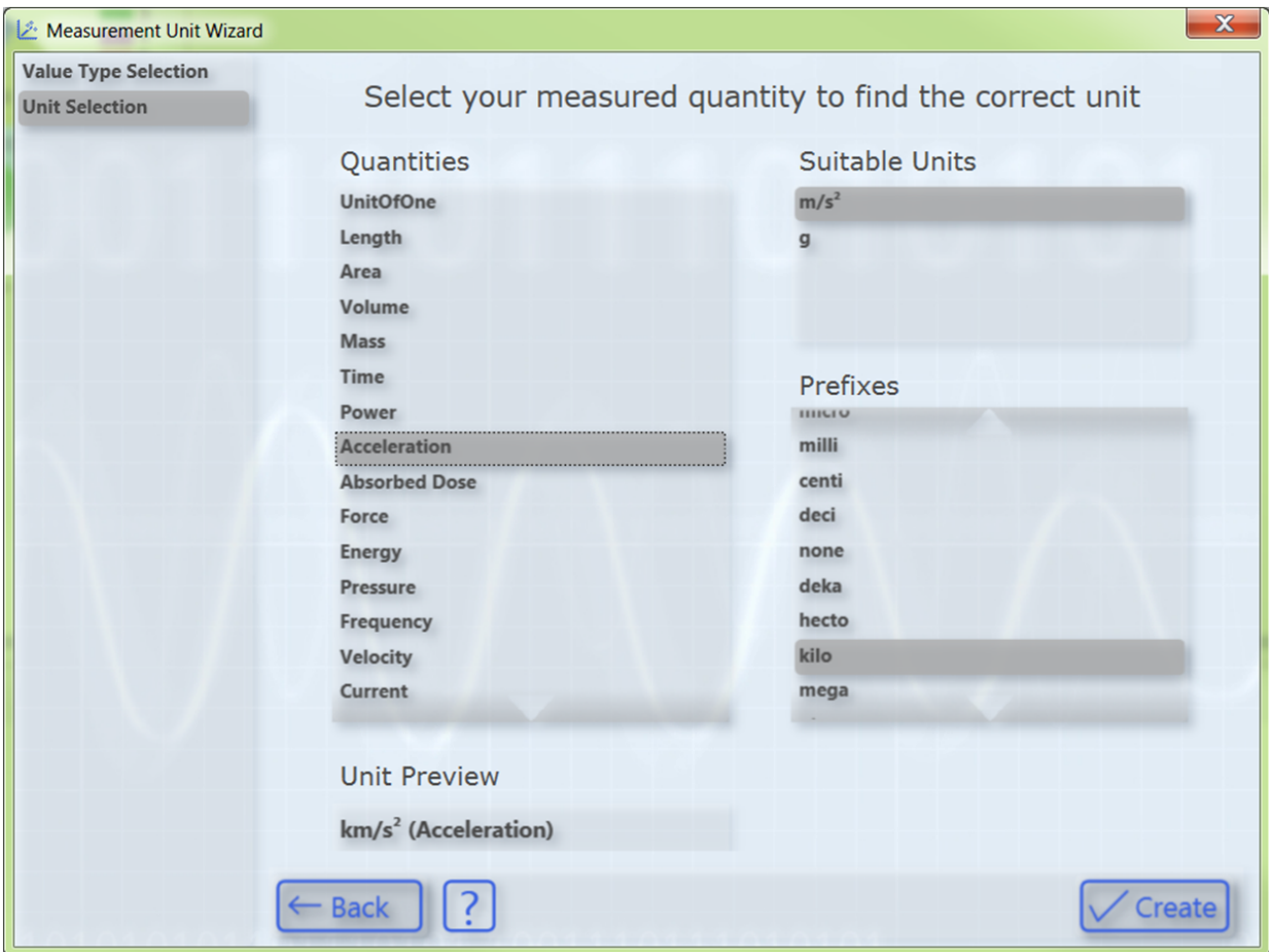
In order to select a unit, first select your measured variable. Subsequently, all units that come into question for the selected measured variable are displayed on the right-hand side. After you have selected the desired unit you can optionally select a prefix. In the "Unit Preview" field at the bottom left you can see how the currently selected unit will be displayed later in the **Properties** window. Finally, you can set the unit with the **Create** button. Since the values have already been scaled appropriately, no conversion of the values takes place in this case. The selected unit is merely entered and displayed.

Unscaled Value

If you have selected the option "Unscaled Value", the following page opens:
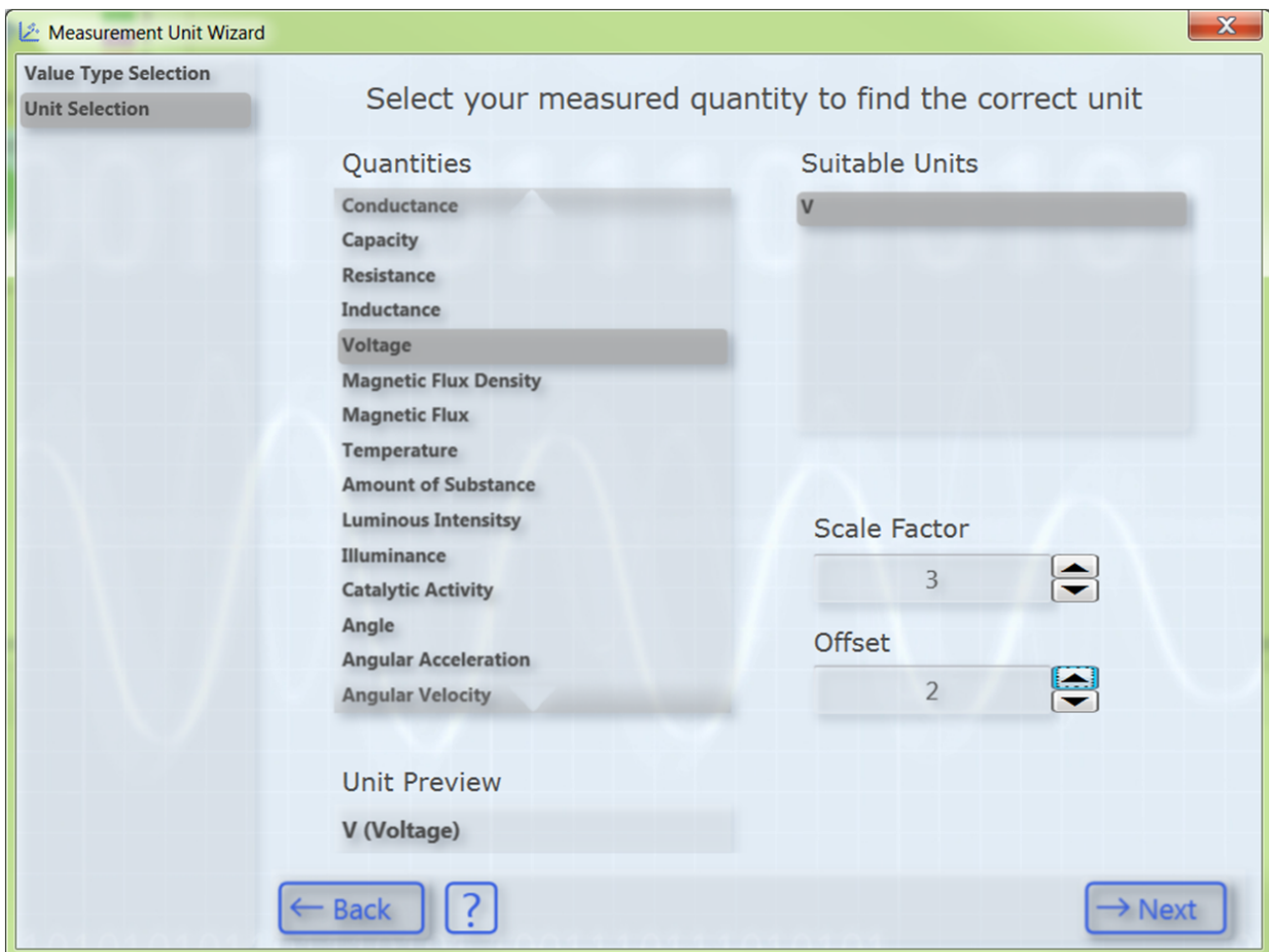
BECKHOFF

In order to select a unit, first select your measured variable. Subsequently, all units that come into question for the selected measured variable are displayed on the right-hand side. After having selected the desired unit you can specify a scaling factor and an offset with which the measured values are to be converted into the appropriate unit.

Subsequently, the **Next** button takes you to the next page, on which you can optionally select a transformation (e.g. a sensor). However, this page only appears if the previously selected unit is a base unit and not a user unit. Otherwise you will be taken directly to the last page of the dialog (Prefix Selection).
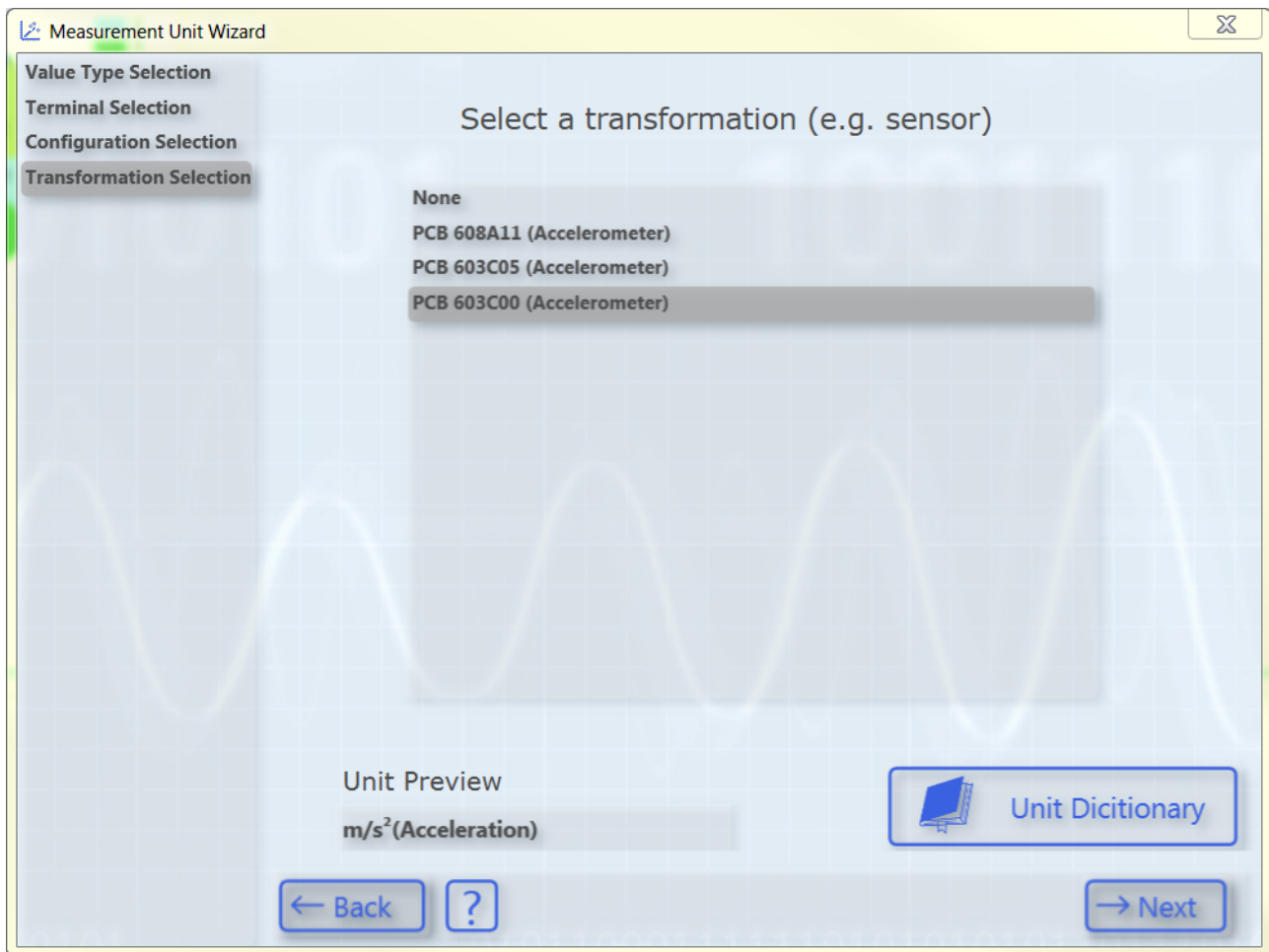
The list shows transformations from one SI unit to another SI unit. The list contains only transformations whose source unit corresponds to the previously selected base unit. The associated target unit is displayed in the "Unit Preview" field at the bottom left if a transformation is selected. If no transformation is required, simply select **None**.

Some transformations are predefined as examples in the TC3 Scope. However, these frequently depend on the specific application case (sensor, measuring setup, etc.), therefore it is not particularly useful to define a large number of generally valid transformations. You can therefore create your own transformations with the aid of the Unit Dictionary, which you can access via the button at the bottom right or via the Scope options [▶ 141]. Detailed information about the Unit Dictionary can be found in the section Unit Dictionary Editor [▶ 101]. Furthermore, there is an option to enter transformations via the PLC attributes (see Configuration of units with the aid of PLC attributes [▶ 100]).

After having selected the desired transformation, the **Next** button opens the next page, on which you can optionally select a user unit.

Once again, only user units that match the previously selected base unit appear. Some frequently used user units are predefined. As with the transformations, further user units can be added via the Unit Dictionary or the PLC attributes.

After selecting the user unit, you will be taken to the last page of this dialog:

Here you can optionally select a prefix for your unit. This is then also displayed in the "Unit Preview". Finally, you can create the configured unit with the aid of the **Create** button.

Terminal Value

If you have selected the option "Terminal Value", the following page opens:

Measurement Unit Wizard

Value Type Selection

Terminal Selection

Select a terminal type as your values source

EL3064
EL3068
EL3101
EL3102
EL3104
EL3161
EL3162
EL3164
EL3174
EL3174-0002
EL3174-0090
EL3174-0032
EL3632
EL3751
EL3751-0020

Unit Dictionary

← Back   ?   → Next
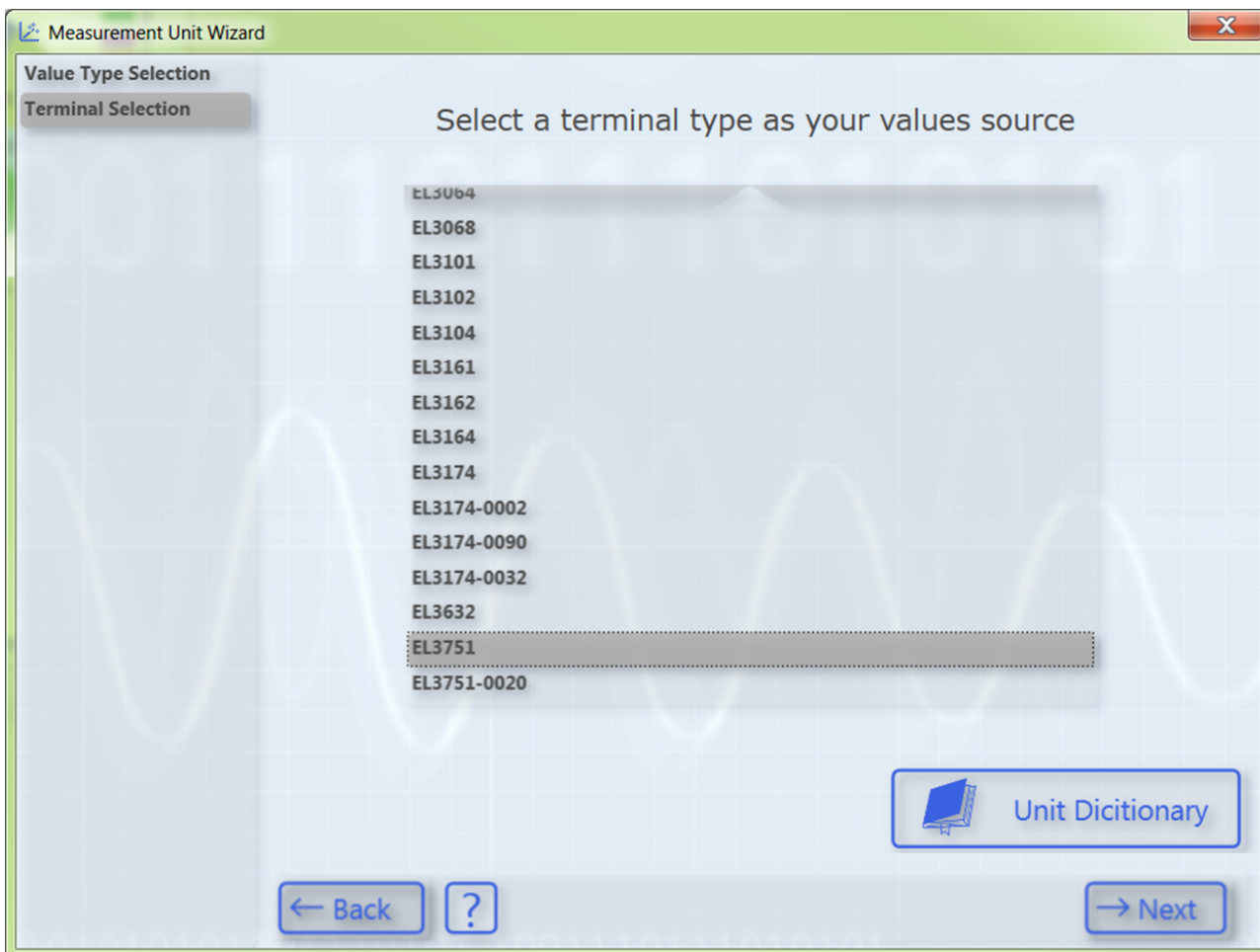
On this page you can select the type of terminal with which you recorded your measured values. Several Beckhoff EtherCAT terminals are already predefined. You can use the Unit Dictionary to view them or to add further terminals (see also Unit Dictionary Editor [▶ 101]).

Since a Beckhoff EtherCAT terminal can be configured in different ways, you must select the set configuration of your terminal on the next page.
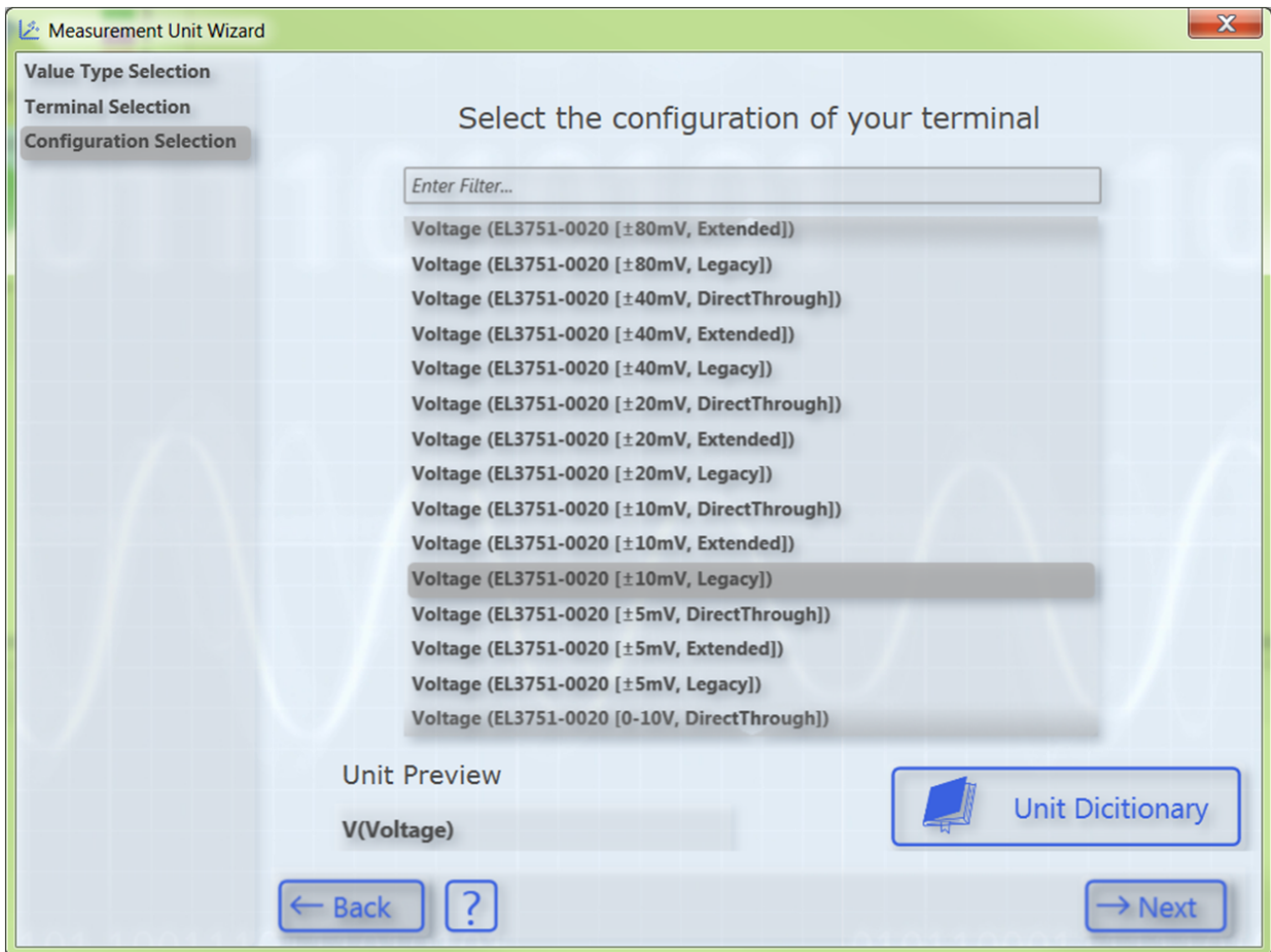
All configuration options of the previously selected terminal type appear in the list. You can filter the configuration options via the text search in order to quickly find the appropriate configuration. The various configuration options are also stored in the Unit Dictionary.

Subsequently, the **Next** button takes you to the next page, on which you can optionally select a transformation (e.g. a sensor).
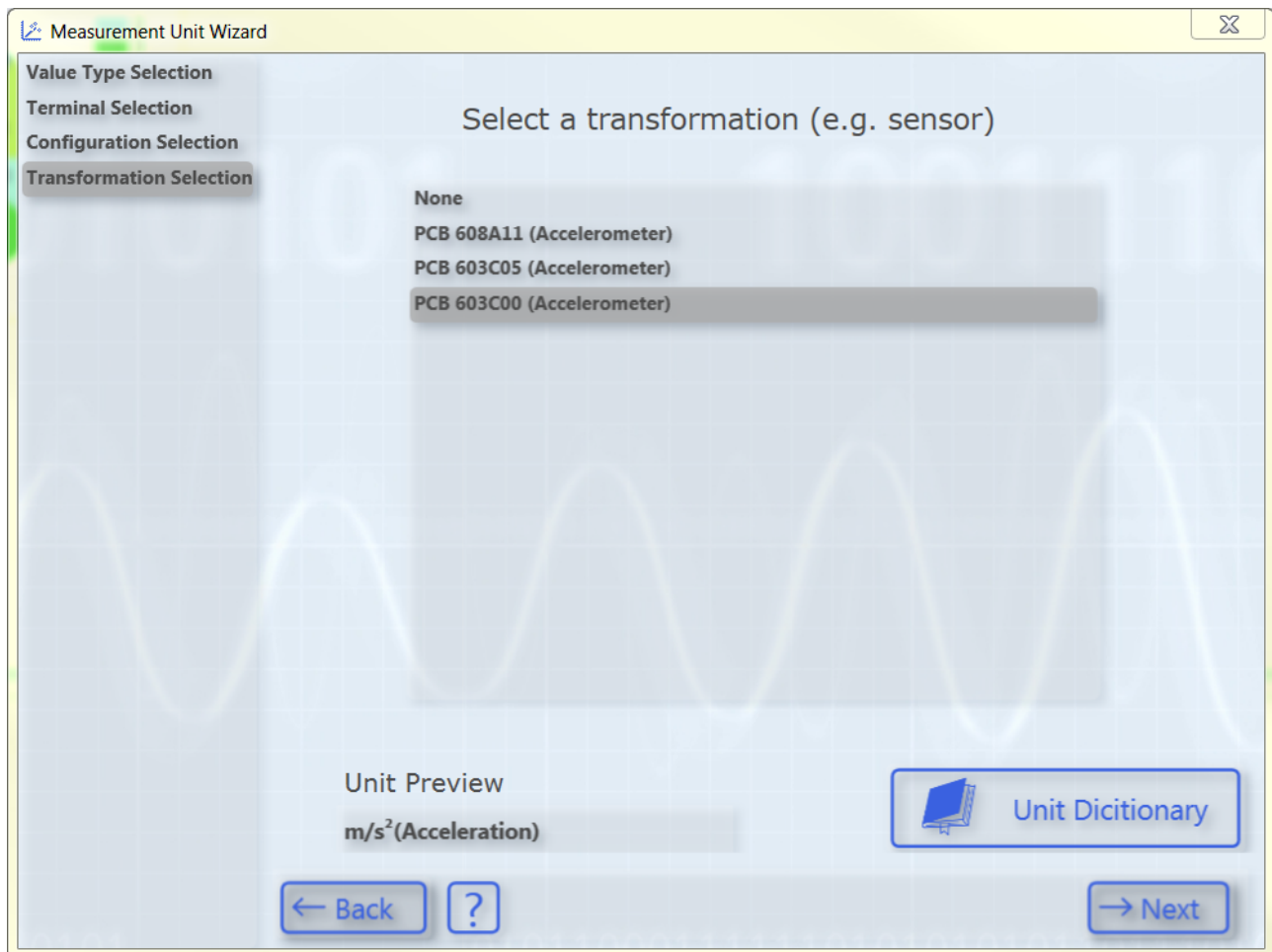
The list shows transformations from one SI unit to another SI unit. The list contains only transformations whose source unit corresponds to the previously selected base unit. You can see the associated target unit in the "Unit Preview" field at the bottom left when a transformation is selected. If no transformation is required, simply select **None**.

Some transformations are predefined as examples in the TC3 Scope. However, these frequently depend on the specific application case (sensor, measuring setup, etc.), therefore it is not useful to define a large number of generally valid transformations. You can therefore create your own transformations with the aid of the Unit Dictionary, which you can access via the button at the bottom right or via the Scope options [▶ 141]. Detailed information about the Unit Dictionary can be found in the section Unit Dictionary Editor [▶ 101]. Furthermore, there is an option to enter transformations via the PLC attributes (see Configuration of units with the aid of PLC attributes [▶ 100]).

After having selected the desired transformation, the **Next** button opens the next page, on which you can optionally select a user unit.

Once again, only user units that match the previously selected base unit appear. Some frequently used user units are predefined. As with the transformations, further user units can be added via the Unit Dictionary or the PLC attributes.

After selecting the user unit, you will be taken to the last page of this dialog:

Here you can optionally select a prefix for your unit. This is then also displayed in the "Unit Preview". Finally, you can create the configured unit with the aid of the **Create** button.

5.1.3.3 Configuration of units with the aid of PLC attributes

In addition to the input option via the Unit Wizard, units can also be configured with the aid of the attributes of a symbol or a data type in the PLC code. This is useful, for example, if a certain variable is recorded more frequently using the TC3 Scope. The unit then only needs to be provided once as an attribute and not reconfigured every time.

PLC attributes are specified directly via the declaration of a variable or the data type in curly brackets, as can be seen in the illustration below.

```
17        {attribute 'BaseUnit' := '<AngularVelocitiy, 0.5, 0, 3>'}
18        fSine                    : LREAL;
```

Three different unit elements can be specified with the aid of the PLC attributes: Base units (BaseUnit), transformations (UnitTransformation) and user units (UserUnit).

The syntax for their input is as follows:

Base units

{attribute 'BaseUnit' := '< BaseUnit, ScaleFactor, Offset, Prefix >'}

e.g.: {attribute 'BaseUnit' := '< AngularVelocity, 0.5, 0, kilo>'}

→ The unit is then krad/s (AngularVelocity) with a ScaleFactor of 0.5.

Transformations

{attribute 'UnitTransformation' := '<SourceUnit, TargetUnit, ScaleFactor, Name>'}

e.g.: {attribute 'UnitTransformation' := '<Current, Temperature, 10, Current-Temperature Converter>'}

→ The resulting unit is the temperature in °C.

User units

{attribute 'UserUnit' := '<BaseUnit, UserUnit, Symbol, ScaleFactor, Offset, Prefix >'}

e.g.: {attribute 'UserUnit' := '<Temperature, Fahrenheit, °F, 1.8, 32, 0>'}

→ The resulting unit is the temperature in °F.

For the base units and prefixes the Enum values can be entered as numbers in addition to the names. This enables any base units and prefixes to be selected even if they are not predefined.

The three different elements can be specified individually or in combination. Note, however, that not all combinations make sense. A transformation and a user unit, for example, should be specified with an appropriate base unit. If no appropriate base unit is specified it will automatically be set to the base unit of the transformation or user unit.

In the case of input errors in the attribute syntax, the default value, i.e. a "Unit of One" is set automatically with a scaling factor of "1", an offset of "0" and the prefix "none".

5.1.3.4   Unit Dictionary Editor

For the user the Unit Dictionary Editor represents the interface to the Unit Dictionary, in which all available unit elements are stored. The predefined elements can be viewed and new values added via the Unit Dictionary Editor.

The Unit Dictionary Editor is available only with a Scope View Professional License.

On the one hand the editor can be called from the Unit Wizard via the **Unit Dictionary** button, which always appears when terminals, transformations or user units can be selected. On the other hand, the editor can be called via the Scope Options [▶ 141].

The Unit Dictionary Editor contains four different tabs containing the various elements:

- Units [▶ 101]
- Sources [▶ 102]
- Transformations [▶ 103]
- User Units [▶ 103]

Units

The **Units** tab contains a list of all base units. These are predefined and cannot be changed using the Unit Dictionary Editor. If additional base units are required, they must be entered by means of the Enum values as PLC attributes (see also Configuration of units with the aid of PLC attributes [▶ 100]).
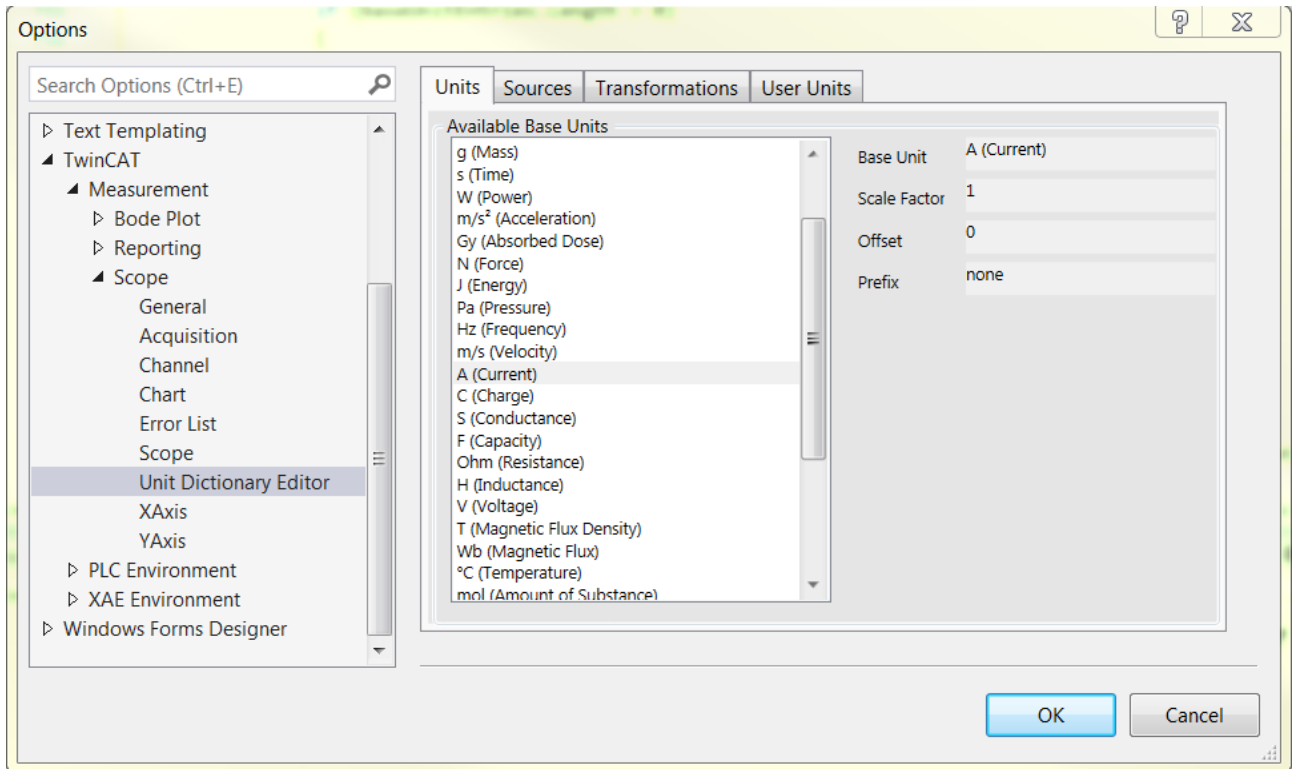
Sources

The **Sources** tab contains the Beckhoff EtherCAT terminals. However, as other sources are also conceivable for the measured values depending on the application case, the tab is generally called "Sources".
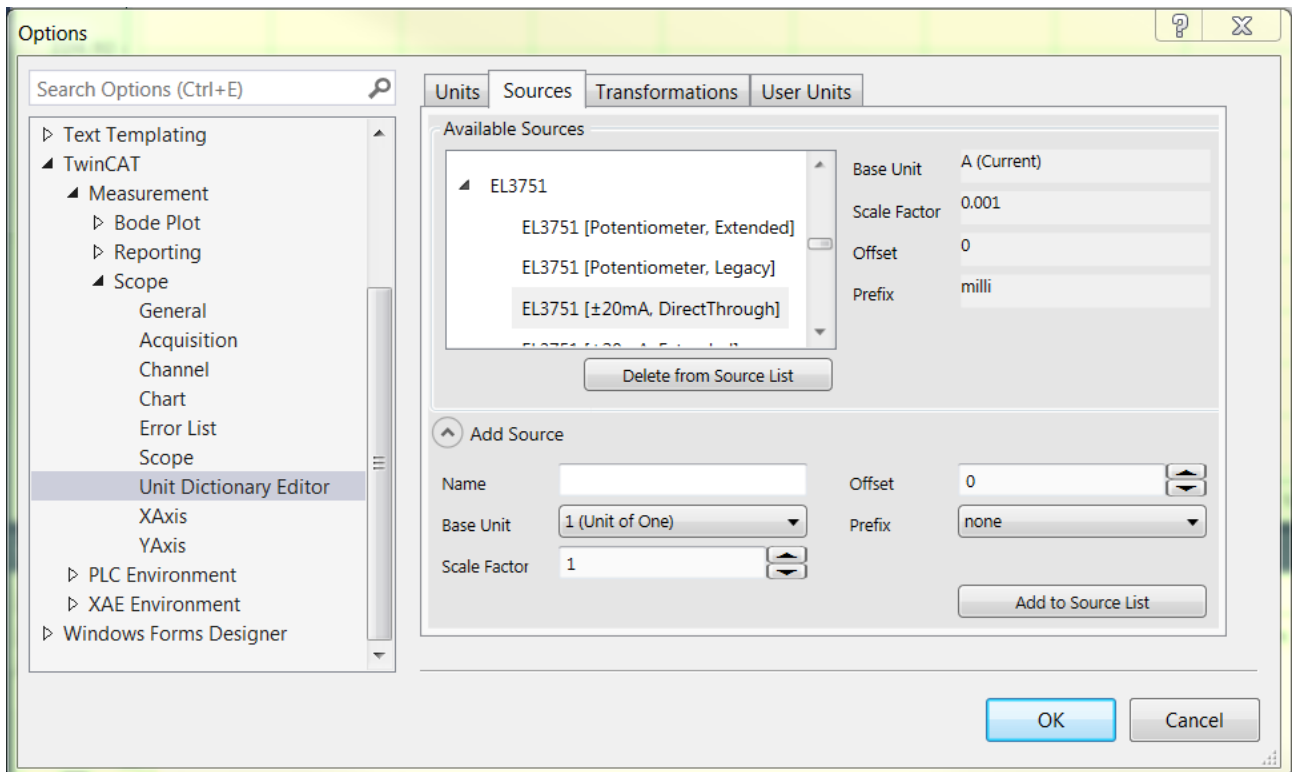
The terminal types with their configuration options are displayed in the list at the top left. To the right of that the properties of the respectively selected terminal can be viewed.

An input field can be opened in the lower area with the **Add Source** button. Further terminals or other user-specific sources can be added there. To do this it is merely necessary to fill out the fields. The element created can then be added using the **Add to Source List** button. A selected element can be deleted again with the **Delete from Source List** button. However, predefined values cannot be deleted from the list.

Note that when entering new "Source" elements, a terminal type and the associated configuration options should be entered so that they can be sorted in the Unit Dictionary Editor and in the Unit Wizard analogous to the existing elements. For example, the terminal type in the above screenshot is "EL3751". Its associated configuration options are added as individual elements. The name of the configuration begins with the designation of the terminal type and contains the respective configuration option behind that in square brackets.

Transformations

The **Transformations** tab contains all predefined transformations.
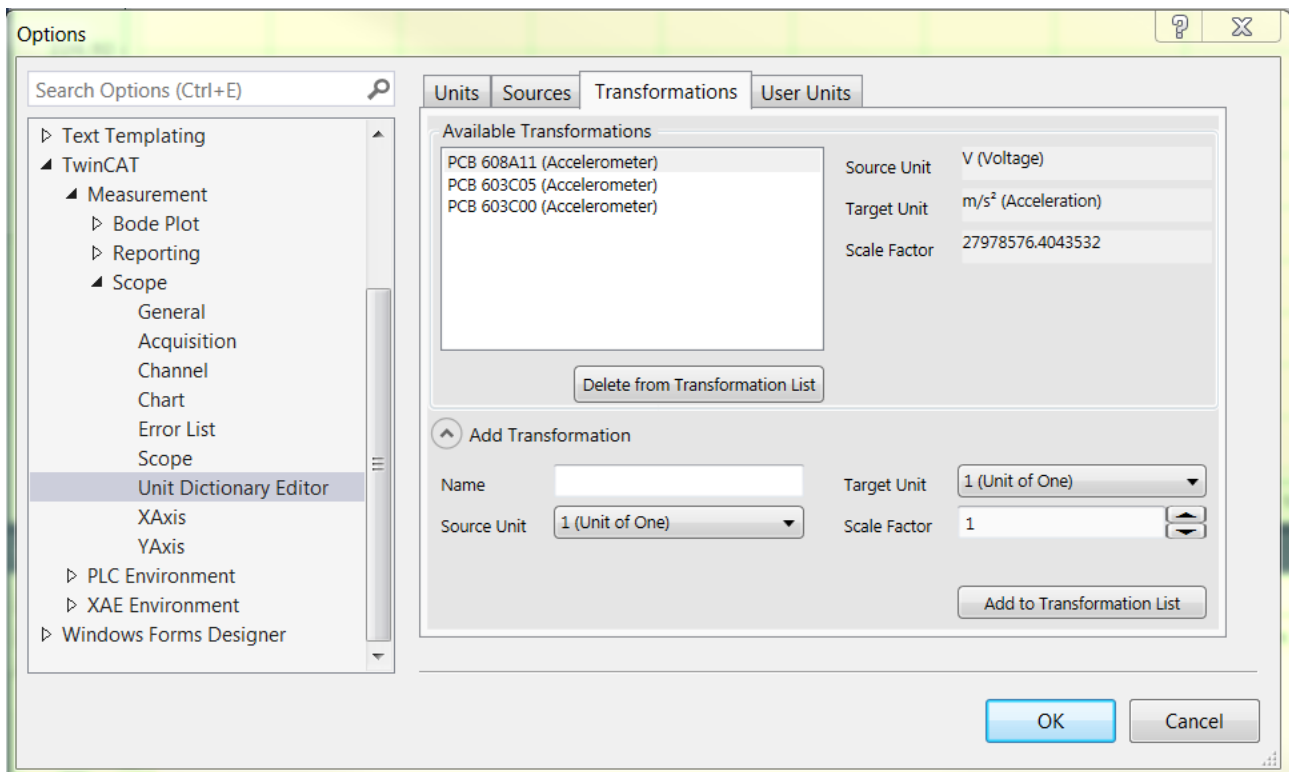
The properties of a selected transformation can be seen on the right. In order to define a new transformation, the input field must be opened via the **Add Transformations** button and the corresponding data entered. The transformation can then be added using the **Add to Transformation List** button. A selected transformation can be deleted again with the aid of the **Delete from Transformation List** button. Note that predefined transformations cannot be deleted.

User Units

The **User Units** tab contains all predefined user units.

BECKHOFF
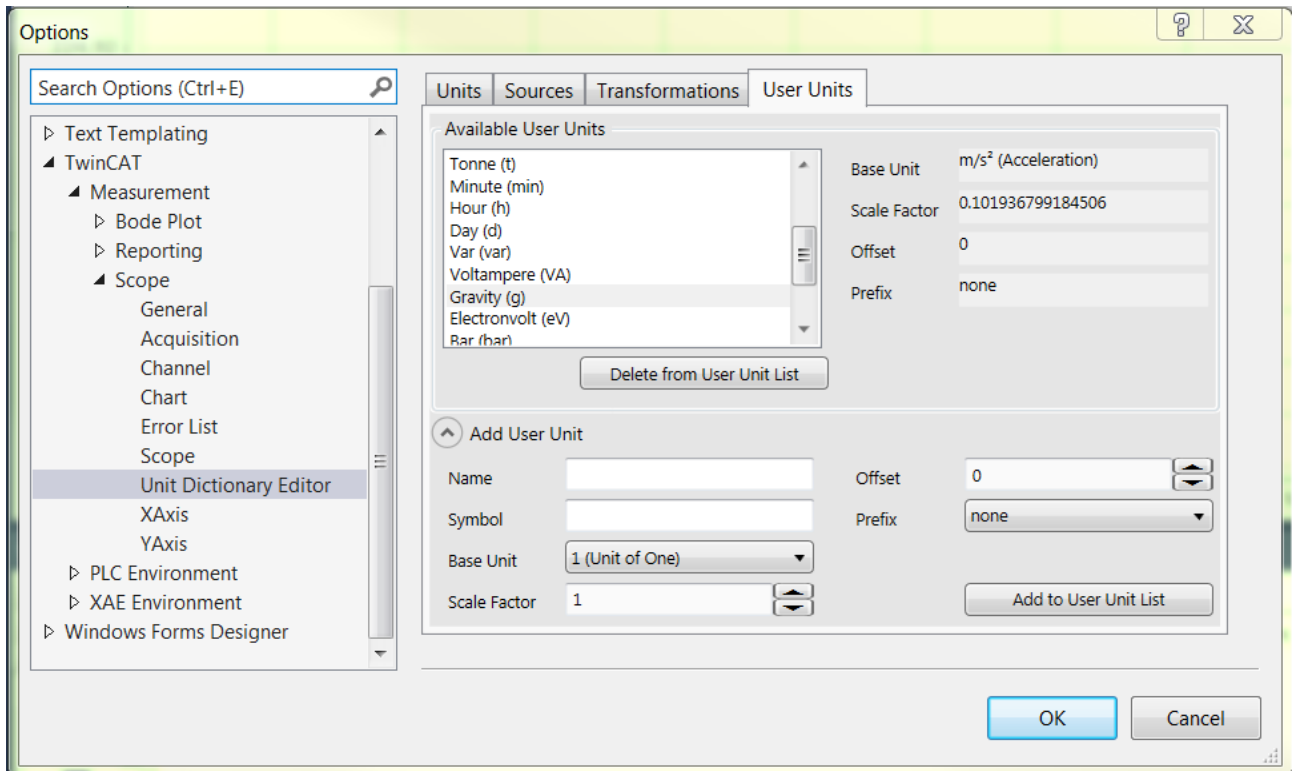
The properties of a selected user unit can be seen on the right-hand side. In order to define a new user unit, the input field must be opened via the **Add User Unit** button and the corresponding data entered. The user unit can then be added using the **Add to User Unit List** button. A selected user unit can be deleted again with the aid of the **Delete from User Unit List** button. Note that predefined user units cannot be deleted.

5.1.4   TwinCAT Target Browser

The TwinCAT Target Browser is the central data management interface in the TwinCAT 3 Engineering. It can provide data from various TwinCAT target systems live via ADS or access historic data in databases. Depending on what TwinCAT 3 Functions are installed on an engineering system, the ADS standard extension of the TwinCAT Target Browser is extended by further extensions (see Extensions [▶ 105]).
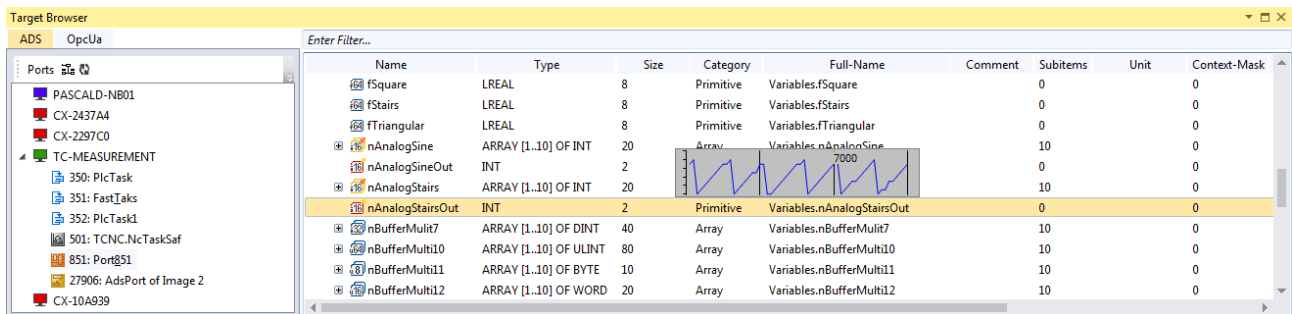
Within Microsoft Visual Studio® the TwinCAT Target Browser is a tool window that is subdivided into two sections. Target systems are displayed on the left-hand side ("Specific Target Area"). You can switch between the various extensions via the tabs. The details of the respective target system or the selected object respectively are displayed on the right-hand side ("Common Symbol Area").

Many extensions support a "Value Preview" for variables. This means that if you select a variable and press and hold the space bar, a small chart appears. In this way you can determine whether data have arrived or exist in a database. A search bar above the right-hand section enables the filtering/reduction of the visible symbols. The entry must be confirmed with the [Enter] key.

Extensions

The following table shows an overview of the current extensions and the product from which they were installed. Further information on the extensions can be found in the associated document sections.

Extension name	Products
ADS [▶ 105]	TwinCAT XAE, TwinCAT Scope, TwinCAT IoT Data Agent, TwinCAT Data base Server
OPC UA [▶ 107]	TwinCAT Scope, TwinCAT IoT Data Agent
TcAnalytics	TwinCAT Analytics Workbench and Service Tool
TcAnalytics File	TwinCAT Analytics Workbench and Service Tool

5.1.4.1 Extension – ADS

The TwinCAT Target Browser ADS extension is used most frequently within the TwinCAT system.
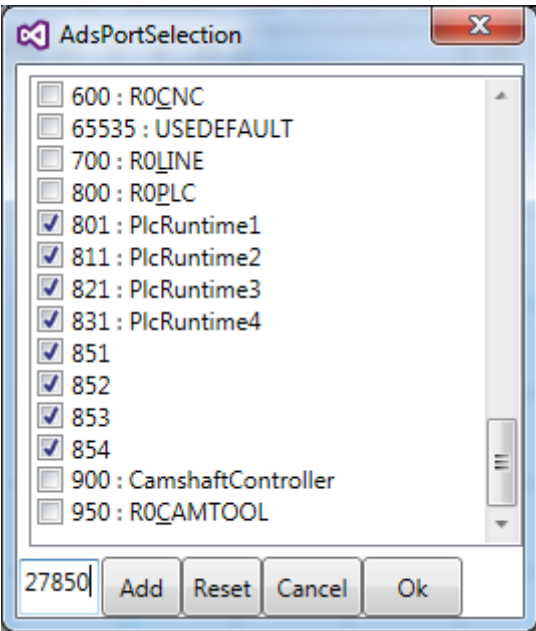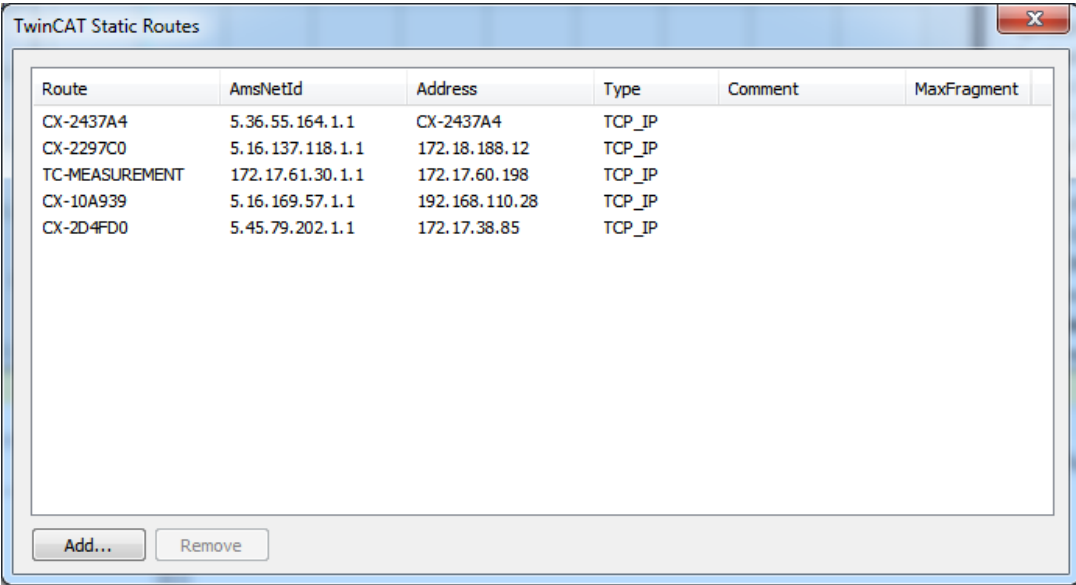
Specific Target Area

All target systems registered to the local TwinCAT 3 Engineering are displayed in a tree structure in the left-hand area of the TwinCAT Target Browser (ADS). In first place is the local system, followed by the target systems such as Industrial PCs or Embedded PCs in the order of registration. The prefixed screen symbol indicates the state of the system (green: run mode, blue: config mode, red: stop mode or unreachable). The available default ADS ports are listed below a target system. If you select a port, the available symbols/variables are displayed in the Common Symbol Area on the right-hand side of the TwinCAT Target Browser.
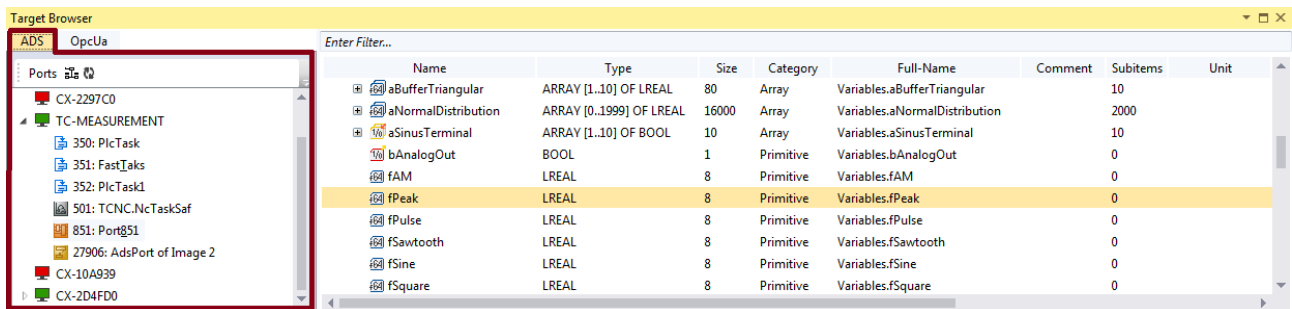
Toolbar

The toolbar of the ADS extension makes the following functions available:

Ports	If an ADS port is not displayed by default, further ports can be added to a selection via this command.
Ports	

Edit Routes	If an ADS route to a target system is missing, further target systems can be added via this button.

Refresh	The display of the target system states can be manually updated with this button.

Common Symbol Area

The ADS symbols available at the selected port are displayed in the right-hand area of the TwinCAT Target Browser. The addresses and the attributes, for example, are also displayed in addition to the name, the data type, the size and the symbol name. Special attributes, such as those of the units, are interpreted and output in their own columns.
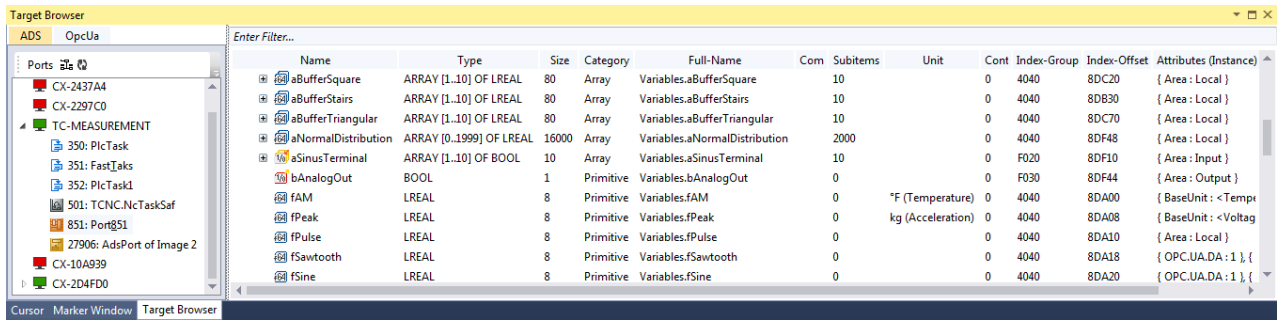
5.1.4.2   Extension - OPC UA

The TwinCAT Target Browser OPC UA extension offers a standardized way into the TwinCAT 3 Engineering.

Specific Target Area

All OPC UA Servers that have been added using the **Add** command in the toolbar are displayed in a tree structure in the left-hand area of the TwinCAT Target Browser (OpcUa). The screen symbol in front of the server designation at the first level of the tree structure indicates the connection status. Below the server the created end points are subdivided into "Anonym" (anonymous) and "Authenticated" (user mode). The encryption method is displayed in brackets for each end point. If you select an end point, the available OPC UA Nodes are displayed on the right-hand side of the TwinCAT Target Browser (see also: Displaying OPC UA Nodes [▶ 108]).
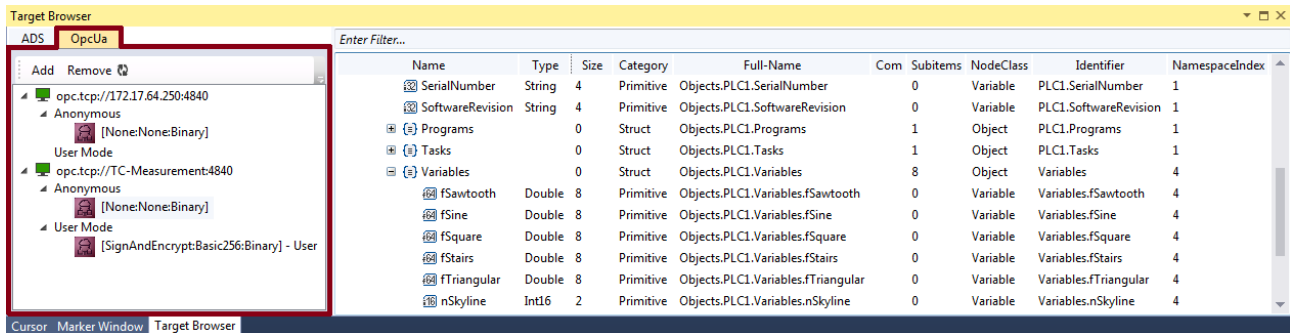
Toolbar

The toolbar of the OPC UA extension makes the following functions available:

Add	New connections to existing OPC UA Servers can be established with this command (see also: Adding OPC UA Servers [▶ 108]).
Remove	A registered server can be removed with this command.
Refresh	The display in the Target Browser tree can be manually updated with this button.

Common Symbol Area

The available OPC UA Nodes are displayed on the right-hand side of the TwinCAT Target Browser. These reflect the hierarchical structure of the PLC project. The node class and the identifier, for example, are also displayed in addition to the name, the data type, the size and the full object name.

Adding OPC UA Servers

1. Click on **Add** in the OPC UA toolbar.
   ⇨ The dialog **OpcUa EndPoint Selection** opens.
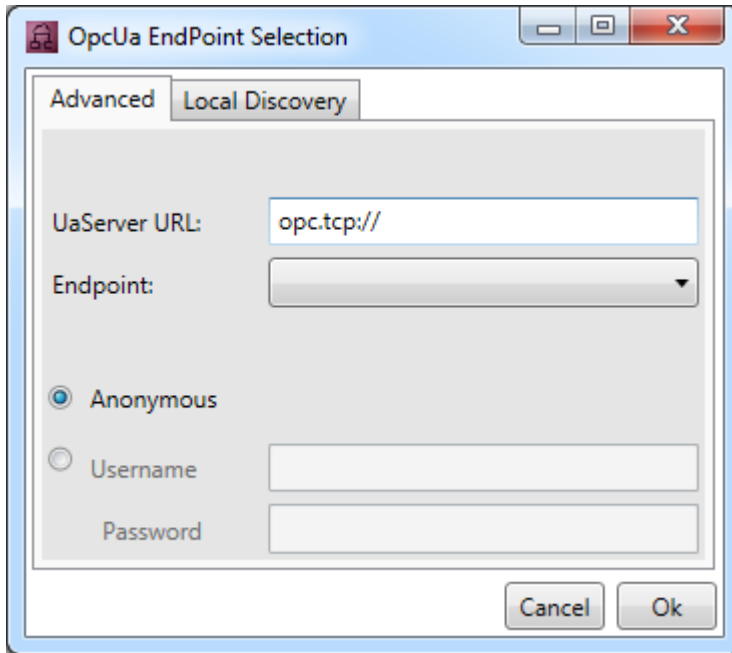
2. Enter the URL of the server.

3. Select the end point from the drop-down list. With OPC UA you can determine via the corresponding end points whether and which method of encryption is used. It is also possible to add several end points to a server. To do this, execute the **Add** command again.
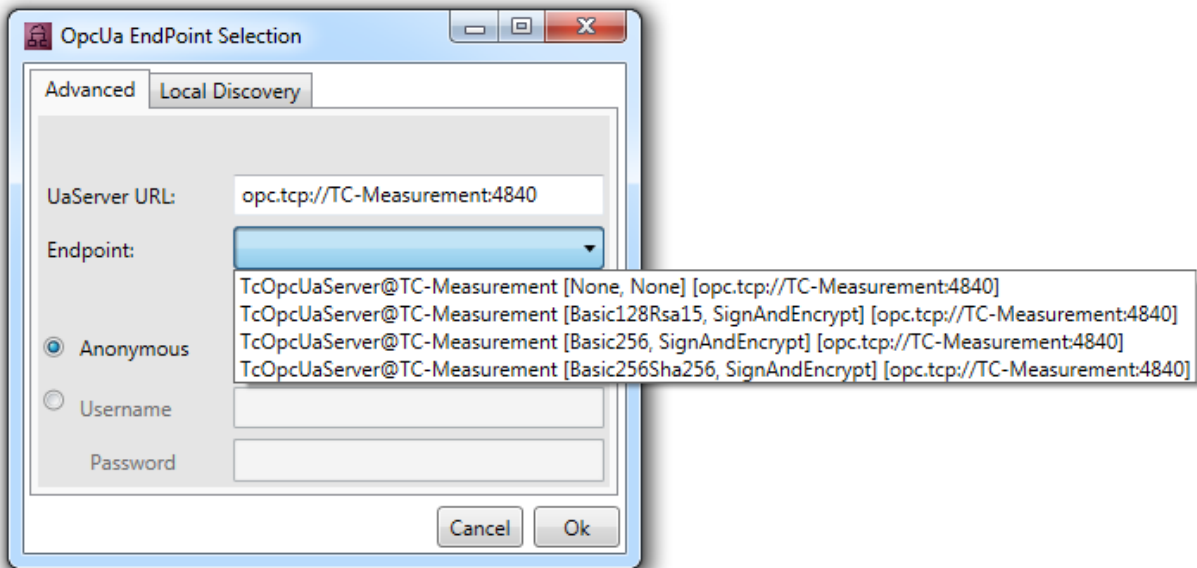
4. Select whether the access is anonymous or authenticated. If the access is authenticated, enter a user name and password. Authenticated access may be required if a password-protected user management has been set up for the OPC UA server (e.g. different user accounts with different rights).

5. Confirm the dialog.

⇨ The OPC UA Server is added to the tree structure in the Target Browser with the selected end points.

Displaying OPC UA Nodes

In order to display the available OPC UA Nodes, select the respective end point in the tree structure on the left-hand side. If you select an end point without certified access, the nodes will be displayed directly. If the selected end point is certified, you must first trust the server certificate in a corresponding dialog.
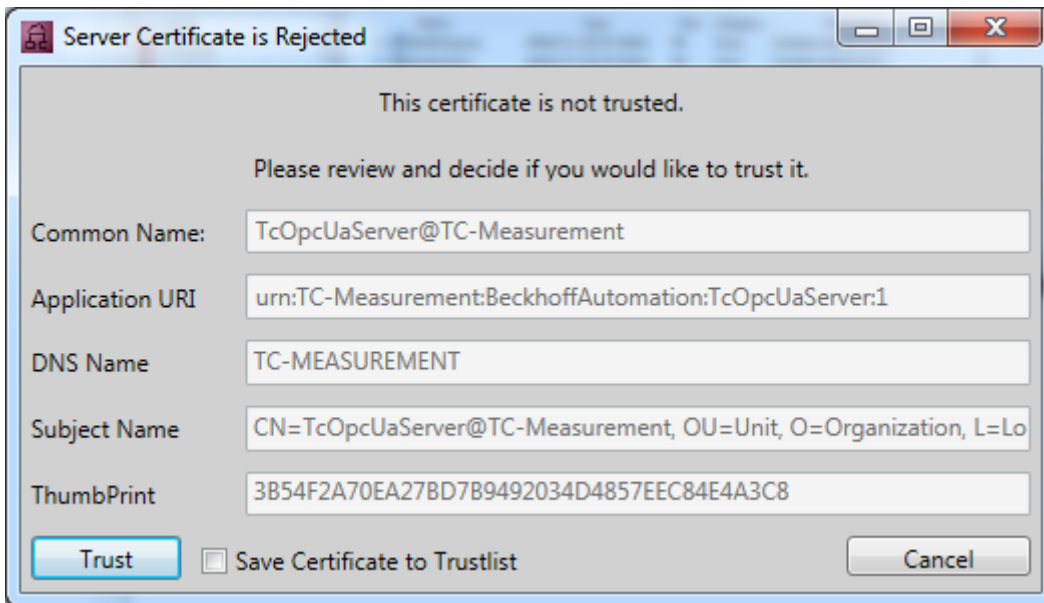
You can trust the certificate in a single case (until the Visual Studio instance is closed) or add it to the list of trusted certificates via the check box **Save Certificate to Trustlist**.

During the first attempt to connect to an OPC UA Server it is additionally necessary to trust the certificate of the client (Target Browser) on the server side. To do this, copy the respective certificate in the certificate directory of the OPC UA Server from the "rejected" folder to the "trusted" folder.

5.1.5   Report

5.1.5.1   Integrating and editing reports

With the help of the Microsoft Visual Studio Report Designer it is possible to generate complex printing templates which are used by the TwinCAT Measurement, for example, to print Scope charts.

Example:

The following report is generated from the recording shown here via the **Print** command in the **File** menu of Visual Studio (**[Ctrl] + [P]**): https://infosys.beckhoff.com/content/1033/TE13xx_TC3_ScopeView/Resources/pdf/18014398691815691.pdf.
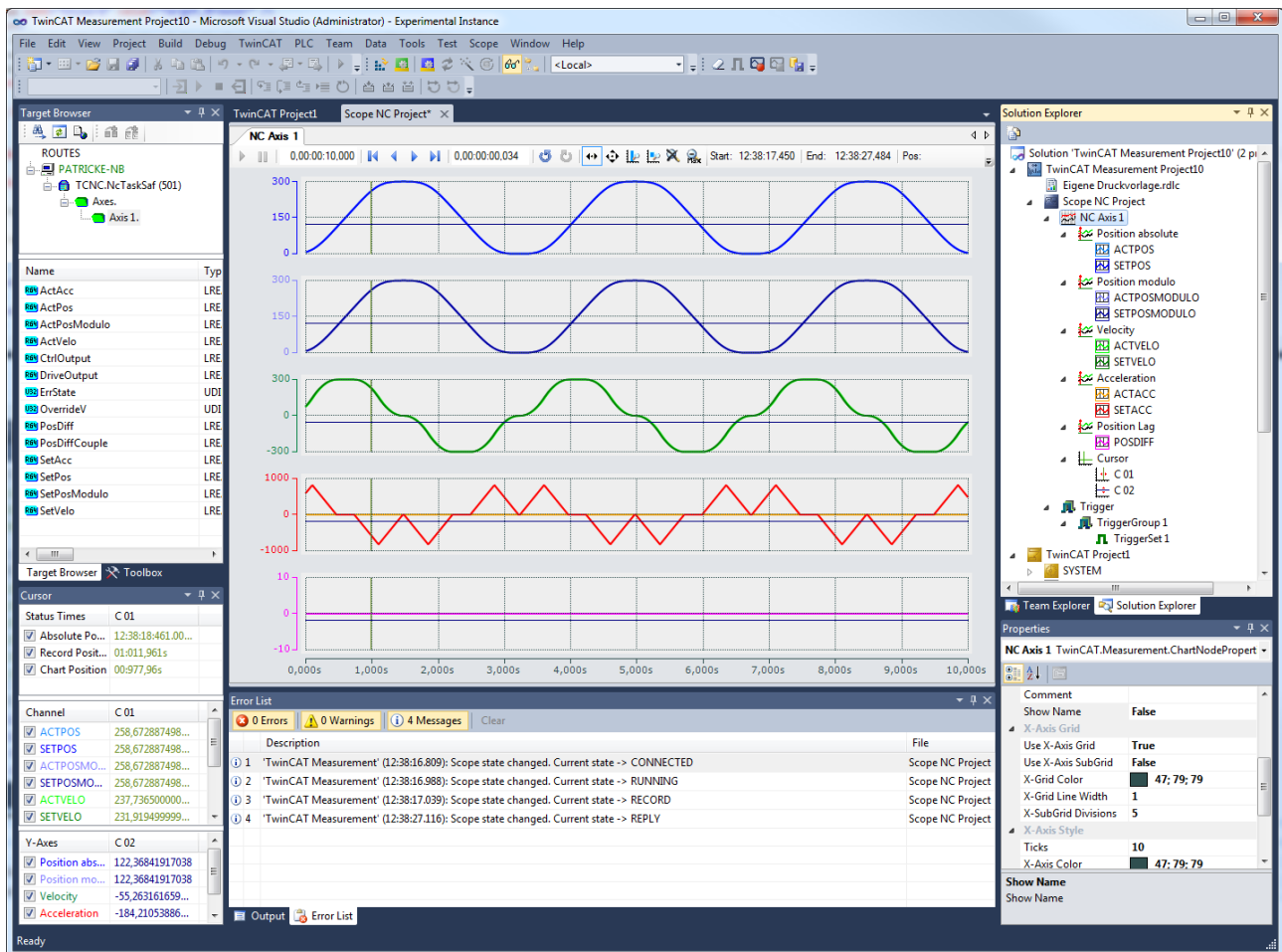
Integration of a report in a TwinCAT Measurement Project

Beckhoff already supplies a ready-made printing template with each installation of TwinCAT 3.1 in the TwinCAT Measurement. It can be added in any existing TwinCAT measurement project via the context menu command **Add > New Item > Standard Report**. This printing template is already contained in the project with the TwinCAT Measurement Template "Measurement Scope Project with Reporting".

Editing a report

To edit a report, open Microsoft Visual Studio Report Designer by double-clicking the print template.
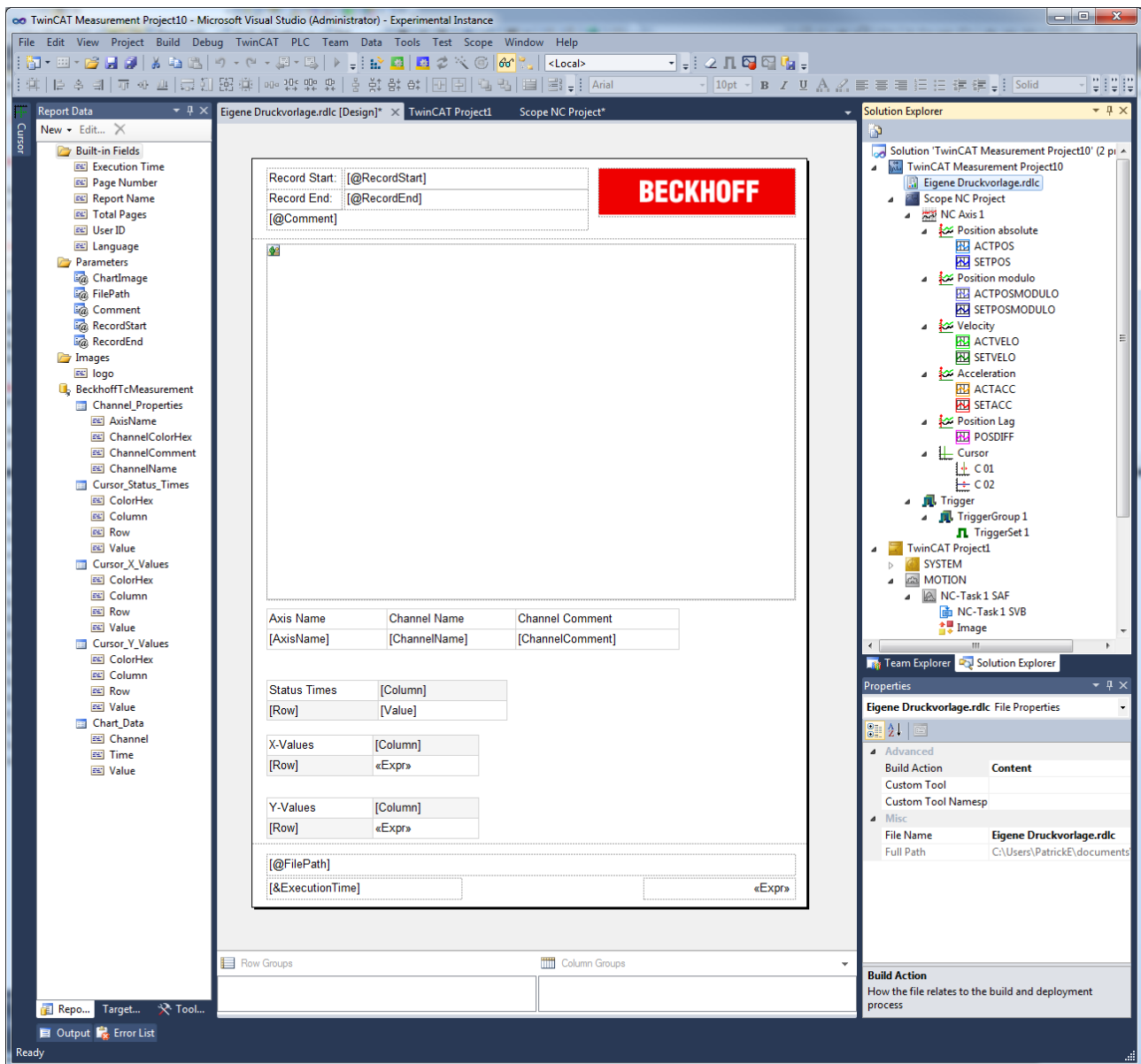
For the editing of report templates, please refer to the following article: Report Designer (Visual Studio).

Data of the data set

For the design of your own printing templates, Beckhoff offers a ready-made data set containing all relevant data of a Scope chart.

- **Built-in Fields**
  - **Execution Time:** Time at which a print job was created
  - **Page Number:** Current page number
  - **Report Name:** Name of the report
  - **Total Pages:** Total number of pages in the document
  - **User ID:** User ID of the user who commissioned the report
  - **Language:** Set system language
- **Parameters**
  - **ChartImage:** Converted image of the chart. Can be converted back to an image using the expression (System.Convert.FromBase64String (Parameters!ChartImage.Value).
  - **FilePath:** Storage location of the associated Scope file
  - **Comment:** The comment saved in the Scope element

- ◦ **RecordStart:** Start time of the recording
- ◦ **RecordStop:** Stop time of the recording
- **Images**
    - ◦ **logo:** Beckhoff logo used in the standard template
- **Channel_Properties**
    - ◦ **AxisName:** Name of the axis
    - ◦ **ChannelColorHex:** Color of the channel in hexadecimal notation
    - ◦ **ChannelComment:** Comment on the channel
    - ◦ **ChannelName:** Name of the channel
- **Cursor_Status_Times**
    - ◦ **ColorHex:** Color of the cursor in hexadecimal notation
    - ◦ **Column:** Name of the cursor column
    - ◦ **Row:** Name of the cursor row
    - ◦ **Value:** Displayed value
- **Cursor_X_Values**
    - ◦ **ColorHex:** Color of the cursor in hexadecimal notation
    - ◦ **Column:** Name of the cursor column
    - ◦ **Row:** Name of the cursor row
    - ◦ **Value:** Displayed value
- **Cursor_Y_Values**
    - ◦ **ColorHex:** Color of the cursor in hexadecimal notation
    - ◦ **Column:** Name of the cursor column
    - ◦ **Row:** Name of the cursor row
    - ◦ **Value:** Displayed value
- **Chart_Data**
    - ◦ **Channel:** Name of the channel
    - ◦ **Time:** Time stamp of the value
    - ◦ **Value:** Displayed value

● Use of Chart_Data

i The Chart_Data are the raw data of the chart. The raw data of the chart are for the creation of your own displays. They are only exported if the setting **Export Raw Data to Report** has been selected (Options dialog [▶ 141]). This setting significantly delays the creation of reports.

5.2   Loading, saving and exporting

5.2.1   Configurations

5.2.1.1   Creating and editing configurations

The following section describes how you can create and edit a Scope configuration.

5.2.1.1.1 Creating and editing a Scope project

Creating a measurement project

In the menu **File > New** select the command **Project**. In the dialog that opens select the category **TwinCAT Measurement** and the required template.
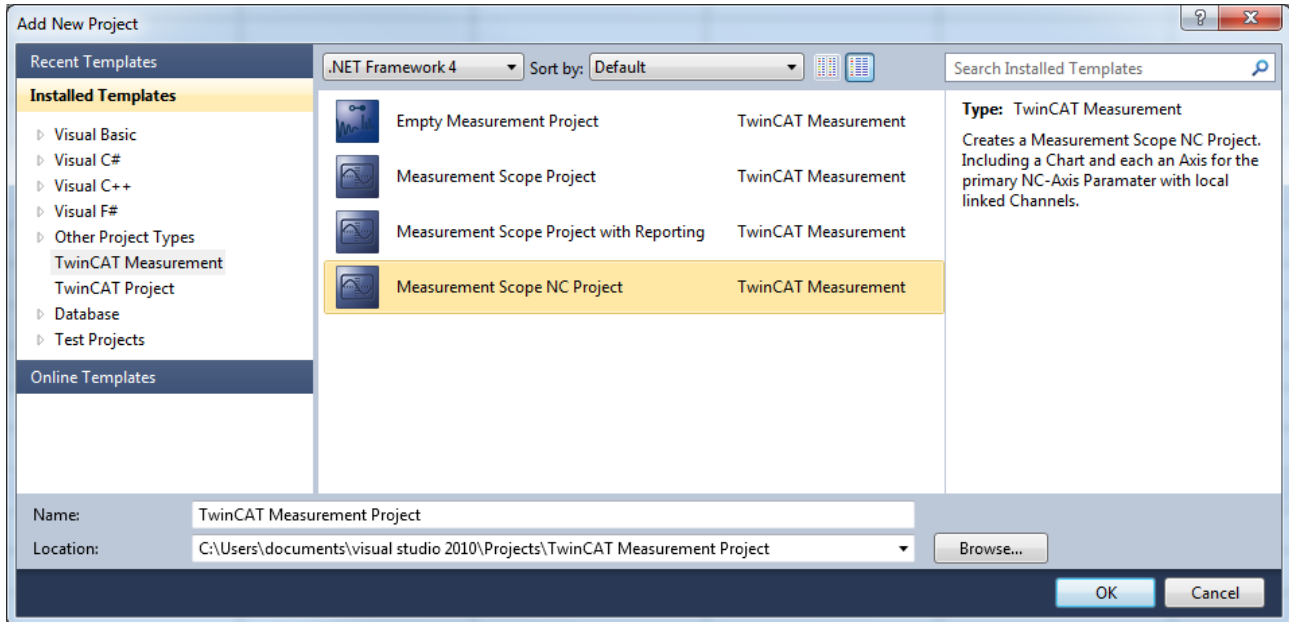
Available templates:

Empty Measurement project	Empty Measurement Project. Scope configurations (.sv2 | .tcscope | .tcscopex) or scope data (.svd | .svdx) can subsequently be inserted here.
Scope YT Project	Contains a Scope instance and a preconfigured YT chart with a corresponding axis.
Scope YT Project with Reporting	See "Scope YT Project" + a ready-made printing template for printing charts.
Scope YT NC Project	Contains a Scope instance, which was specially preconfigured for working with drive axes. The template can easily be extended to the number of actually configured drive axes.
Scope XY Project	Contains a Scope instance and a preconfigured XY chart with a corresponding axis.
Scope XY Project with Reporting	See "Scope XY Project" + a ready-made printing template for printing charts.
Scope Array Bar Project	Contains a Scope instance and a preconfigured array bar chart with a corresponding axis.

Add Scope report/project

In the context menu of the Measurement project select the command **Add > New Item...**, then select the required template in the dialog that opens.
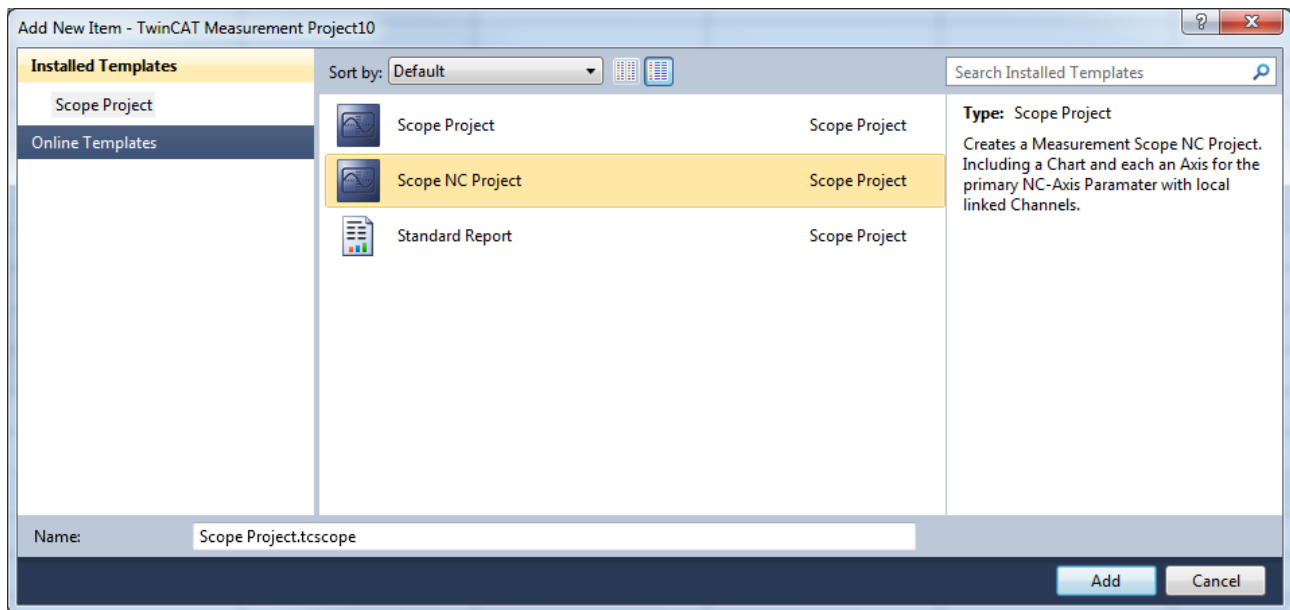
Available templates:

Scope Project	Scope instance with a pre-configured chart and an axis.
Scope NC Project	Scope instance that has been specially pre-configured for working with axes.
Standard Report	Printing template for printing charts.

Adding an existing Scope project to a measurement project

In the context menu of the measurement project, select the command **Add > Existing Item** and, in the dialog which then opens, select the desired file (.tcscopex | .tcscope | .sv2 | .svdx | .svd) or drag the desired file by drag & drop onto the project in the **Solution Explorer**.

Adding variables

Adding variable or channels

The prerequisite is a TwinCAT Measurement project with a Scope View. To add channels to the configuration, double-click in the **Target Browser** on the corresponding channels in the tree structure or mark channels and select the command **Add Symbols** (**Add Subsymbols**) in the context menu or the toolbar. The symbols are now inserted at the last-selected place in the Scope configuration. Alternatively, you can add selected elements anywhere in the configuration via drag & drop.

If you select a folder, an Ads Big type or an array containing subsymbols, the command **Add Subsymbols** adds a channel for each subsymbol. The command **Add Symbol** only adds one channel with the information for the selected symbol.
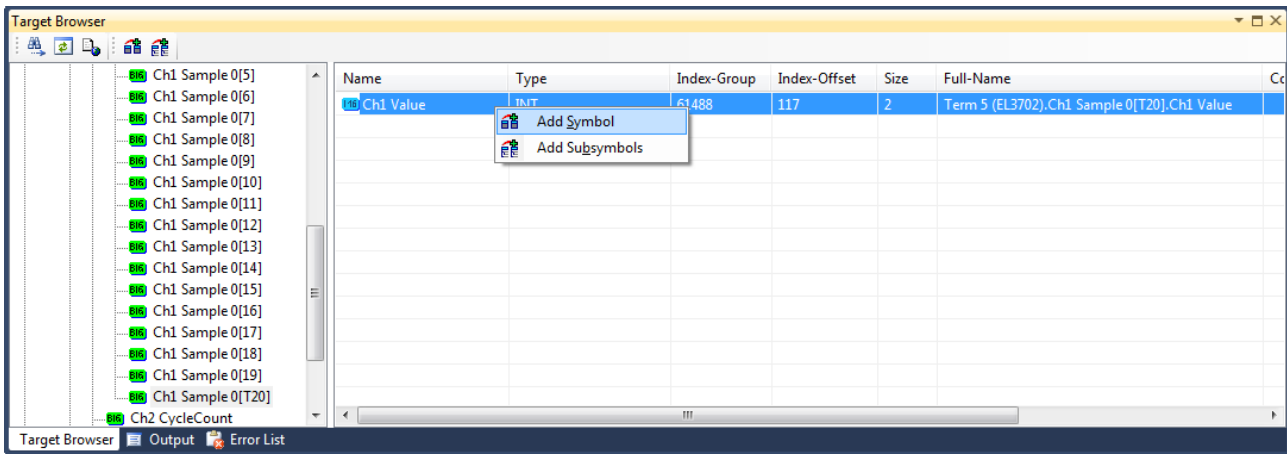
Disabling and enabling a channel

A channel can be disabled in the configuration in order to prevent it being recorded. However, it is retained when saving the configuration.

In the **Solution Explorer**, mark the channel to be deactivated and set the **Disabled** property in the **Properties** window to TRUE.

Task selection

If you select a target computer in the tree structure, the connected tasks become visible. These are marked in color:

- Green = PLC task
- Blue = NC task
- Red = other

If a task is not displayed, you can use **Enable Server Ports** to call up a selection menu that displays the restriction to particular tasks.
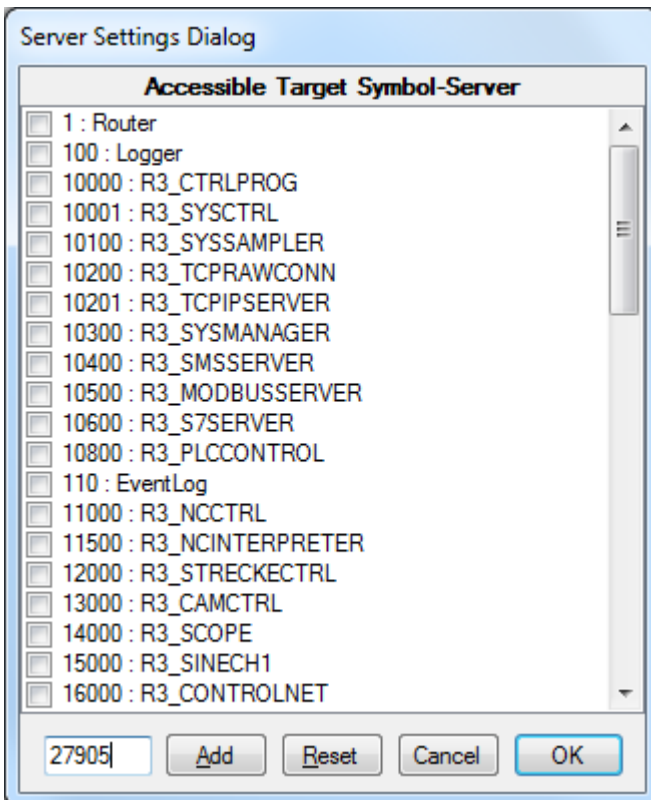
All popular Ads ports are displayed in the dialog, regardless of whether server tasks are concerned. If a port is not listed here it can be entered in the text field at the bottom left. All selected ports that are servers are adopted after confirming with **OK**. Subsequently a check is made as to which of the ports exist, in order to update the display.

Viewing the symbol list

If a symbol list is available for the selected task, you can call it by selecting. All symbols of a task are transferred into a hierarchical view according to separators (e.g. '.'), allowing simple browsing.

Settings

The option **Name Detail Level** in the settings can be used to set the detail level of the channel name formed from the symbol name. If the detail level is zero, only the part of the symbol name after the last dot is transferred. Each further level then adds the term before the respective next full stop. For example, with a detail level of "0" the symbol "Main.Signals.Sine" becomes "Sine", with a detail level of "1" it becomes "Signals.Sine", with a detail level of "2" or higher "Main.Signals.Sine".
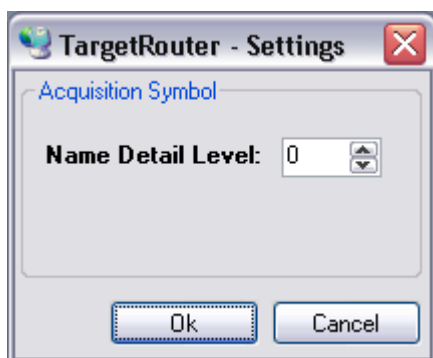
Starting and stopping a recording

Once you have selected all channels for the recording you can start the recording with the **Record** command in the TwinCAT Measurement toolbar ( ) and stop it again with the **Stop Record** command ( )

Adding and deleting elements

Adding a new chart

Select the **New YT Chart** or **New XY Chart** command in the context menu of the Scope instance.

Adding a new axis

Select the **New Axis** command in the context menu of the Scope / Chart instance.

Adding a new channel

- Select the **New Empty Channel** command in the context menu of the Scope / Chart / Axis instance.
- Drag & drop the symbol data from the **Target Browser** to an existing Scope / Chart / Axis instance.
- Select the **Add to Scope** command in the context menu of the variable in the PLC editor.

Deletion of elements

- In the **Solution Explorer**, mark the element to be deleted and press the **[Del]** button.
- Select the **Delete** command in the context menu of the element.

Shifting and copying elements

Moving and copying individual elements

Each element can be moved with drag & drop in the **Solution Explorer**. To copy an element, press the [Ctrl] button and shift the element by drag & drop.

Notes:

- If you move an element to another Scope, it is copied. Thus, it is still available in the original Scope.
- When moving an element, it must always be placed on an element of a higher hierarchical level.
- The described shift operations are available within a Scope even after the start of recording.

5.2.1.1.2 Creating and editing a Wizard project

The Measurement Wizard assists you in creating and configuring a user-defined Measurement Scope project by guiding you through the necessary steps with several successive dialogs.

Structure and mode of operation of the Measurement Wizard

In the main area of the wizard you will be instructed with the help of various design elements to select the settings necessary for the Measurement Scope project. An option or setting is highlighted in gray after selection. The wizard usually moves automatically to the next configuration step. Otherwise a button labeled **Next** appears at the bottom of the screen via which you can proceed to the next configuration step.

The steps that have already been completed are displayed in the left-hand area of the Wizard. The current configuration step is highlighted in gray. You can return to an already completed configuration step via the progress indicator or the **Back** button at the bottom edge of the wizard and edit the setting again.
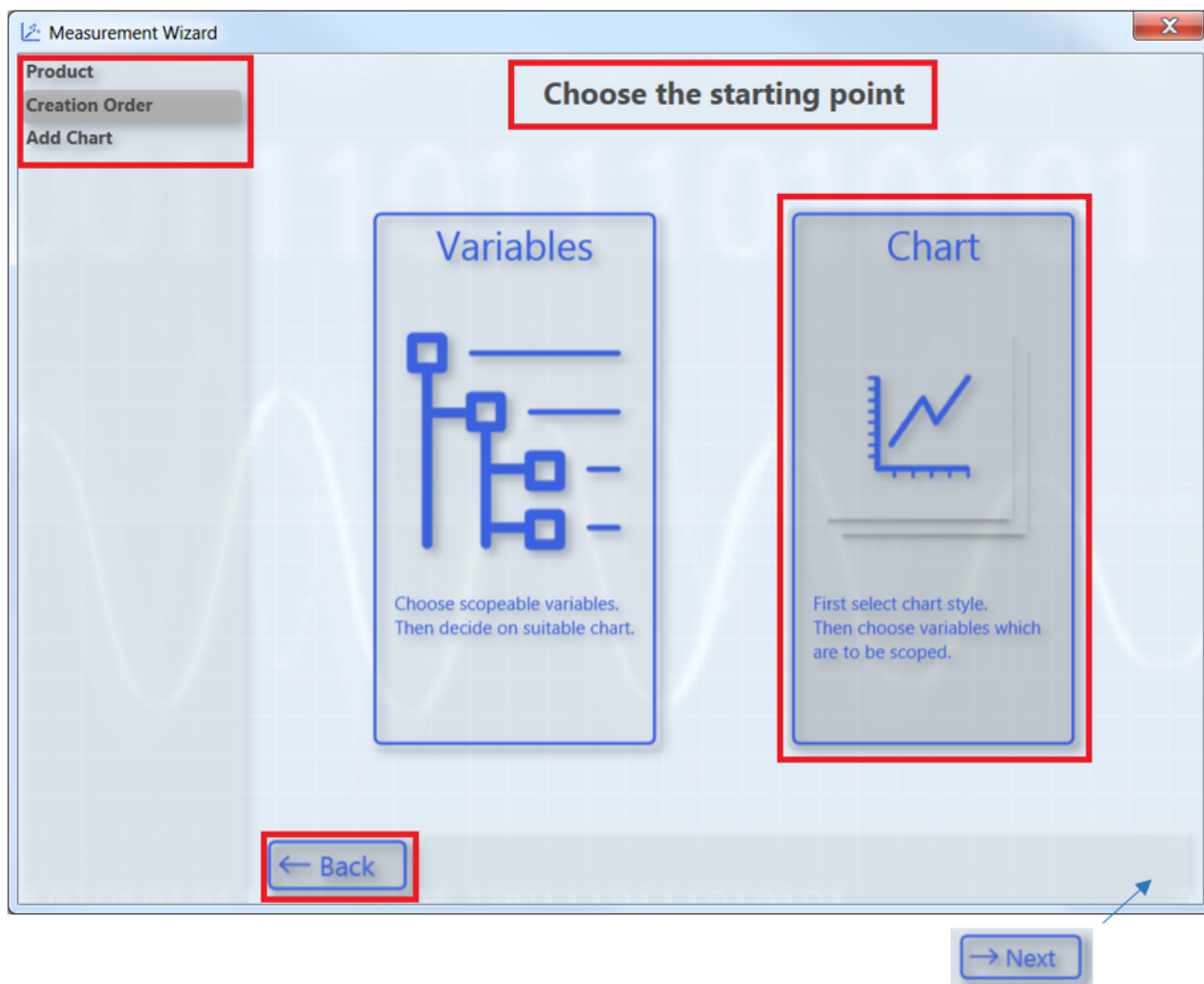
If you change a setting that had already been made, a dialog will inform you that you have changed the old configuration.
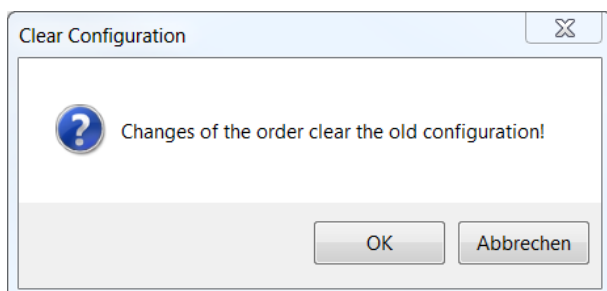
Creating and configuring a Measurement Wizard project

✓ TwinCAT is in RUN mode and the PLC project in online mode.

1. In the menu **File > New** select the command **Project**.

2. In the dialog that opens, select the TwinCAT Measurement category and the **Measurement Wizard Project** template.
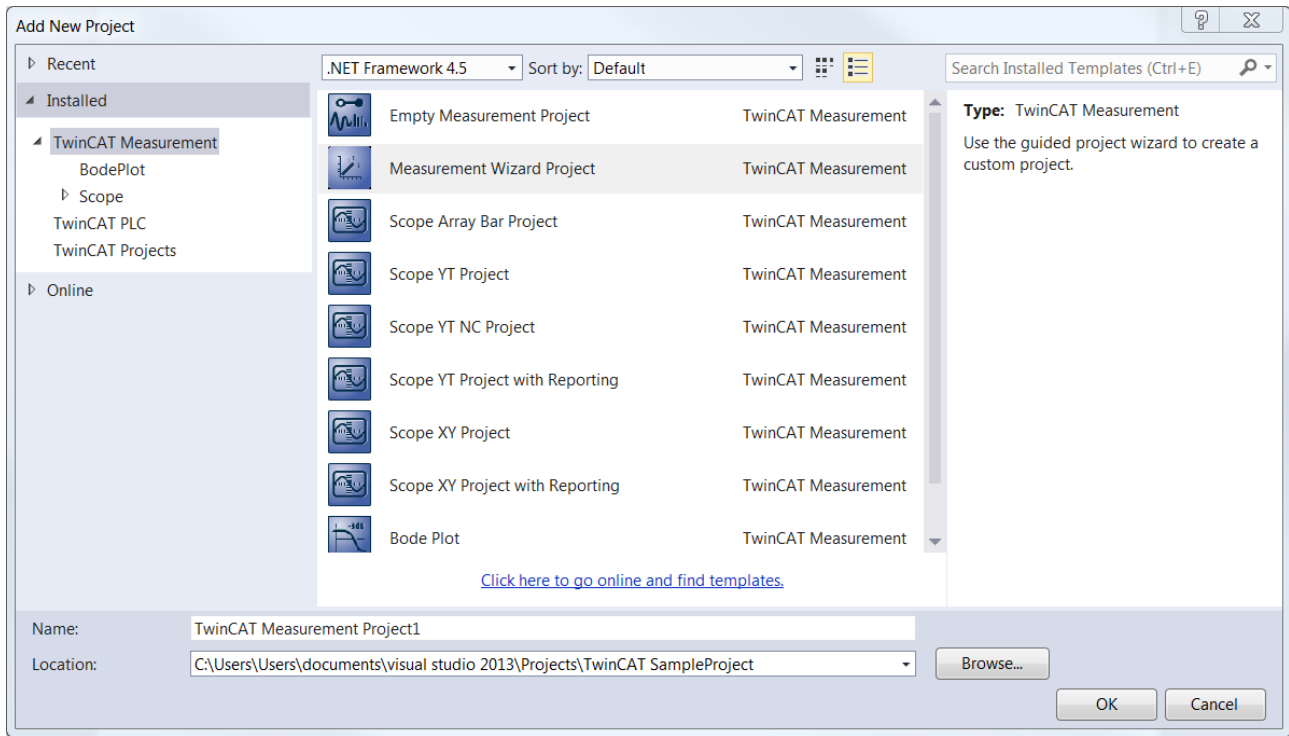
⇨ The Measurement Wizard for TC3 Scope View, TC3 Bode Plot and TC3 Filter Designer opens.
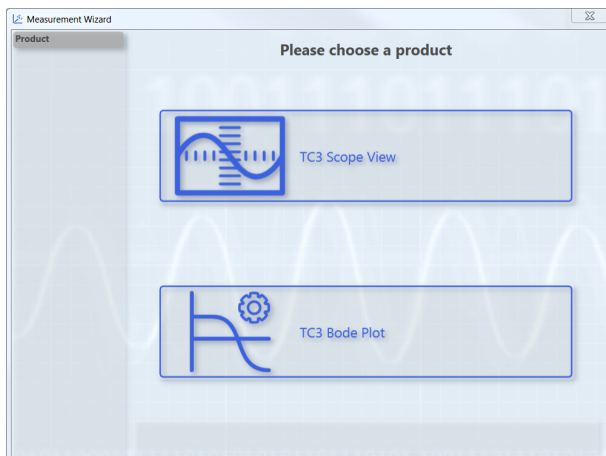
3. Select **TC3 Scope View**.
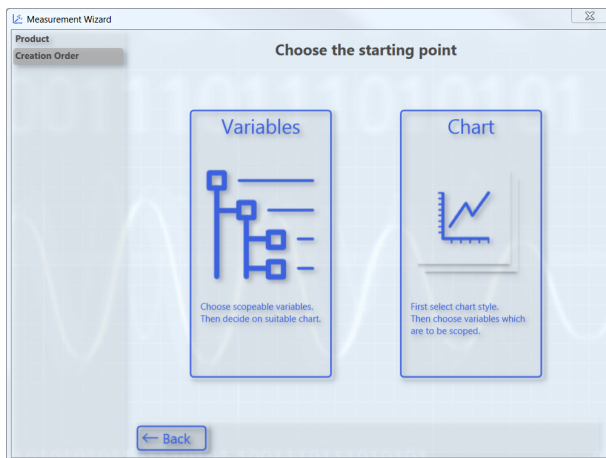
⇨ If you wish to select the variables to be recorded and displayed as a graph first and then the appropriate chart, select the "Variables" option [▶ 120]. If you wish to select the chart first and then the variables, select the "Chart" option [▶ 121].

Selecting variables and then chart (Variables option)

✓ In the Measurement Wizard you selected the **Variables** option as the starting point.

1. Select the target system and the corresponding ADS port. Optionally you can open the **TwinCAT Static Routes** dialog via the **Manage Targets** button in order to obtain information about the routes of the local TwinCAT system and to add new routes. Click on **Next.**
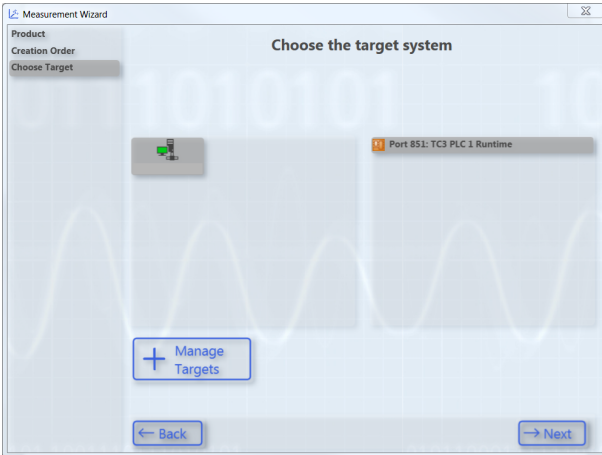
2. Select the variables that are to be recorded and displayed as a graph. To do this, navigate through the menu or enter a variable name in the filter text field. Optionally you can select a different scaling factor and an offset. Furthermore, you have the option to specify a warning level and an alert level. With these values you configure the dynamic style function for these variables. From the warning value the graph is drawn in yellow and from the alert in red. Click on **Next**.
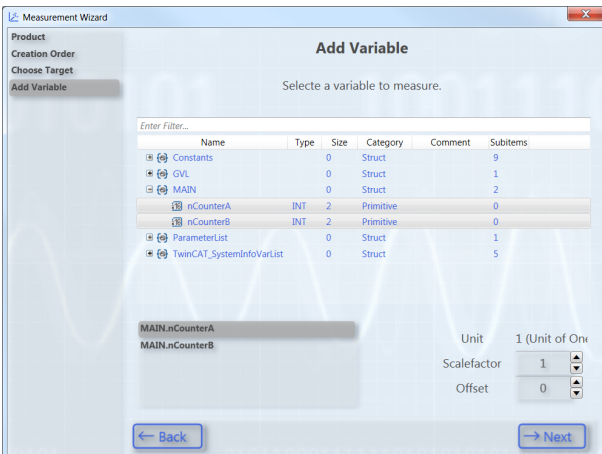
3. Select the chart type. You can only select chart types that match your variable selection. Here is an example of an XY chart.
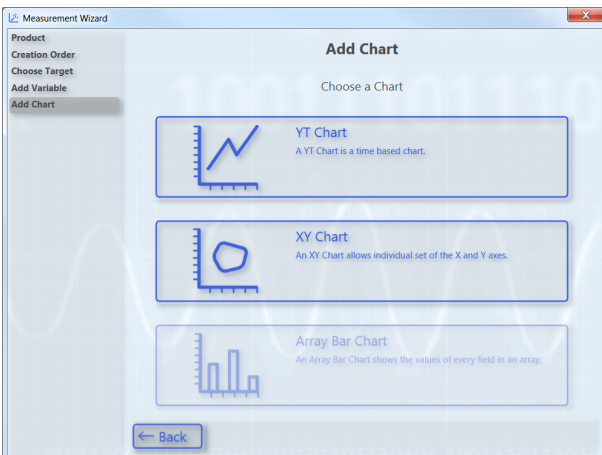
4. Configure the axes. To do this, assign variables to the axes by marking the respective pairs. You can reset the selection with **Undo**. Click on **Next**.
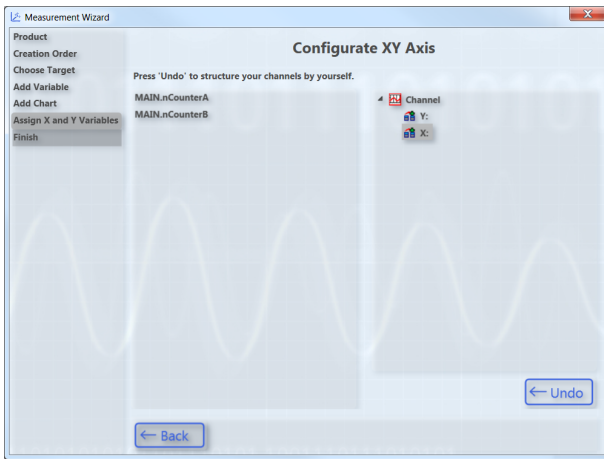
⇨ The subsequent configuration step **Finish** shows an overview of the settings.
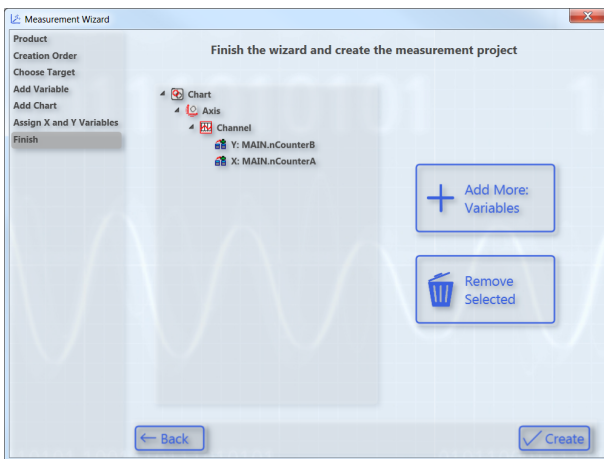
5. To delete a configured element again, mark it and click on **Remove Selected**. To add further variables to the configuration, click on **Add More: Variables.** Click on **Create** to end the configuration and create the project.

⇨ A TwinCAT Measurement Project with a ScopeWizard project is added to the Solution.

Selecting chart and then variables (Chart option)

✓ In the Measurement Wizard you selected the **Chart** option as the starting point.

1. Select the chart type.
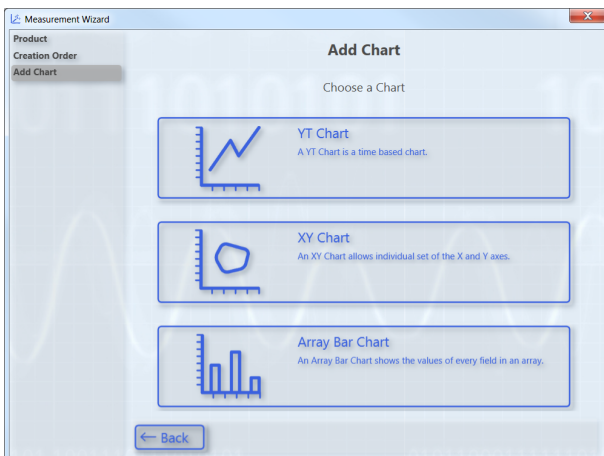
2. Select the target system. Optionally you can open the **TwinCAT Static Routes** dialog via the **Manage Targets** button in order to obtain information about the routes of the local TwinCAT system and to add new routes. Click on **Next.**
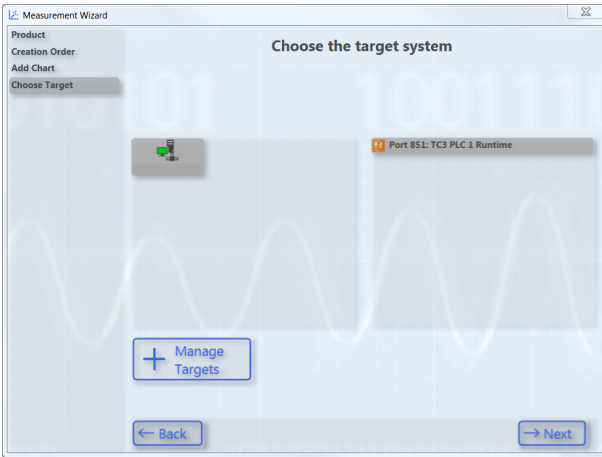
3. Assign a variable to the Y-axis. To do this, navigate through the menu or enter a variable name in the filter text field. Optionally you can select a different scaling factor and an offset. Furthermore, you have the option to specify a warning level and an alert level. With these values you configure the dynamic style function for these variables. From the warning value the graph is drawn in yellow and from the alert in red. Click on **Next**.
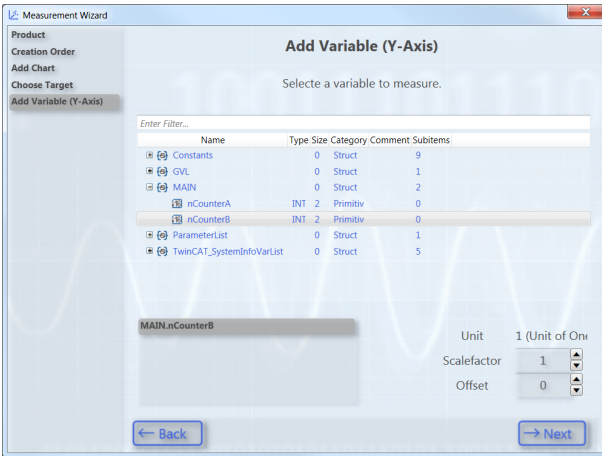
4. Assign a variable to the X-axis. To do this, navigate through the menu or enter a variable name in the filter text field. Optionally you can select a different scaling factor and an offset. Click on **Next**.
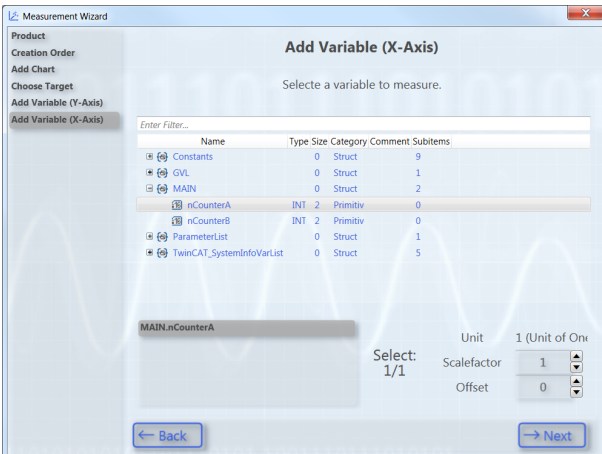
⇨ The subsequent configuration step **Finish** shows an overview of the settings.
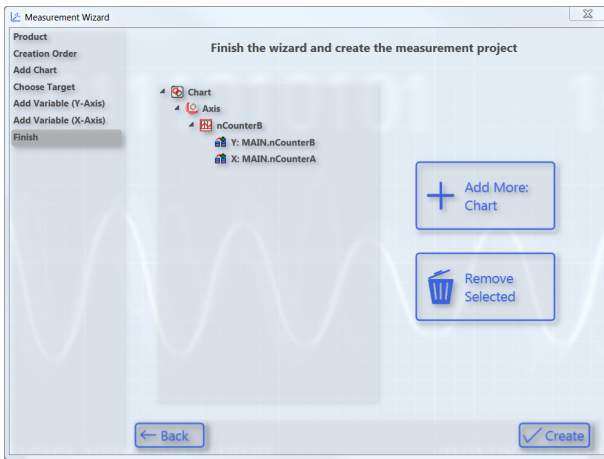
5. To delete a configured element again, mark it and click on **Remove Selected**. To add a further chart to the configuration, click on **Add More: Chart.** Click on **Create** to end the configuration and create the project.

⇨ A TwinCAT Measurement Project with a ScopeWizard project is added to the Solution.

Editing Scope projects with the Scope Wizard

To edit a Measurement Wizard project or a Measurement Scope project with the aid of the Measurement Wizard, select **Update Project in Scope Wizard** in the context menu of the scope project in the **Solution Explorer**. The Measurement Wizard opens in which you can modify the configuration of the project.

> **ℹ** **Project update with Scope Wizard**
>
> The prerequisite for this function is that the original target system of the configuration is available on the engineering system.

5.2.1.2   Saving and loading a configuration

Saving a Scope configuration:

In the **File** menu select the command **Save**. (default keyboard shortcut: **[Ctrl] + [S]**)

Loading a Scope configuration

-   In the **File > Open** menu, select the **Project/Solution** command and, in the dialog which then opens, select the project folder or the Measurement Project.
-   Scope files (.tcscope | .sv2 | .svd) can be opened directly by double-clicking. Since a Measurement Project as the basis is missing, a selection dialog appears in which you can create a new project (Empty Measurement Project). The Scope called by a double-click is added to the newly created project. At the same time a copy of the file is placed in the new project directory.
-   Measurement project files (.tcmproj) and solutions (.sln) can be opened directly by double-clicking.

5.2.1.3   Sending a configuration by e-mail

In the context menu of the Scope instance to be sent, select the command **Send Project By Email**.

If the Scope contains data (Scope state: replay), you can select whether the data should also be sent.

5.2.2 Data

5.2.2.1 Formats

The TwinCAT Scope has been on the market since 2008 and now supports a series of its **own** file formats due to new developments. This page provides an overview of them. All formats can always be used with the latest Scope version.

Configuration:

Scope version	File format
All in TwinCAT Scope2 (TwinCAT 2)	.sv2
From TwinCAT 3 Scope	.tcscope
Since TwinCAT 3 Scope version 3.3.3140.0	.tcscopex

The configuration files can always be converted into the more up-to-date formats. i.e. .sv2 >> .tcscope, .tcscope >> .tcscopex and also .sv2 >> .tcscopex.

Data:

The data are saved in .svd files in TwinCAT Scope 2 and also in TwinCAT 3 Scope up to version 3.3.3140.0. From version 3.3.3140.0 the files are saved in the new .svdx format. Old .svd files can also be converted into .svdx again.

From version 3.3.3140.0 a .tcscopex file is saved for the configuration. The data are written to a .svdx file.

5.2.2.2 Saving and loading data

Data that have been recorded with the TC3 Scope View can be saved as a .svdx file. A .svdx file is a specific, non-disclosed data format of the TwinCAT 3 Scope View.

Saving data

- Select the **Save Data** command in the **Scope** menu or in the toolbar. This creates a Scope data file (svdx) that can be added directly to the measurement project or saved in a directory via the directory selection dialog.
- If you add multiple files to a measurement project, the name of the svdx file is automatically incremented with a digit.

Loading data

The following options are available for loading data:

- Double-click the desired .svdx file in the file system. Visual Studio opens and the dialog **New Project** appears. Select the **Empty Measurement Project**. The .svdx file is added to the new measurement project.
- If a measurement project already exists, the .svdx file can be selected and opened via the command **Add Existing Item**.
- Open the measurement project, to which the .svdx file that was created has been added. In menu **File > Open** select the command **Project/Solution**; in the dialog that opens open the corresponding project.

Reloading of svdx files

If already save data are no longer displayed due to a disconnection of the Scope's internal communication to the Scope Server, the **Reload** command is automatically available in the context menu for the .svdx file. You can reload the data with this command.
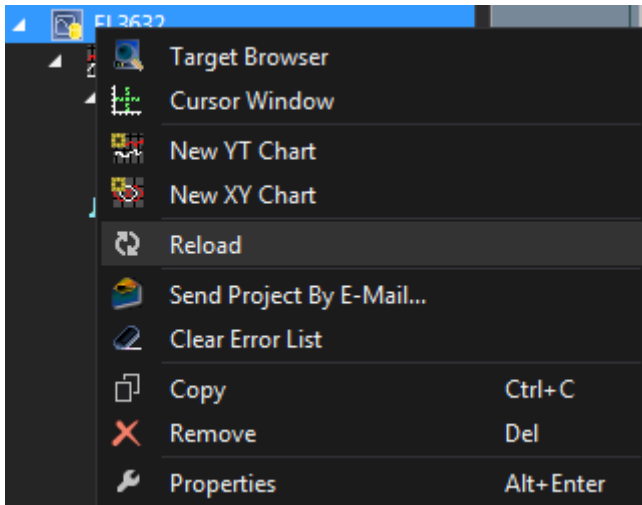
Extracting Scope configurations from an svdx file

A .svdx file is visually identified as such. A yellow data symbol is displayed in the **Solution Explorer**. This enables exclusively data analysis within the file.

If you want to reuse the configuration, for example to make a new recording with the same configuration, select the command **Extract Scope Configuration** in the **Scope** menu.
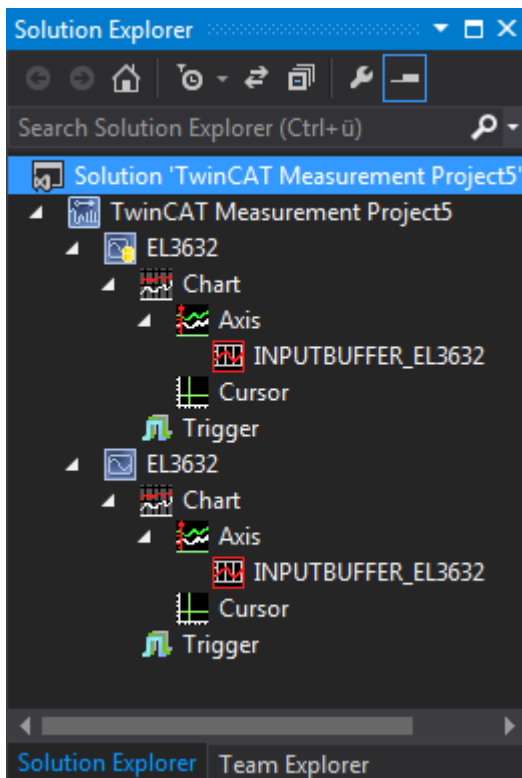
5.2.2.3   Exporting data

In order to enable the further processing of the data recorded with the TwinCAT Scope, the data can be exported in various other data formats. The TwinCAT Measurement Export Wizard is provided in order to make this as simple and as individual as possible.

1. Open the Export Wizard via the **Scope** Menu > **Export…**

2. On the first page you can select the desired target format. The original svd format is thereby selectable again as the target simply in order to reduce existing scope recordings with regard to the number of channels or their duration. Further formats are csv, svb, tdms and dat.
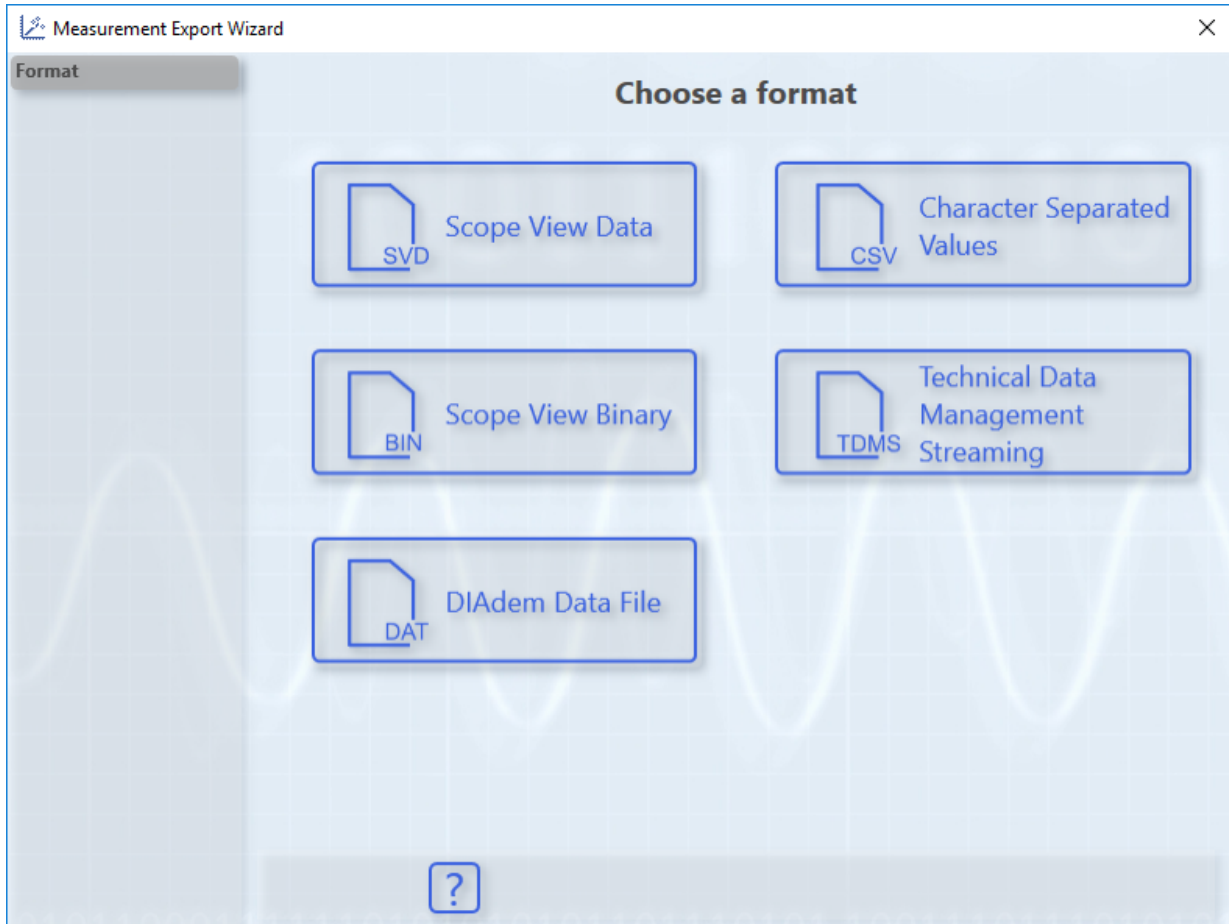
3. Subsequently, the channels to be exported in the new format can be selected via **Select channels**. Confirm with **Next**.

4. The timeframe that was defined by the recording is selected in the **Select time period** dialog. Select this timeframe graphically with the mouse or textually with **Export start time** and **Export end time**. Confirm with **Next**.
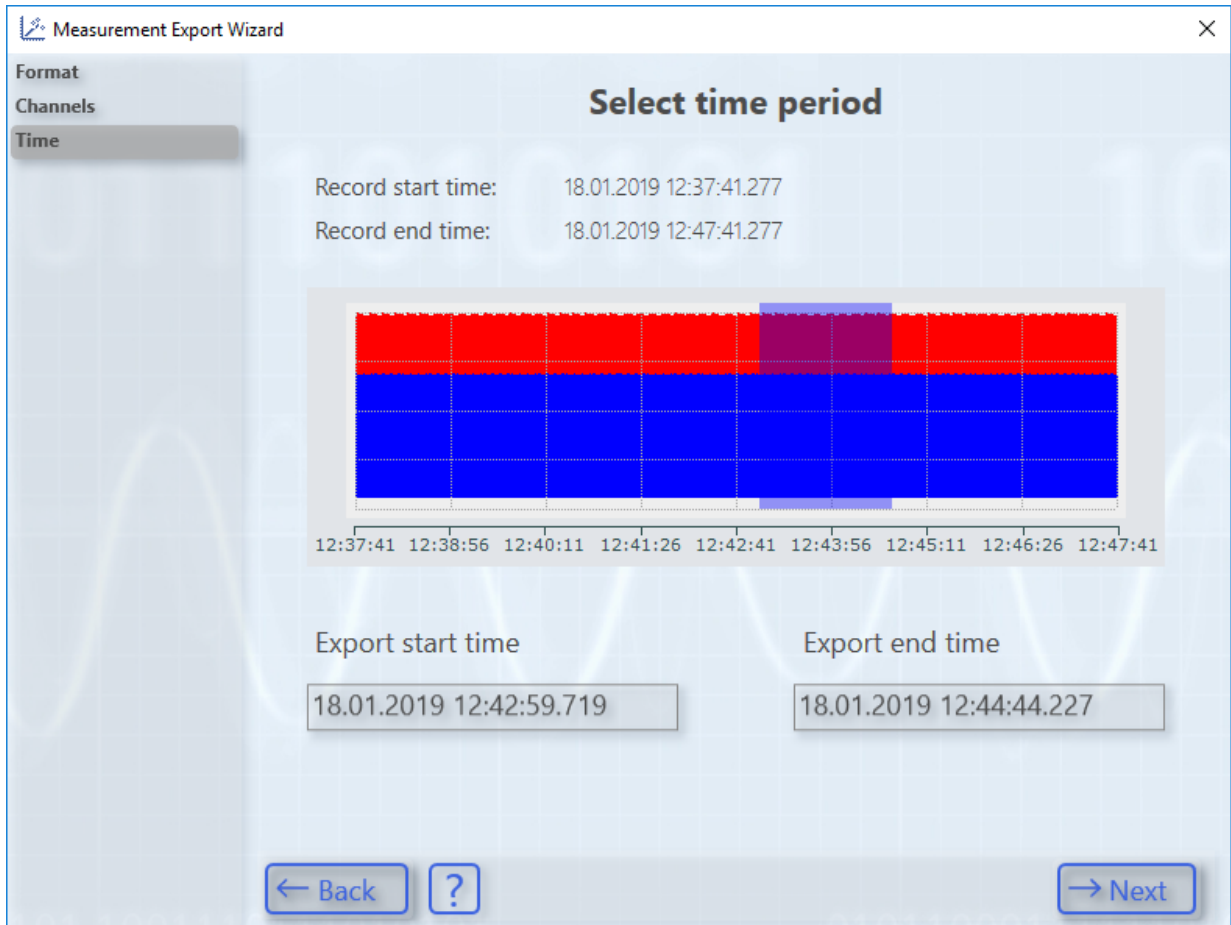
5. **Configure Properties for CSV-Export** enables individual settings to be made for each data format. Confirm your settings with **Next.**
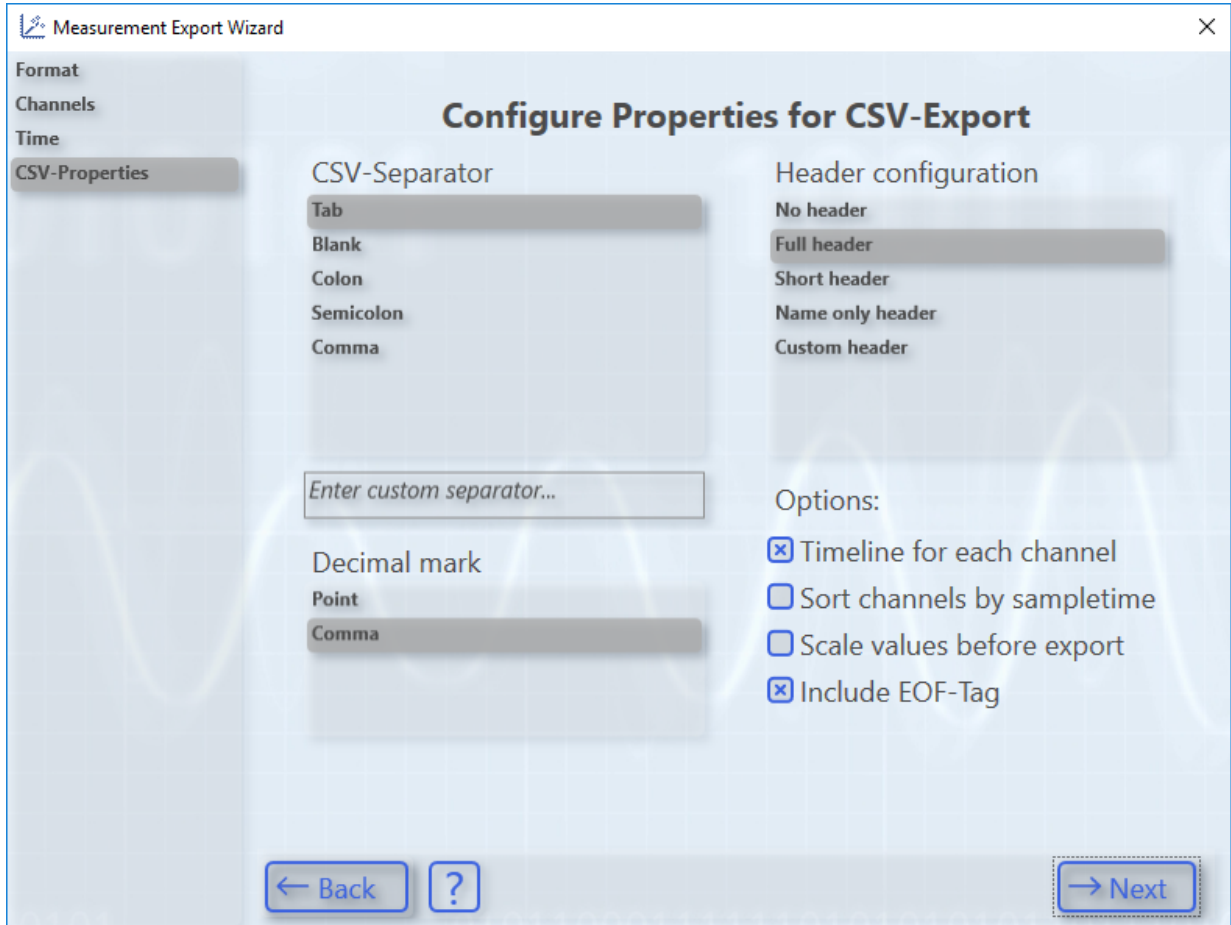
6. **Selected CSV-Configuration** provides a summary. The generation of the new file/files is started with **Create**.
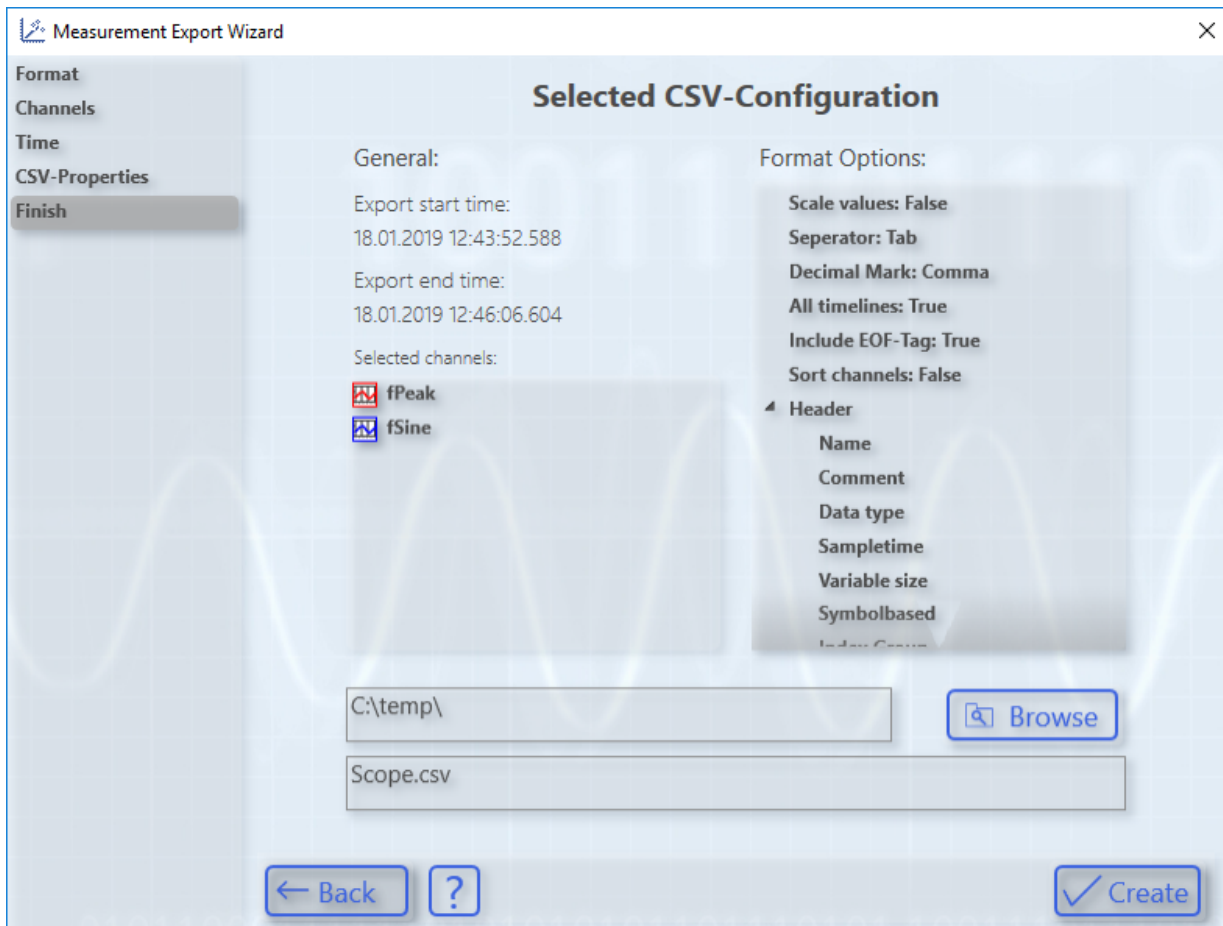
5.2.2.3.1 Export to csv

The export of data to a csv file enables the following individual settings to be made.
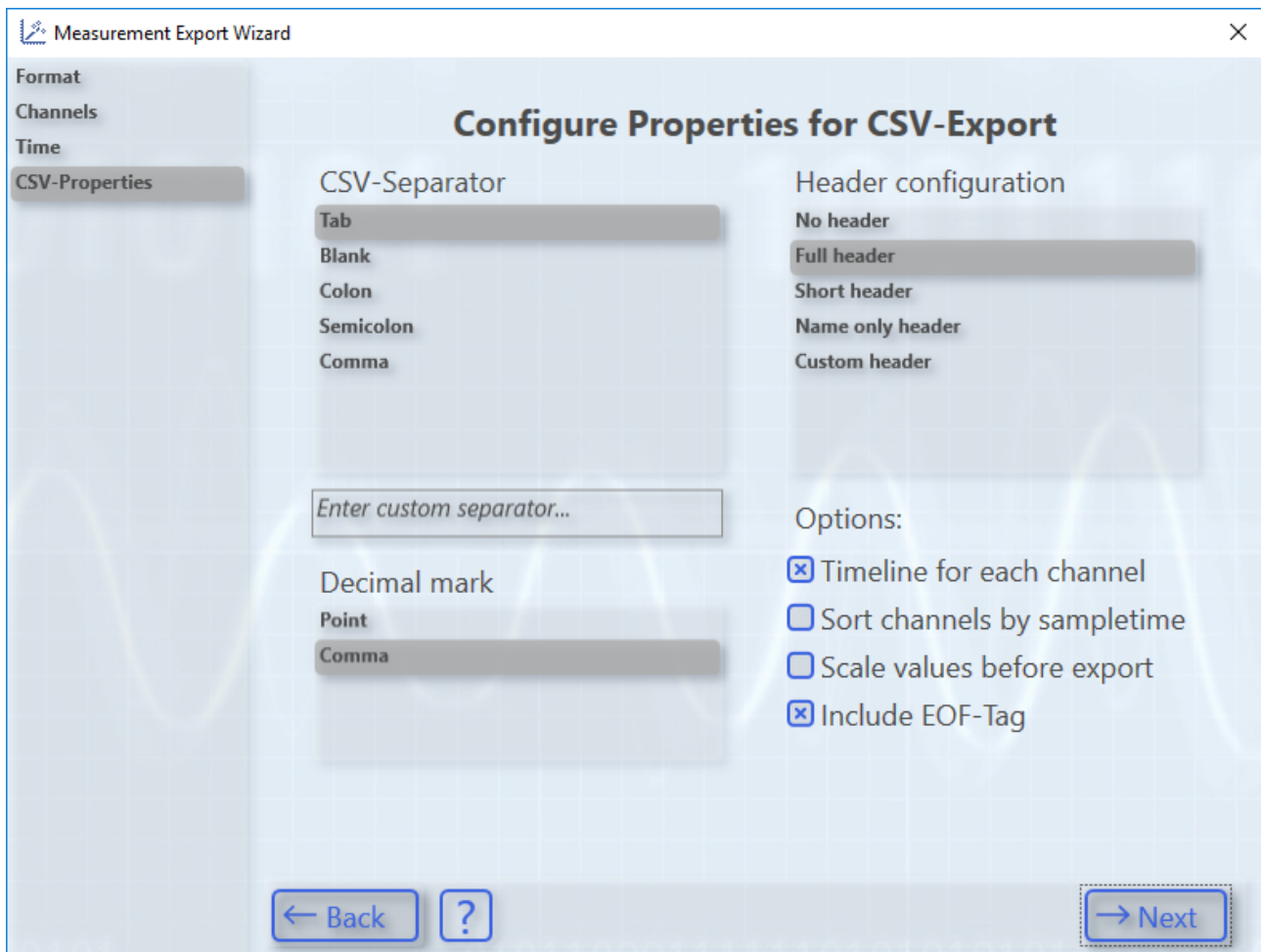
The separators and decimal markings to be used for the export can be selected. To do this the header with some additional information can be switched completely on or off. An individual header is also possible via the **Custom header** option. The page opens as an intermediate page by confirming **Next**.

Furthermore, you can decide in the options to export the time axis, which is inserted for each channel, only for different sampling times. Sorting can then be done on the basis of the sampling rate. You can also export scaled or unscaled values. An End-Of-File tag can optionally be written to the csv.

Individualization of the csv header when selecting **Custom header**:

5.2.2.3.2 Export to svb

The TwinCAT Scope also supports the export of data to a binary format specified by Beckhoff. The file extension of the created file is svb. The following parameters can be set:
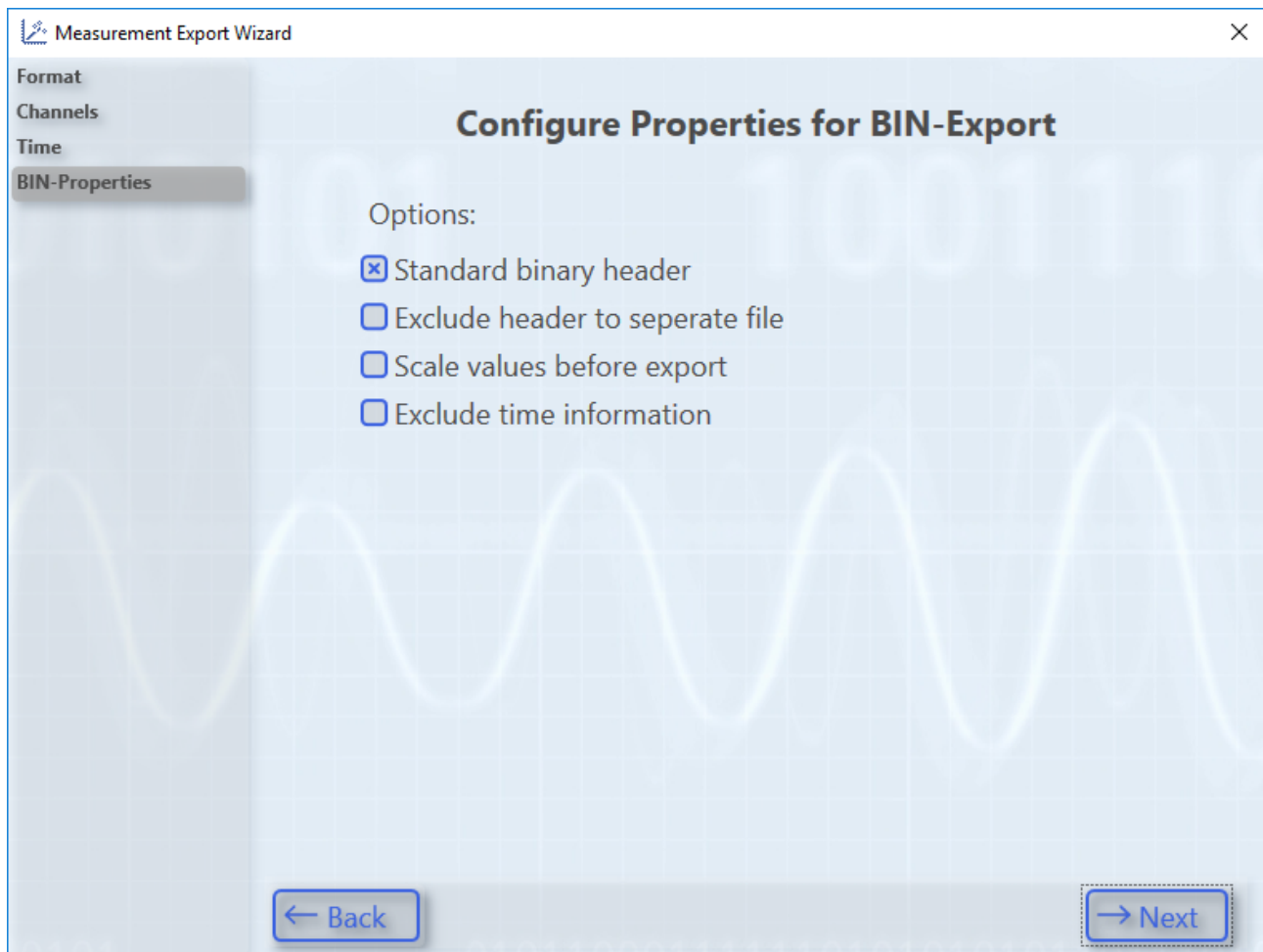
The header can be shown and hidden and also moved to its own file. You can choose whether to export the values in scaled or unscaled form, with and without sampling rate/time information.

Format description for developers

The binary data format is based on the text format but contains no separators. Instead the byte lengths of all variable data types (such as character strings) are contained in the corresponding headers. All times are specified in the file time format: 1 tick = 100 ns; origin is 1.1.1601 0h. All characters are stored in UTF8.

Variable	Size (Bytes) or Number of character	DataType
MAIN-HEADER		
HeaderSize	8	Int64
NameSize	4	Int32
Name	NameSize	Array of Char
StartTime	8	Int64
EndTime	8	Int64
ChannelCount	4	Int32

Variable	Size (Bytes)	DataType
Channel - Header #1		
ChannelHeaderSize	8	Int64
NameSize	4	Int32
Name	NameSize	Array of Char
NetIdSize	4	Int32
NetId	NetIdSize	Array of Char
Port	4	Int32
Sample Time	8	Int64
SymbolBased	1	bool (as byte)
SymoblNameSize	4	Int32
Symbol name:	SymbolNameSize	Array of Char
CommentSize	4	Int32
Comment	CommentSize	Array of Char
IndexGroup	8	Int64
IndexOffset	8	Int64
DataTypeSize	4	Int32
DataType	DataTypeSize	Array of Char
DataTypeId	4	Int32
VariableSize	4	Int32
Samples in File	8	Int64
Data in File	8	Int64
File-StartPosition	8	Int64
Offset	8	Real64
Scalefactor	8	Real64
Bitmask	8	Int64
Channel-Header #2		
...		
Channel-Header #MainHeader.ChannelCount		

Variable	Size (Bytes)	DataType
Data Channel #1		
DataPoint #1		
Timestamp	4	UInt32
Value	ChannelHeader1.VariableSize	ChannelHeader.DataType
DataPoint #2		
...		
DataPoint ChannelHeader #1.Samples In File		
Data Channel #2		
...		

Data Channel #MainHeader.ChannelCount

5.2.2.3.3 Export to tdms

The tdms format is a commonly used format in measurement technology and can be used with the aid of the TwinCAT 3 Scope View Professional License (TE1300).

Apart from an author and a title for the measurement project conducted, you can also make settings with regard to the sampling rate or time axis. Furthermore, you can adopt physical units from the Scope and decide whether or not the values will be scaled. A grouping can also be specified for the tdms file.
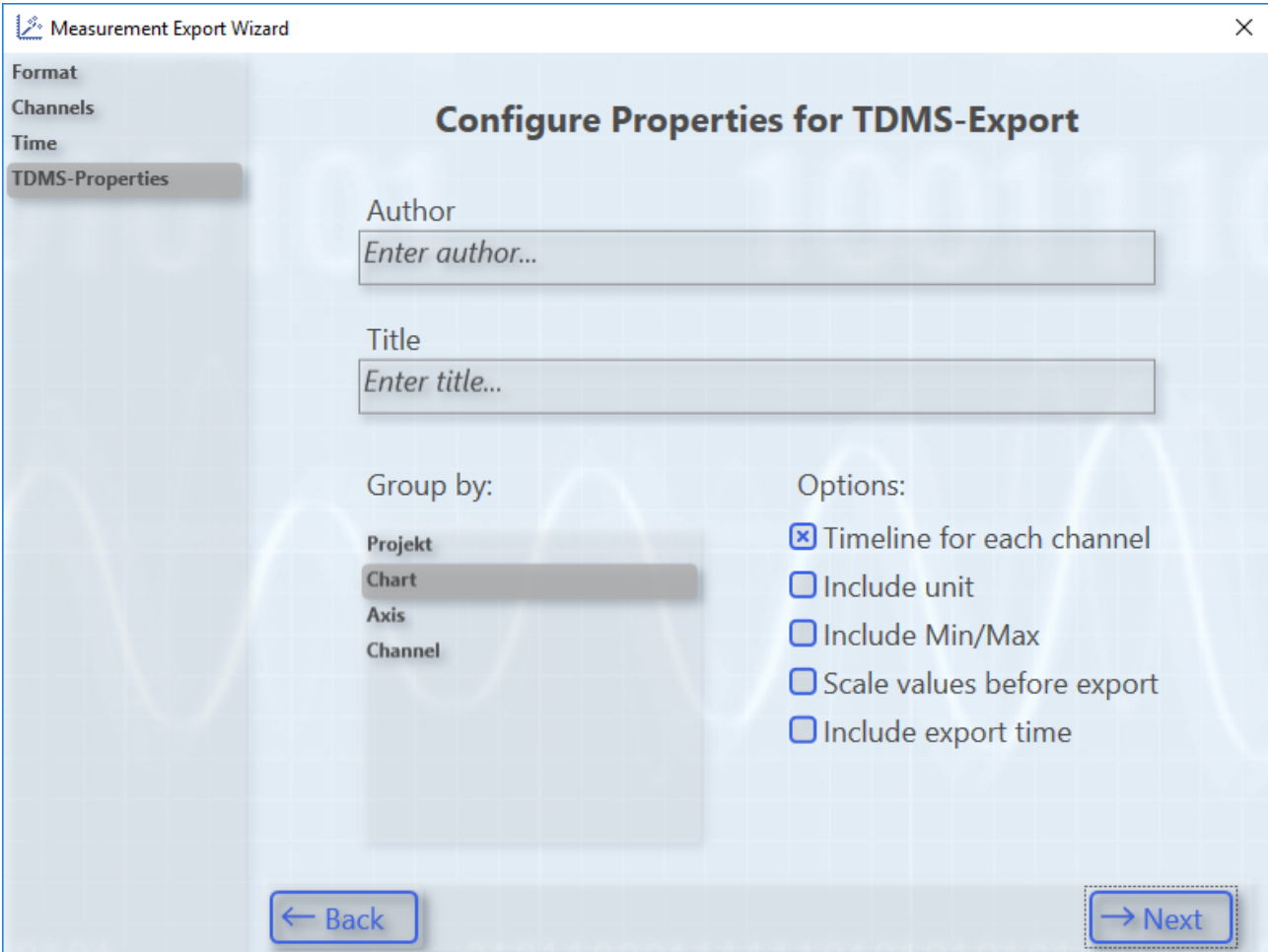
5.2.2.3.4 Export to dat

The dat format is a commonly used format in measurement technology and can be used with the aid of the TwinCAT 3 Scope View Professional License (TE1300). The implementation of dat is not standardized, for which reason minor deviations can occur in the implementation.

The following settings can be made:
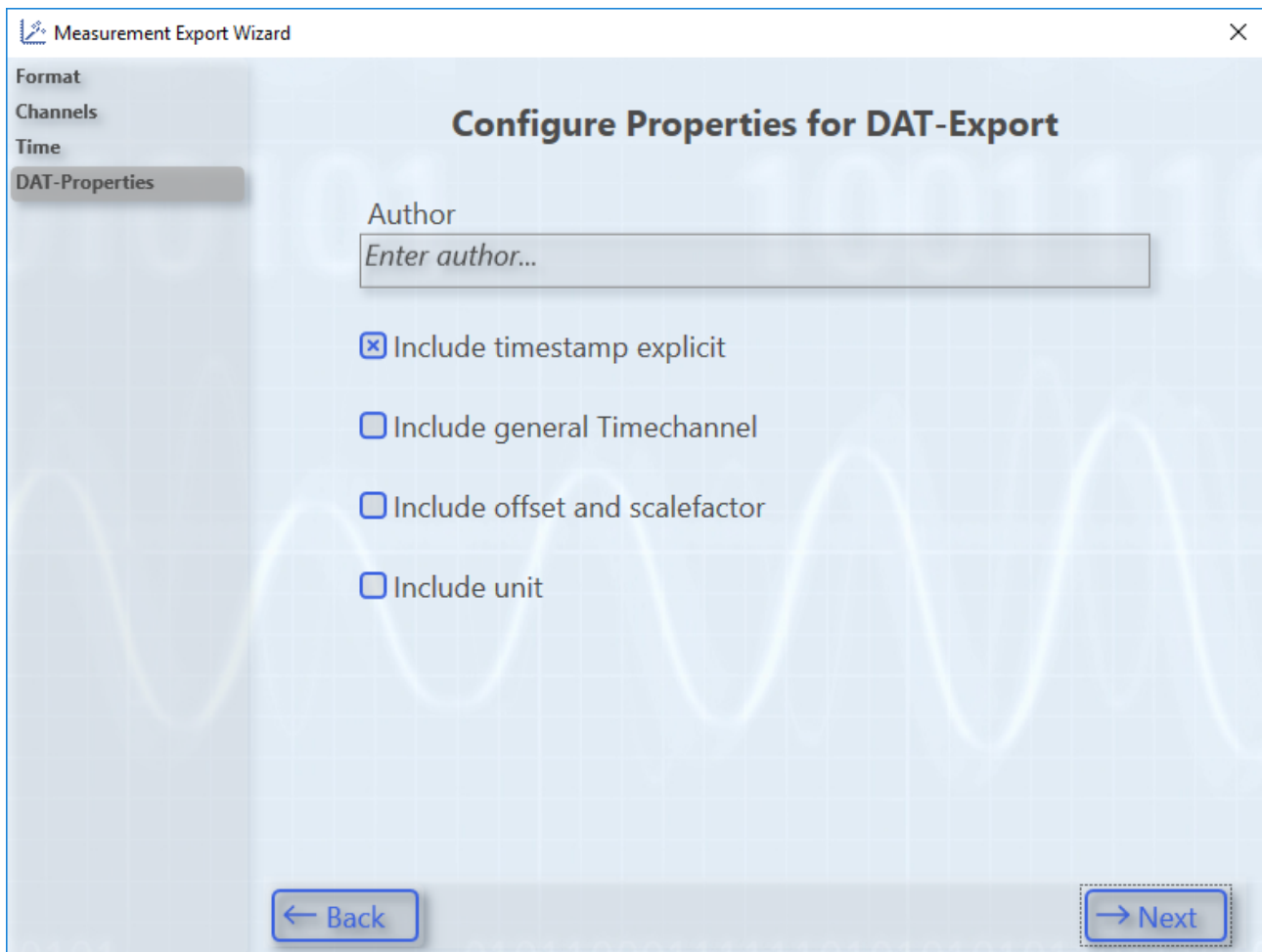
5.2.2.3.5   Automated export

Along with the Scope, the **TwinCAT 3 ScopeExportTool** is additionally installed in the TE130X Scope View and in the TF3300 Scope Server directory. This can be used to subsequently convert contents of svd files into the supported export formats - without Visual Studio. The tool comes complete with its documentation. It always requires a Professional License for the tdms and dat formats.
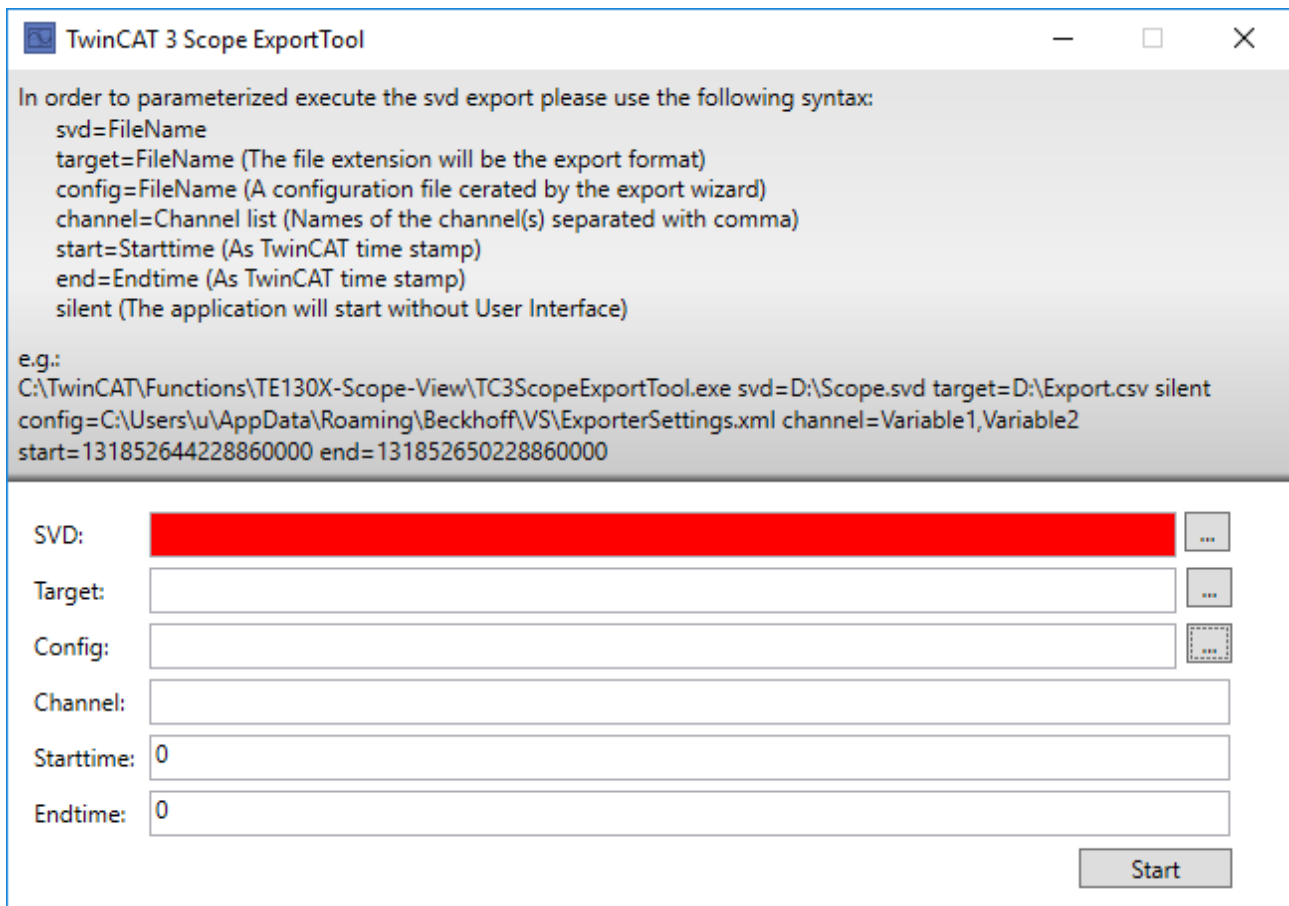
The TC3ScopeExport tool can also be used for export via a command line call, based on specified parameters. The interface shown here can be suppressed by entering the keyword "silent" in the command line. In this case the export is executed in the background.

To automate the export, the tool can be called from the PLC via NT_StartProcess. A small code example with fixed strings and fbStartExport as instance of NT_StartProcess is shown below:

```
fbStartExport(
    NETID:= '',
    PATHSTR:= 'C:\TwinCAT\Functions\TE130X-Scope-View\TC3ScopeExportTool.exe',
    DIRNAME:= 'C:\TwinCAT\Functions\TE130X-Scope-View',
    COMNDLINE:= '"svd=c:\Scope Project.svd" target=c:\TestExport.csv silent',
    START:= bStart,
    TMOUT:= T#20S,
    BUSY=> ,
    ERR=> ,
    ERRID=> );
```

5.3   Menus and options

5.3.1   Menu commands

You can select various Scope functions in the **Scope** menu. The menu will have different contents depending on the object selected in the **Solution Explorer**. (For this reason, the screenshot shown below differs from the description). You can also take the corresponding key assignments from the **Scope** menu.

SCOPE	ADS	ANALYZE	WINDOW	HELP

Target Browser

Cursor Window

Save Data

Apply Defaults

Send Project By E-Mail...

Clear Error List

New YT Chart

New XY Chart

New Array Bar Chart

Copy	Ctrl+C
Paste	Ctrl+V
Delete	Del

Save Data

Export to CSV

Export to Binary

Extract Scope Configuration

Export to DAT

Export to TDMS

Upload Configuration to Target

Scope Messages ▶

Local Scope Server...

Graphic Library ▶

Options...

Target Browser	Opens the **Target Browser**.
Cursor Window	Opens the cursor window.
Send Project By E-Mail...	Corresponds to the command **Send Project By E-Mail...** in the context menu of the Scope instance to be sent. If the Scope contains data (Scope state: replay), you can select whether the data should also be sent.
Clear error list	Deletes all entries (Error | Warning | Message) of the currently active Scope from the error list.
Change Ads Symbol...	Dialog for the replacement of character strings. The symbol names of all channels located below the selected element are edited.
Change Index Group...	Dialog for incrementing / decrementing the Index Group / Index Offset. The acquisitions of all channels located below the selected element are edited.
New YT Chart	Creates a new YT chart within the Scope.
New XY Chart	Creates a new XY chart within the Scope.
New Array Bar Chart	Creates a new array bar chart within the Scope.
New Axis	Creates a new axis within the chart. If the selected element is not a chart, then a new chart is additionally created.
New Empty Channel	Creates a new channel within the axis. If no axis is selected, then a new axis is additionally created.
Delete	Deletes the element currently selected in the Solution Explorer.
Export to CSV	Exports the data to a CSV or TXT file.
Export to Binary	Exports the data to a binary file.
Export to DAT	Exports the data to a DAT file.
Export to TDMS	Exports the data to a TDMS file.
Extract Scope Configuration	Extracts the Scope Configuration from the selected .svd file.
Upload Configuration To Target	Downloads the current configuration as a .tcscope file to a user-selected target system.
Save Data	After stopping the recording, the current data including the configuration can be saved in an .svd file. The generated Scope data file (.svd) can be added directly to the Measurement Project or saved in a directory of your choice via the directory selection dialog.
Scope Messages	Some frequently occurring message boxes in the scope are equipped with a checkbox – **Remember my answer and don't ask again!** – so that the question is not repeated the next time. This makes the handling of the program individually adjustable for each user. If a message box has been deactivated, it can be activated again here.
Local Scope Server...	Opens the configuration interface of the Scope Server.
Options...	Opens the Visual Studio Options window and selects the Scope entry.

5.3.2   Toolbar commands

The Scope recording can be started and stopped and the recorded data saved with the commands from the TwinCAT Measurement toolbar.

If the toolbar is not visible by default or was closed, you can call it up again via **View > Toolbars**.
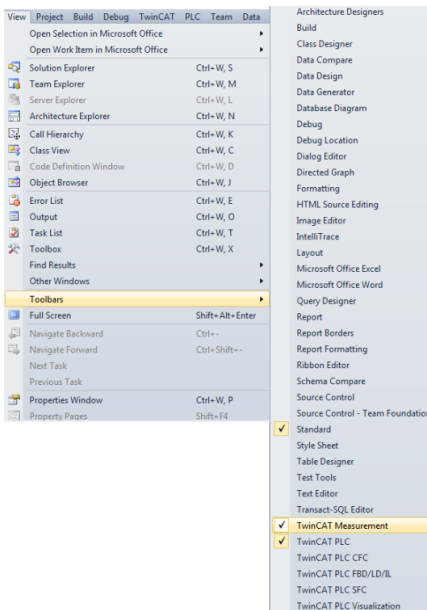
The following elements are available by default in the toolbar:

	Start Record	All required servers are called and the recording settings and connected channels are entered. If data are already present from a preceding session, then a query appears asking whether the current data should be saved. Afterwards the recording is started in accordance with the above settings. After the start it is no longer possible to change the Scope settings or acquisition or to add or remove channels.
	Stop Record	The recording is stopped. However, the recorded data remain on the connected servers and can be viewed. After stopping the recording, it is possible to change the Scope configuration. The current data will then be lost, however, if they are not saved.

You can expand the toolbar and adapt it individually as you wish. To do this, select the **Add or Remove Buttons** command in the toolbar context menu (can be displayed on the right-hand side of the toolbar) and, in the menu which then opens, select the **Customize** command. The **Customize** dialog opens.
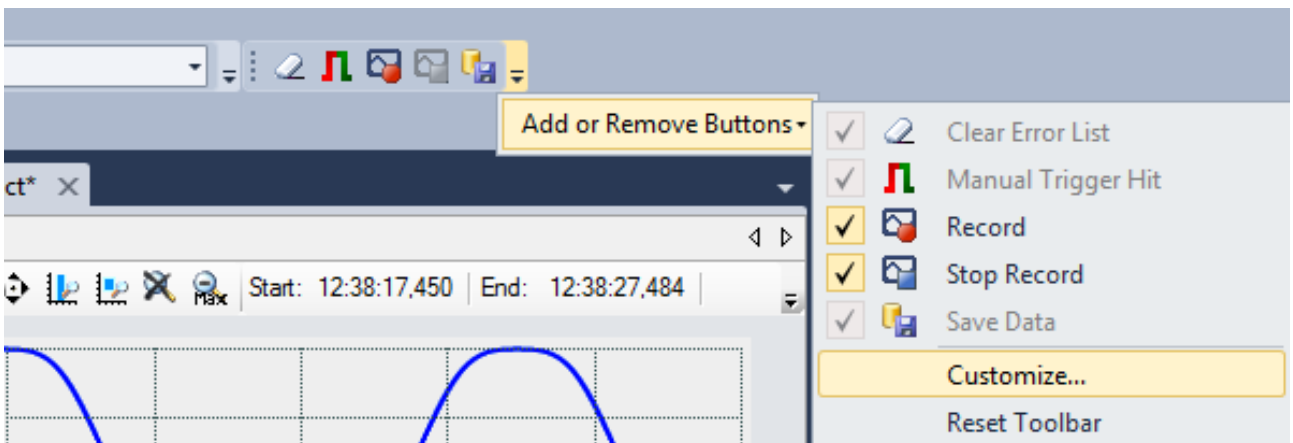
The commands belonging to the Scope can be found under **Add Command** in the **Scope** category.
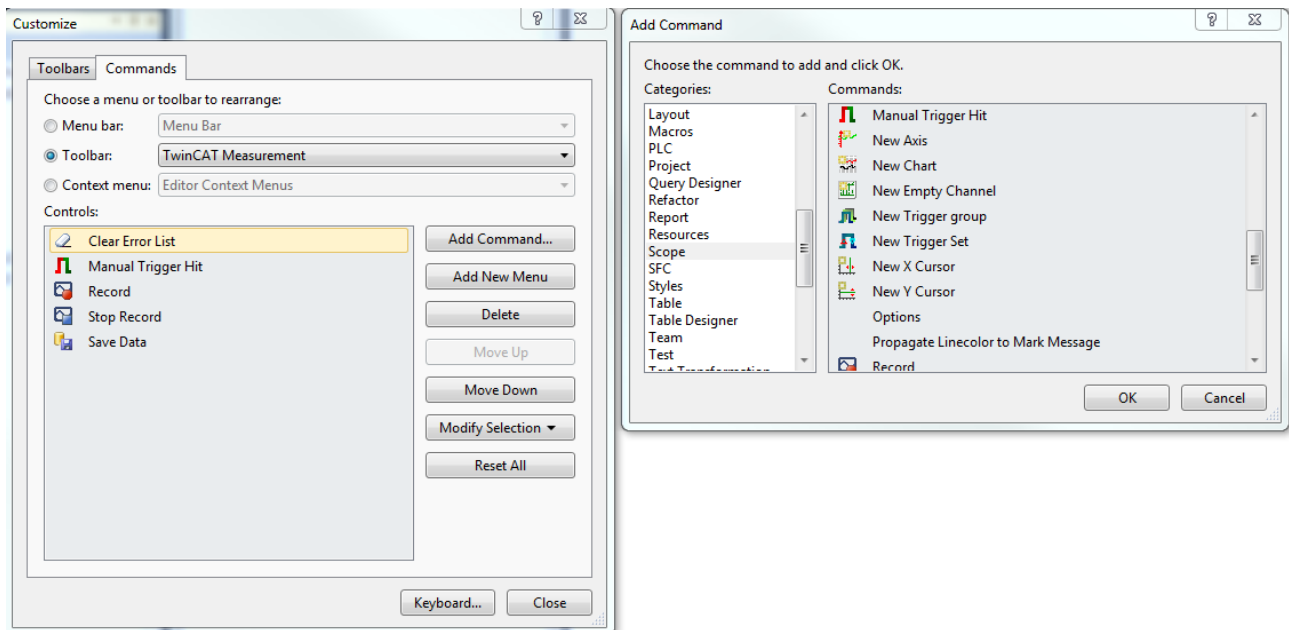
5.3.3   Options

You can define the standard settings in the TwinCAT Measurement options. Open the options in Visual Studio via the **Options**… command in the **Scope** menu.

Options dialog
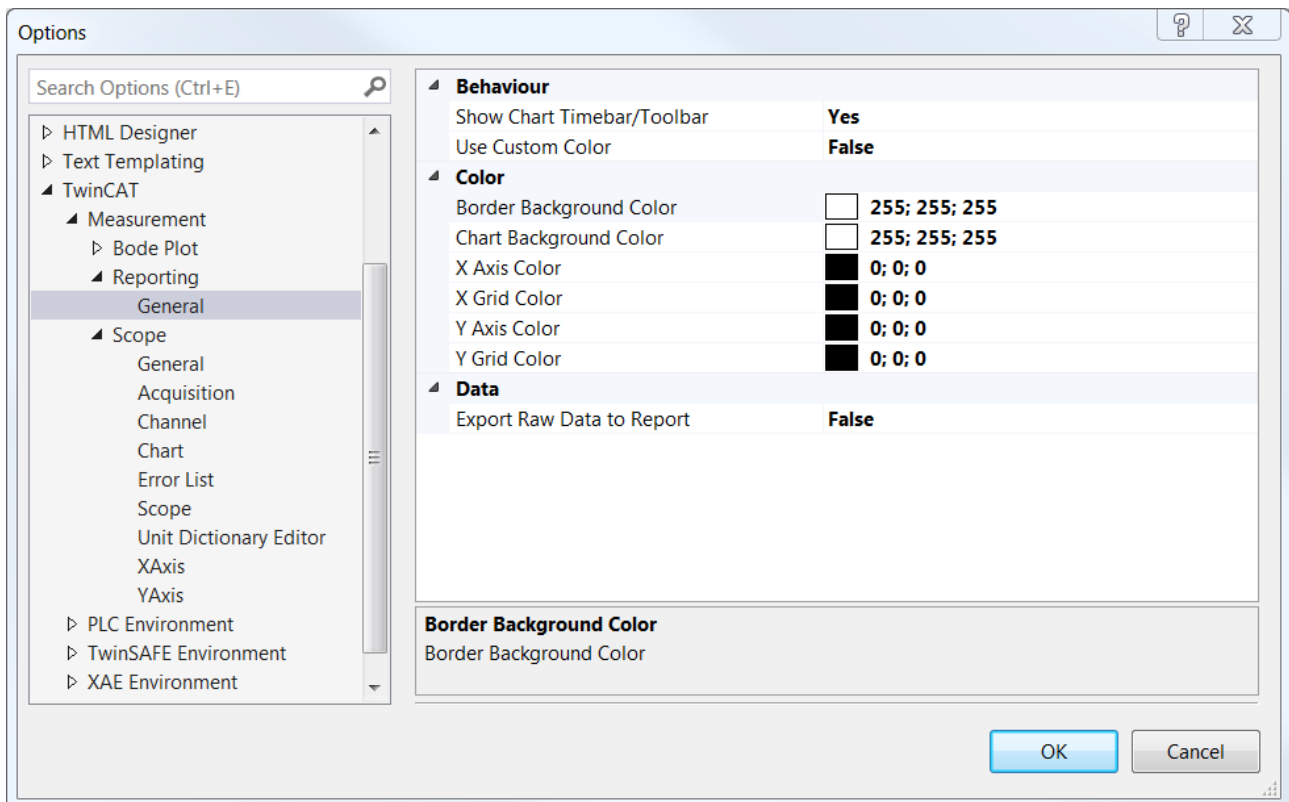
Reporting

General category

Show Chart Timebar/Toolbar	Specifies whether the time bar and toolbar of the Scope chart should be visible in the printing template.
Use Custom Color	TRUE: The following color settings will be applied before the chart is exported for the report. (You can select color combinations that reduce the ink consumption for printing).
Border Background Color	Color for the border of the chart
Chart Background Color	Color for the background of the chart
X-Axis Color	Color for the labeling of the X-axis
X-Axis Color	Color for the grid of the X-axis
Y-Axis Color	Color for the labeling of the Y-axis
Y-Axis Color	Color for the grid of the Y-axis
Export Raw Data to Report	Provides the raw chart data for the report. (Significantly extends the time it takes to generate a report | Only required for creating custom charts).

Scope

General	Settings for the tracing of the Scope View product components
Acquisition	Standard acquisition data (will be used if empty channels are inserted) See: Acquisition [▶ 34]
Channel	Standard channel settings (will be used if empty channels are inserted) See: YT channel style [▶ 32]
Chart	Standard chart settings (will be used if new charts are inserted) See: YT chart properties [▶ 27]
Error list	• Clear on Record: Deletes the respective error messages for the Scope instance from the error list as soon as a Scope changes to the record mode. • Show Clear List Command: Shows the context menu command **Clear Error List...**. • Show Messages: Shows messages from the Scope in the error list. • Show Warnings: Shows warnings from the Scope in the error list.
Scope	Standard Scope settings (will be used if new Scope instances are inserted) See also: Scope nodes [▶ 23]
Unit Dictionary Editor	The Unit Dictionary is a sort of dictionary for all unit elements. These include base units, transformations and user units. All available unit elements stored in the dictionary can be viewed with the aid of the Unit Dictionary Editor. In addition, there is an option to add new values via the editor. See: Units [▶ 85] and Unit Dictionary Editor [▶ 101]
X-axis	Standard axis settings for the X-axis (will be used if new axes are inserted) See: YT axis properties [▶ 31]
Y-axis	Standard axis settings for the Y-axis (will be used if new axes are inserted) See: YT axis properties [▶ 31]

5.4   Scope Diagnostics

5.4.1   Debug messages

For a very deep diagnosis of the behavior of the TwinCAT Scope itself, debug messages can be activated in both main components, View and Server. This can take place at various levels. The messages are described in corresponding trace log files in the TwinCAT Functions directory, in each case under View and Server.

View:

In the Options [▶ 141] under General there is an option to activate a trace log for the control, the charting and the measurement. The target path is also specified and adjustable.
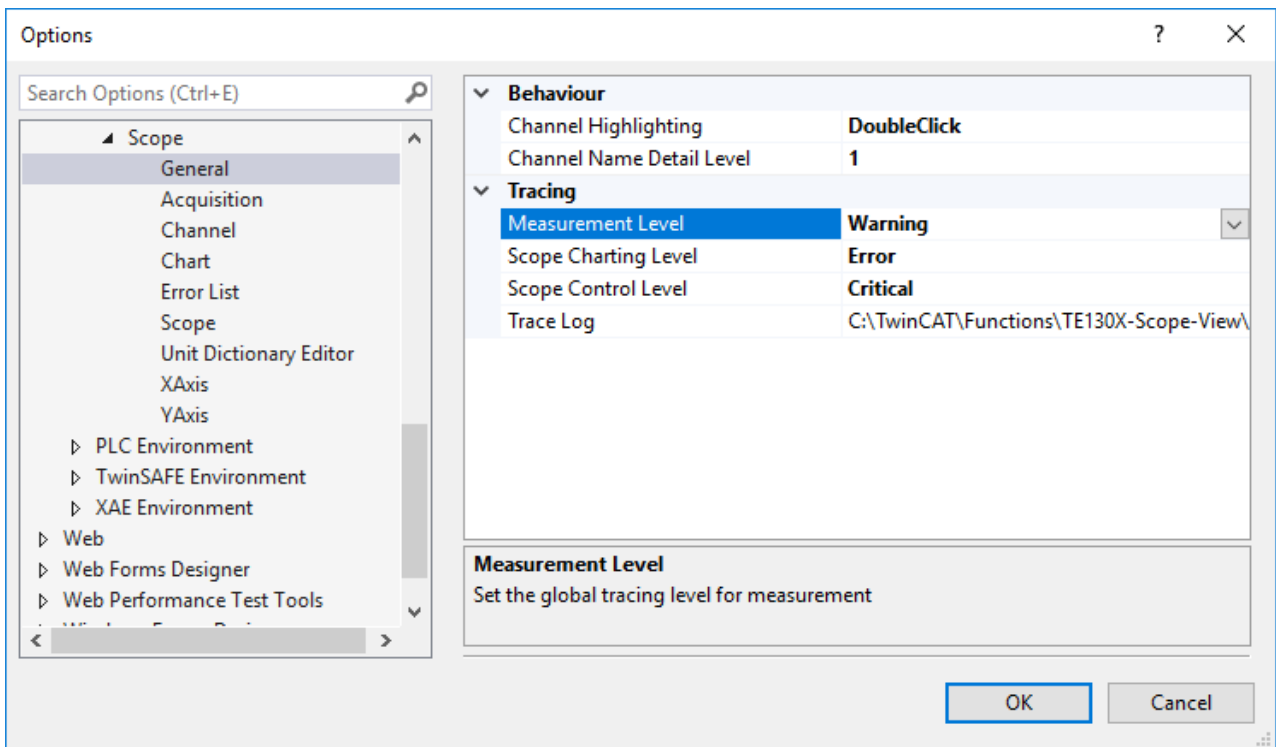
Server:

The settings for the server are to be made in the corresponding server interface. This is also described accordingly in the TF3300 documentation.
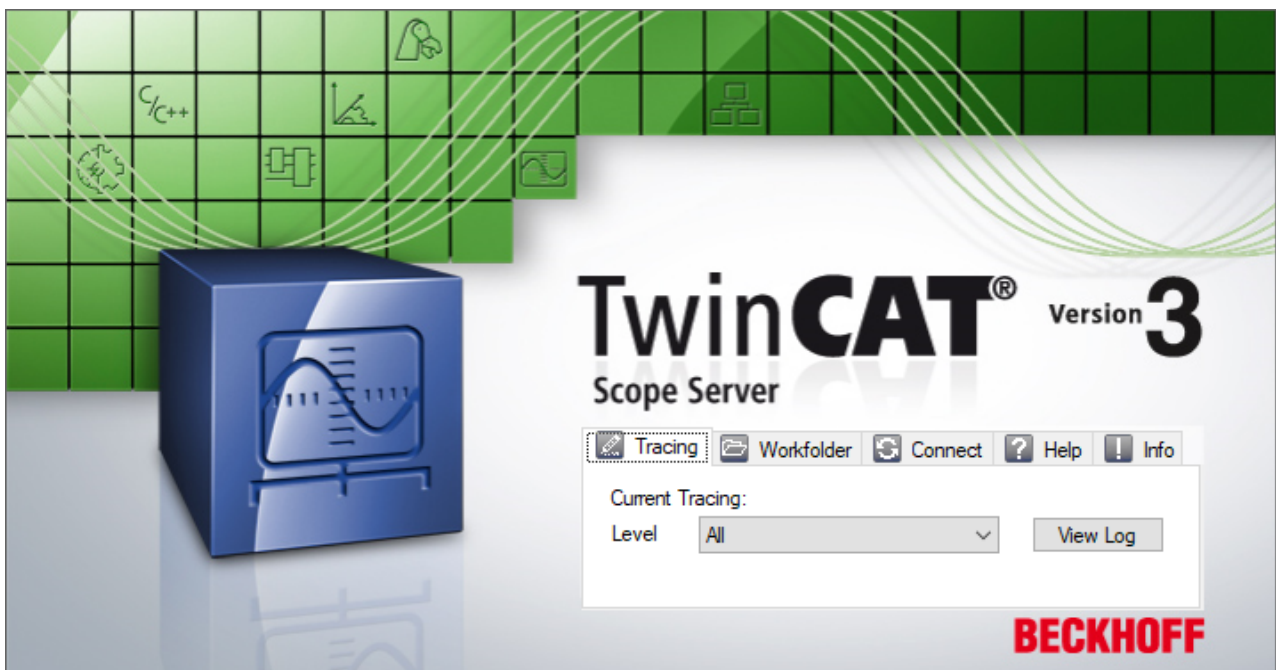
All trace logs can be sent with the Support Information Report [▶ 149].

5.4.2   Scope Statistics tool

The Scope Statistics tool is used for the statistical evaluation of Scope projects. The tool is called with the **Scope Statistics** command in the context menu of the respective Scope project.
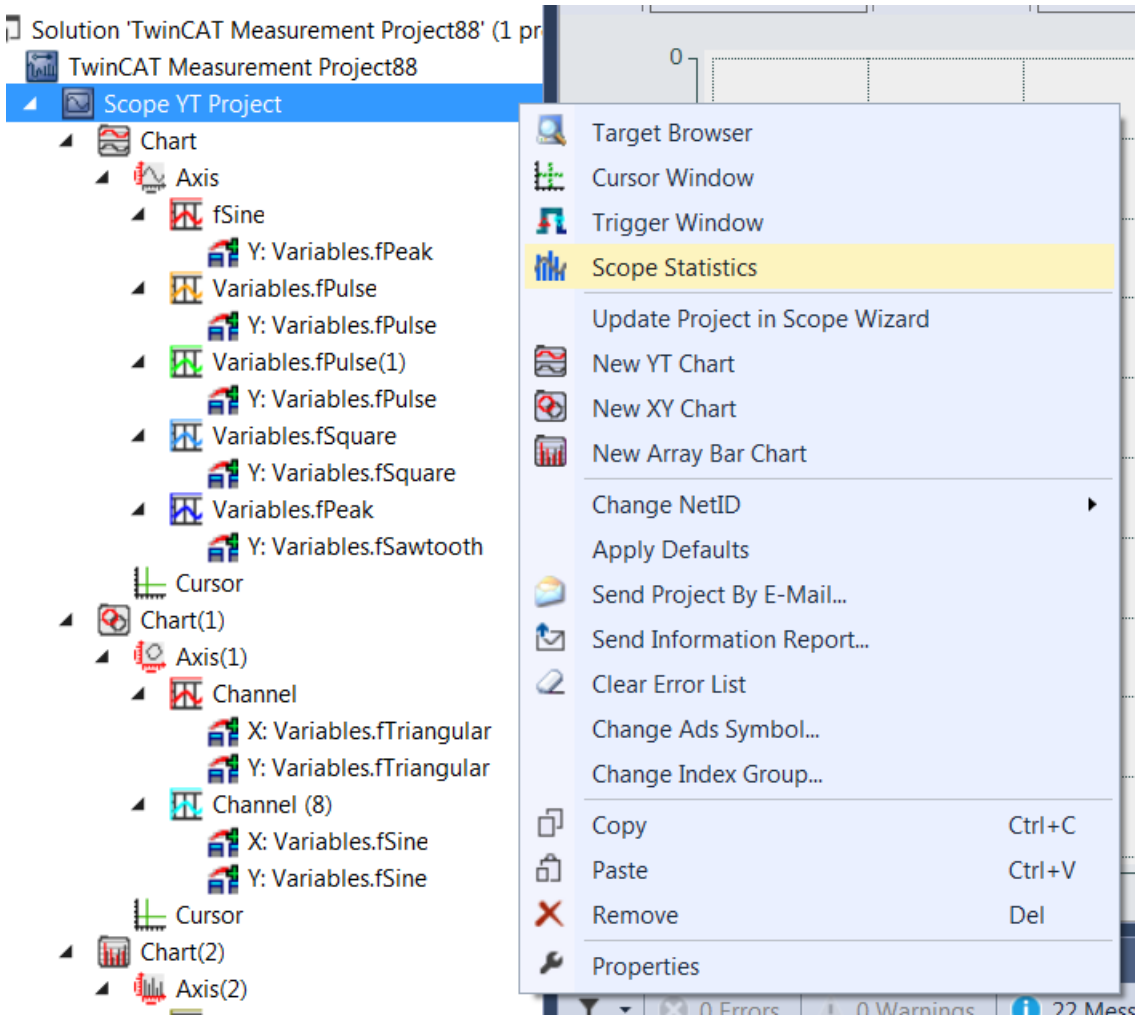
The statistics tool has seven different tabs for different topical areas.

General

The **General** tab contains general information regarding the Scope project and provides an overview of the elements contained therein.
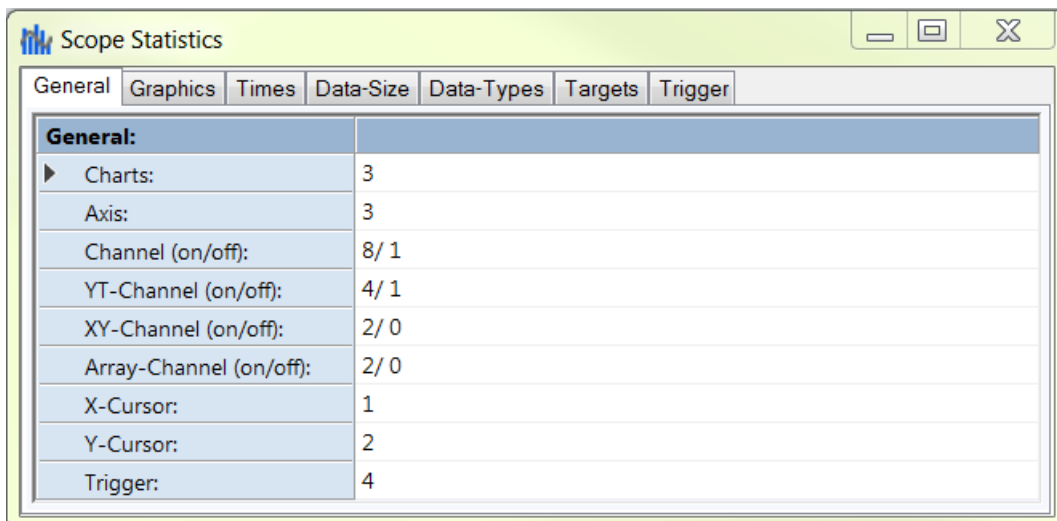
Charts	Number of charts in the project.
Axis	Number of axes in the project.
Channel (on/off)	Number of channels in the project that are enabled (on) or disabled (off) respectively.
YT-Channel (on/off)	Number of YT channels in the project that are enabled (on) or disabled (off) respectively.
XY-Channel (on/off)	Number of XY channels in the project that are enabled (on) or disabled (off) respectively.
Array-Channel (on/off)	Number of array channels in the project that are enabled (on) or disabled (off) respectively.
X-Cursor	Number of X cursors in the project.
Y-Cursor	Number of Y cursors in the project.
Trigger	Number of triggers in the project.

Graphics

The **Graphics** tab contains information on the graphic properties of the respective project, as these affect the drawing performance.
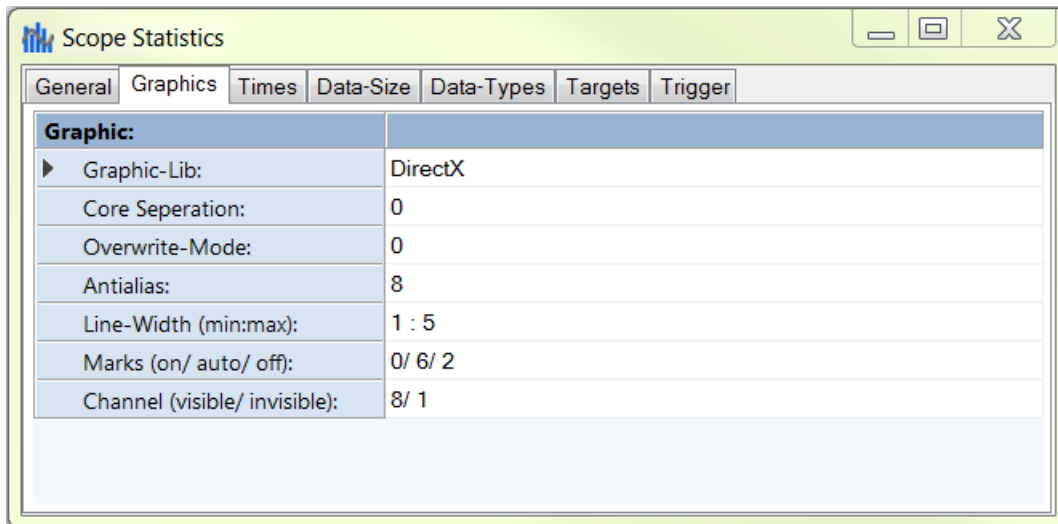
Graphic-Lib	Graphic library currently in use.
Core Separation	Number of charts that use core separation.
Overwrite-Mode	Number of charts for which the overwrite mode is activated.
Antialias	Number of charts for which the "antialiasing" function is activated.
Line-Width (min:max)	Minimum and maximum set line width of the channels.
Marks (on/ auto/ off)	Number of channels for which the marks are activated (on), set to automatic (auto) or deactivated (off) respectively.
Channel (visible/ invisible)	Number of visible or invisible channels respectively.

Times

The **Times** tab provides information on the time settings of the project.
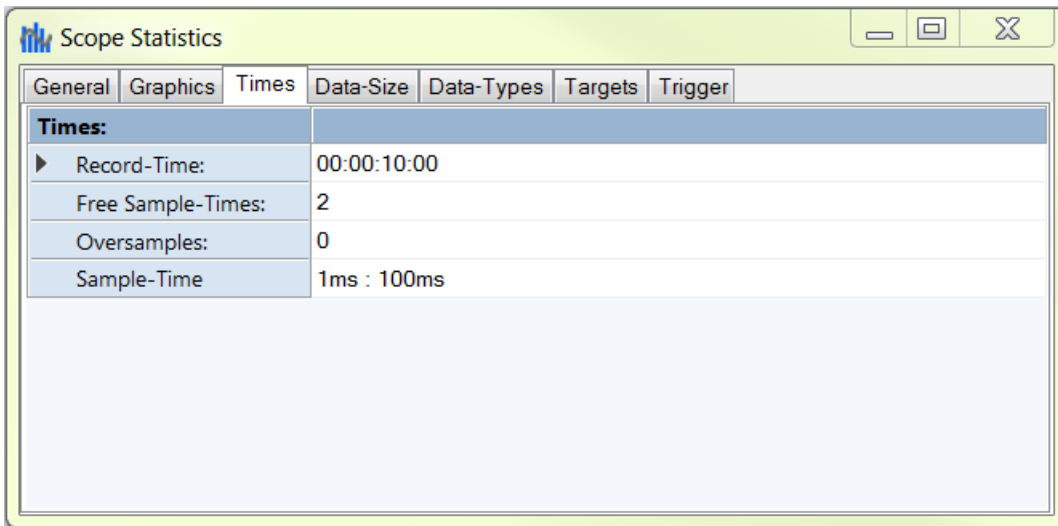
Record-Time	Set recording time of the project.
Free Sample-Times	Number of channels for which the sample state "FreeSampleTime" is set. This means that for these channels a free sampling rate has been set that is longer than the TaskSampleTime.
Oversamples	Number of channels for which oversampling is activated.
Sample-Time (min:max)	Minimum and maximum set sampling rate within the project.

Data-Size

The **Data-Size** tab contains information about the memory sizes.
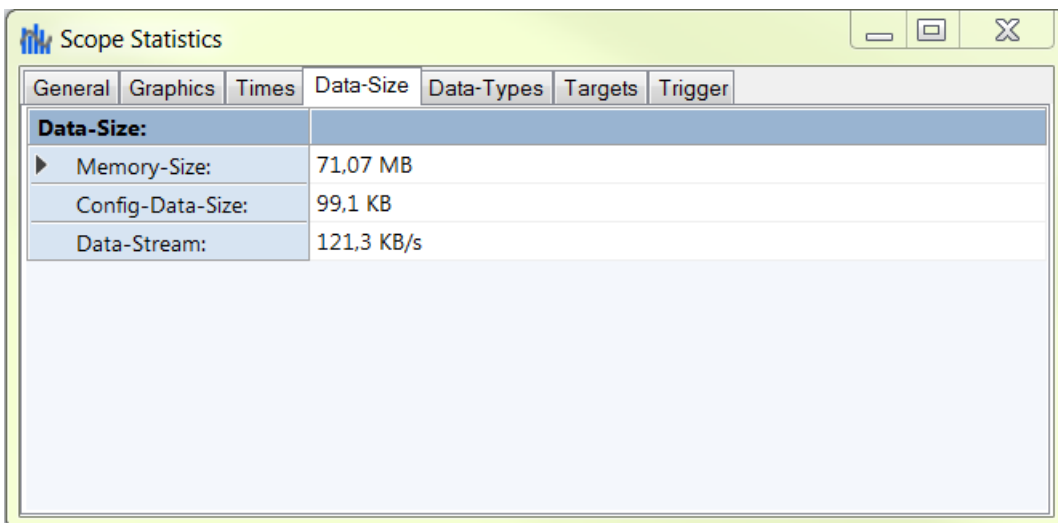
Memory-Size	File size of the complete recording.
Config-Data-Size	File size of the configuration file.
Data-Stream	Data stream during the recording.

Data-Types

The **Data-Types** tab contains information about the data types within the Scope project.
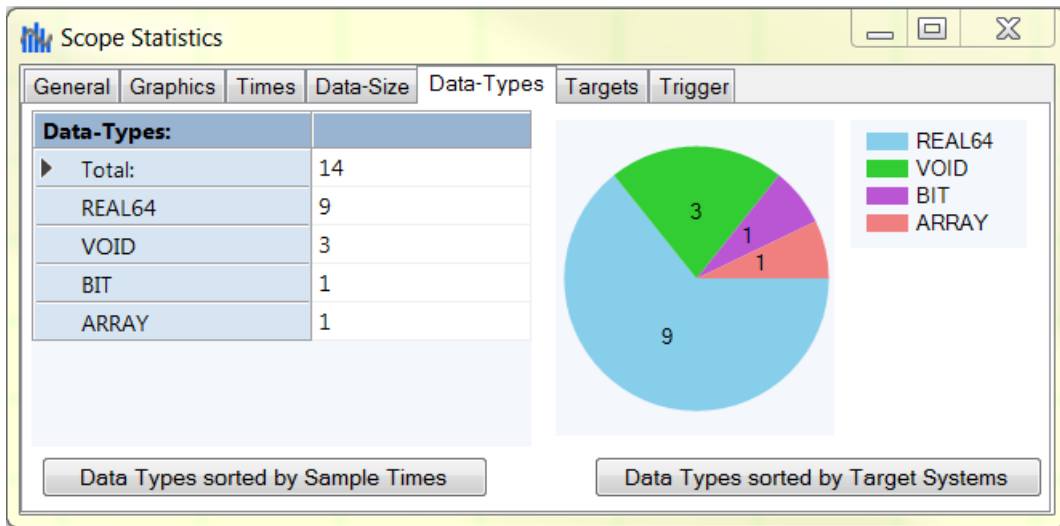
The various data types of the channel acquisition are illustrated in a table and as a pie chart. Note here that in the "ARRAY" line only the arrays of the array bar chart are listed. Other arrays, for example those used for oversampling purposes, are listed under the data types of their element.

To obtain a detailed overview, a table can be called via the **Data-Types sorted by Sample-Times** button in which the data types are listed in order of sampling rate.

Data-Types:	Total	1ms	0,5ms	90ms	5ms	100ms
▶ Total:	14	7	2	2	1	2
REAL64	9	7		2		
VOID	3		2		1	
BIT	1					1
ARRAY	1					1

The data types sorted according to target system are illustrated in a table via the **Data-Types sorted by Target-System** button.

Data-Types:	Total	172.17.214.54.1.1: 851 [0]	172.17.61.30.1.1: 851 [0]
▶ Total:	14	9	5
REAL64	9	9	
VOID	3		3
BIT	1		1
ARRAY	1		1

Targets

The **Targets** tab contains information on the target systems contained in the project.
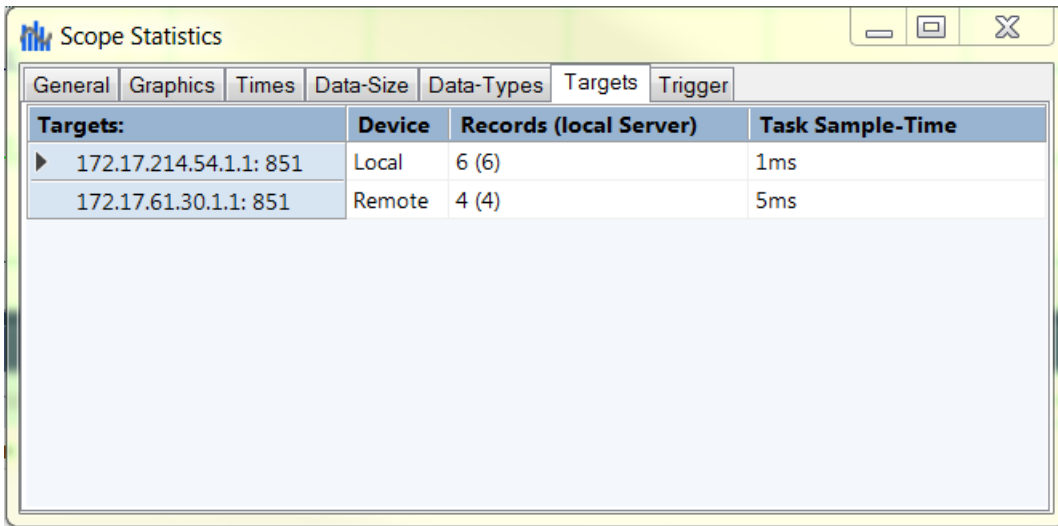
The following data are provided for each target system:

Targets	AMS-NetId and target port of the target system.
Device	Information about whether the system is a local or a remote access system.
Records (local Server)	Total number of records and the number of records on the local Scope Server.
Task Sample-Time	Task sample time for the respective target system.

Trigger

The **Trigger** tab contains information on all triggers that exist in the project.
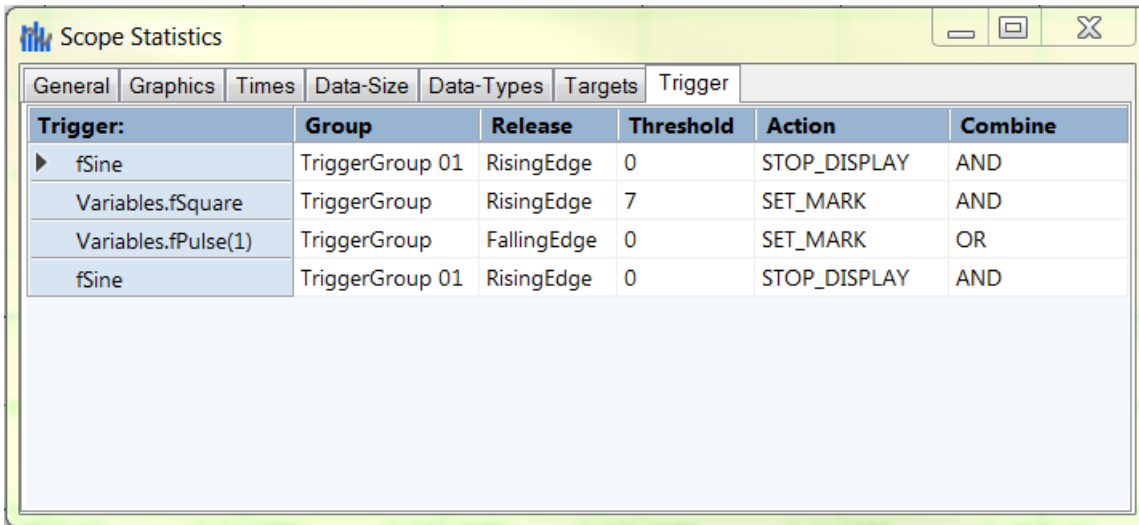
The following data are displayed for each trigger:

Trigger	Associated channel whose state is to be tested for the selected trigger conditions.
Group	Trigger group to which the respective trigger belongs.
Release	Indicates whether the trigger condition is a rising or falling edge.
Threshold	Limit value of the trigger condition.
Action	Trigger action that is executed when it is triggered.
Combine	Selected linking options to the other trigger conditions.

5.4.3 Support Information Report

The Support Information Report is a tool for collecting product information for submission to Beckhoff technical support. Collecting product-related data such as TwinCAT version/build, product version, image version and device type reduces email traffic significantly and enables more efficient advice.

Plug-in mechanism

Various Beckhoff products interface with the Support Information Report via a plug-in mechanism. These products, such as the TwinCAT Database Server, have a Support Information Report entry in the corresponding product menu.

Creating and submitting a Support Information Report

✓ A Support Information Report is open.

1. Use the **Behaviour** text field to describe the behavior that occurred in as much detail as possible.

2. In the **Attachment** area, you can add files (screenshots etc.) to the report via the **Add Attachment** button, if required. Files can optionally be selected via remote access. To do this, select a target from the **Remote System** dropdown list. Depending on the selected target, it may be possible to browse Windows CE devices.

3. Enter your contact details and select a Beckhoff subsidiary for your country.
   This information is obligatory for submitting the Support Information Report.

4. You will be offered the option to store your contact details for future Support Information Reports. To do this, tick the **Store personal data** check box.

5. The product-specific plug-ins can be found in the lower section of the Support Information Report. Tick the **Include in report** check box. The information required for the product is added automatically, if it is available. The screenshot shows the current configuration of a TwinCAT Database Server in the form of an XML file as an example.

6. Submitting the Support Information Report:

   • If the device has an email connection, you can submit the Support Information Report directly to the Beckhoff subsidiary for your country via the **Send Report** button.

   • If the device does not have an email connection, you can save the Support Information Report locally as a .zip file via the **Save .zip** button and then make it available via FTP, USB etc.

6 .NET API

If you wish use the Scope View Control in your own .NET-based HMI, the TE1300 setup must be carried out on the target system. Via the component installation you can select whether the View or even just the binaries are to be installed. Moreover, in this case the Engineering license for TE1300 has to be activated on the target system.

6.1 API Documentation

The attached class library offers an extensive source of information for developers who work with the technologies and services of TwinCAT Scope. Amongst other things, the library contains instructions and reference documents as well as example codes.

In order to find the desired contents, navigate through the table of contents, use the search function or switch to one of the sections of the library with the aid of a quick link.

For example codes, switch in the left-hand area of the user interface to the **Search** tab and search for the keyword "examples".

In the search results you will find example codes for the trigger modules, cursor and many more.

See also:

Class library documentation

6.2 Integration in a WPF (Windows Presentation Foundation) project

In order to use the ScopeViewControl within WPF applications as well, you require a WindowsFormsHost as a the basis, since the ScopeViewControl is a Windows Forms UserControl.

1. Position a WindowsFormsHost control from the toolbar in the user interface of the WPF application you created.
2. Initialize the ScopeViewControl as described in the Windows Forms examples, and add the created ScopeViewControl to the WindowsFormsHost. When using the ScopeViewControl within the WPF application, only the "ScopeViewControl.Operating.SupportWpfRefresh" property needs to be set.

A sample implementation is shown below:

```
namespace WpfApplication
{
 ///
 /// Interaction logic for MainWindow.xaml
 ///
 public partial class MainWindow : Window
 {
    public MainWindow()
    {
     InitializeComponent();
     ScopeViewControl scopeViewControl = new ScopeViewControl();
     scopeViewControl.Operating.SupportWpfRefresh = true;
     // Initialisierung des ScopeViewControl
     windowsFormsHost1.Child = scopeViewControl;
    }
 }
}
```

7 Samples

7.1 TwinCAT 3 Scope View – first steps

In order to allow a fast startup and to simplify work with the TwinCAT 3 Scope software oscilloscope, information is summarized and the first steps briefly described on this page.

1. Installation

A Scope Server is always installed with a Scope View, since otherwise no saved data can be displayed locally in the View. In contrast, the Scope component installation can install the Scope server without Scope View on a target device.

2. Licensing

Regardless of whether you have installed the Scope View via its own setup or via the TwinCAT 3 XAE setup, the "base" license is initially activated for Scope View and Server. Please refer to the Overview [▶ 7] to see which functions can be enabled with which license key.

3. Creating a new project

You can select the Scope View under the **Beckhoff** entry in the Windows start menu.
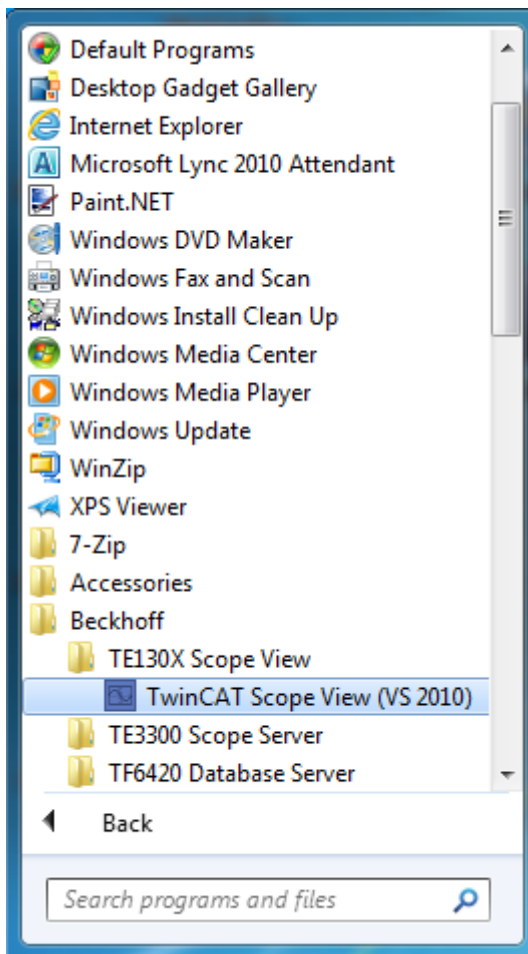
The Visual Studio opens in its shell or in a full version if installed. You can choose between various templates.
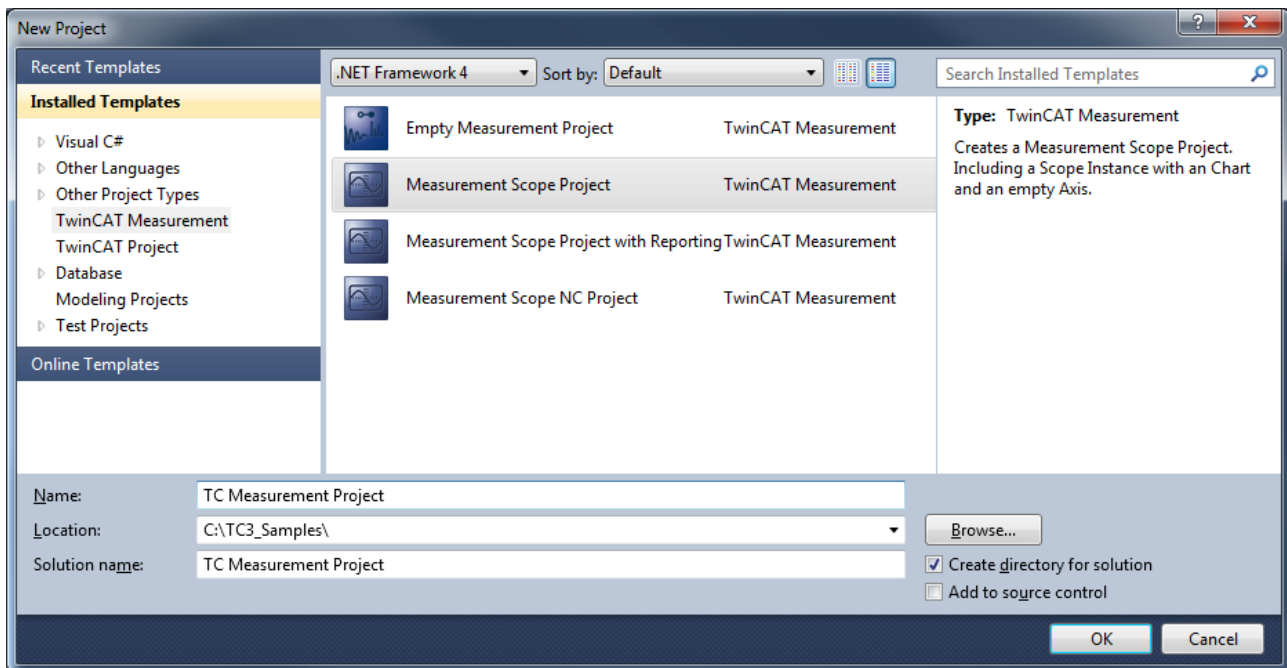
In the **Solution Explorer** the starting point for the new Scope configuration assembles itself automatically.
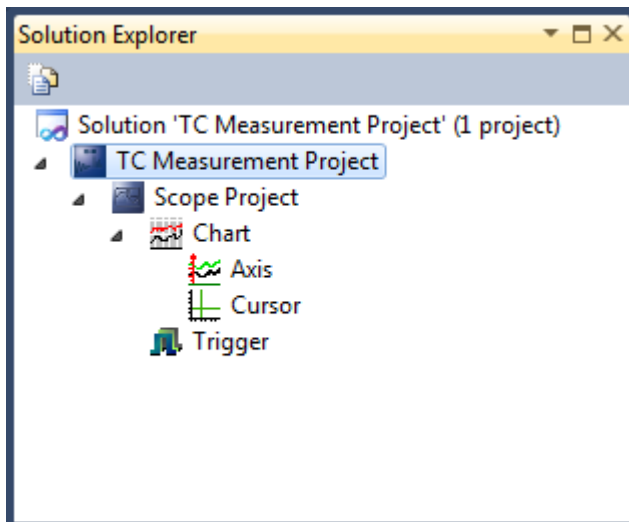
4. Selection of channels

You add variables or channels to the generated configuration. You can select them via the **Target Browser**. When using for the first time it is possible that the **Target Browser** will not be directly visible as a window in Visual Studio. In this case you can open it via the **Target Browser** command in the context menu of the Scope object or in the **Scope** menu. You can dock the **Target Browser** in any place in the interface.
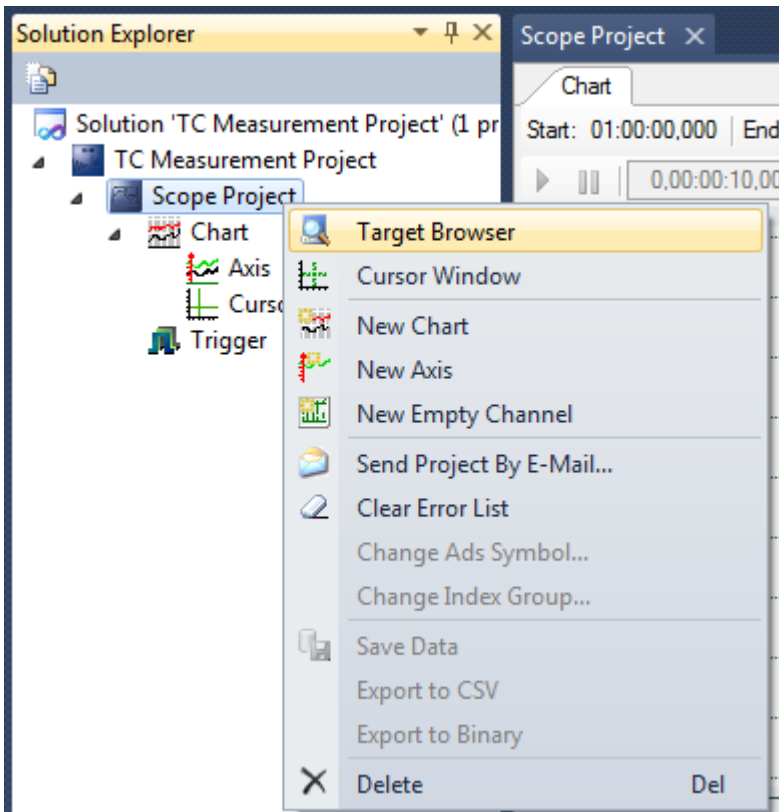
Select the respective target system in order to display the desired variables.
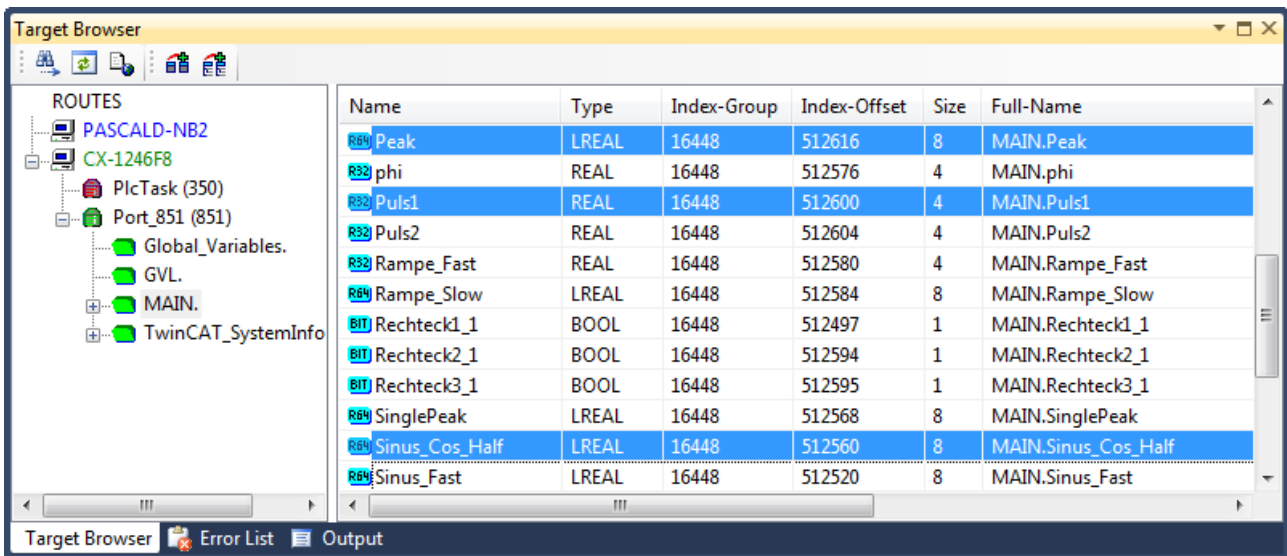
You can select the variables via multiple selection and add them to the configuration via the context menu command **Add Symbol** or by drag & drop.
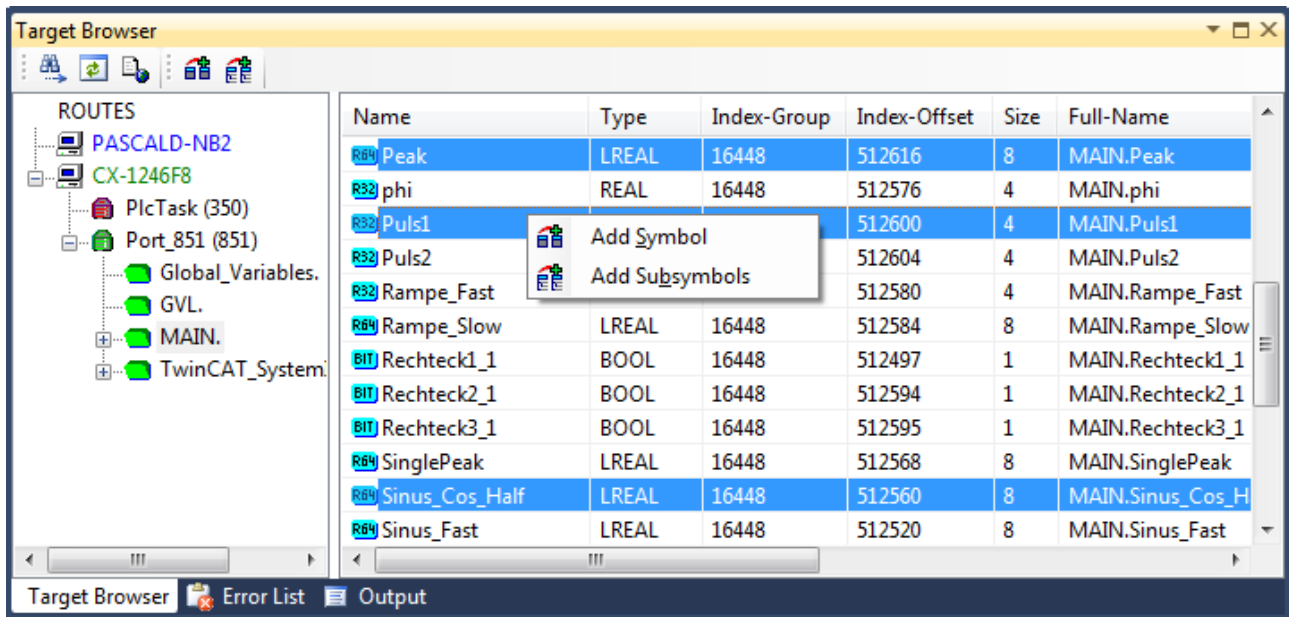
The selected channels are assigned to the last-selected axis. In this case only one axis is present.
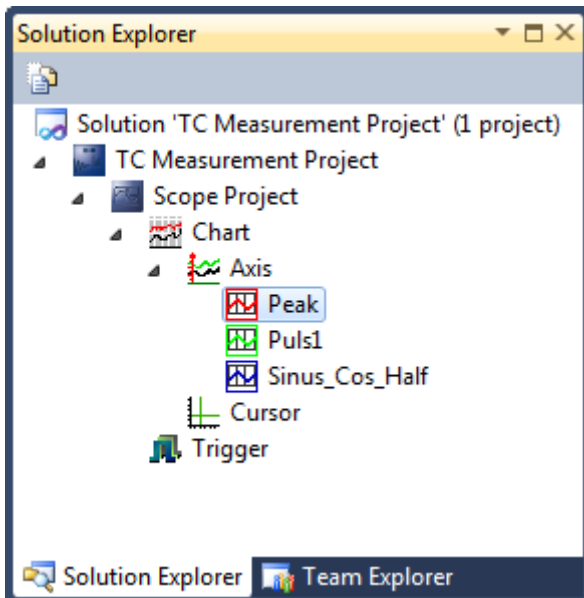
5. Defining the settings

Before making a recording, you have to define the basic settings for the recording in the Scope properties. Standard is a ten-minute recording which is started manually and automatically stopped after the respective time. The settings for all hierarchical levels – Scope, Chart, Axis and Channel – as well as for trigger and cursor can be made in the standard **Properties** window. The corresponding element must be selected in the **Solution Explorer**. If the **Properties** window is not yet open, you can open it via the **Properties** command in the **View** menu.
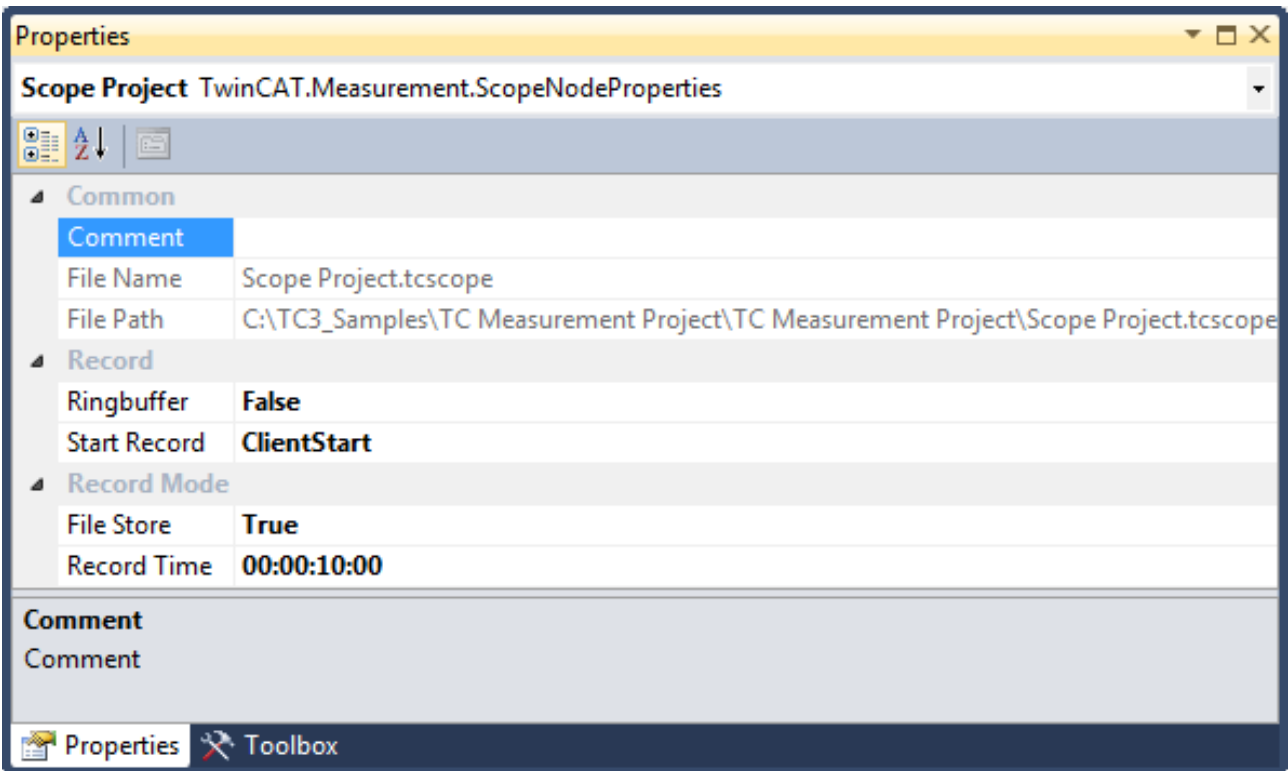
If the default settings are not changed, the recording can be started. When using for the first time you must activate the TwinCAT Measurement toolbar in the menu **View > Toolbars**.
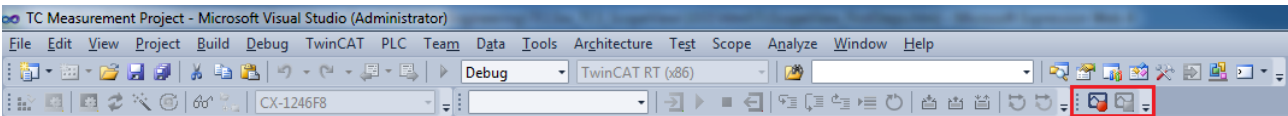
The recording is started by clicking on the icon with the red recording symbol.
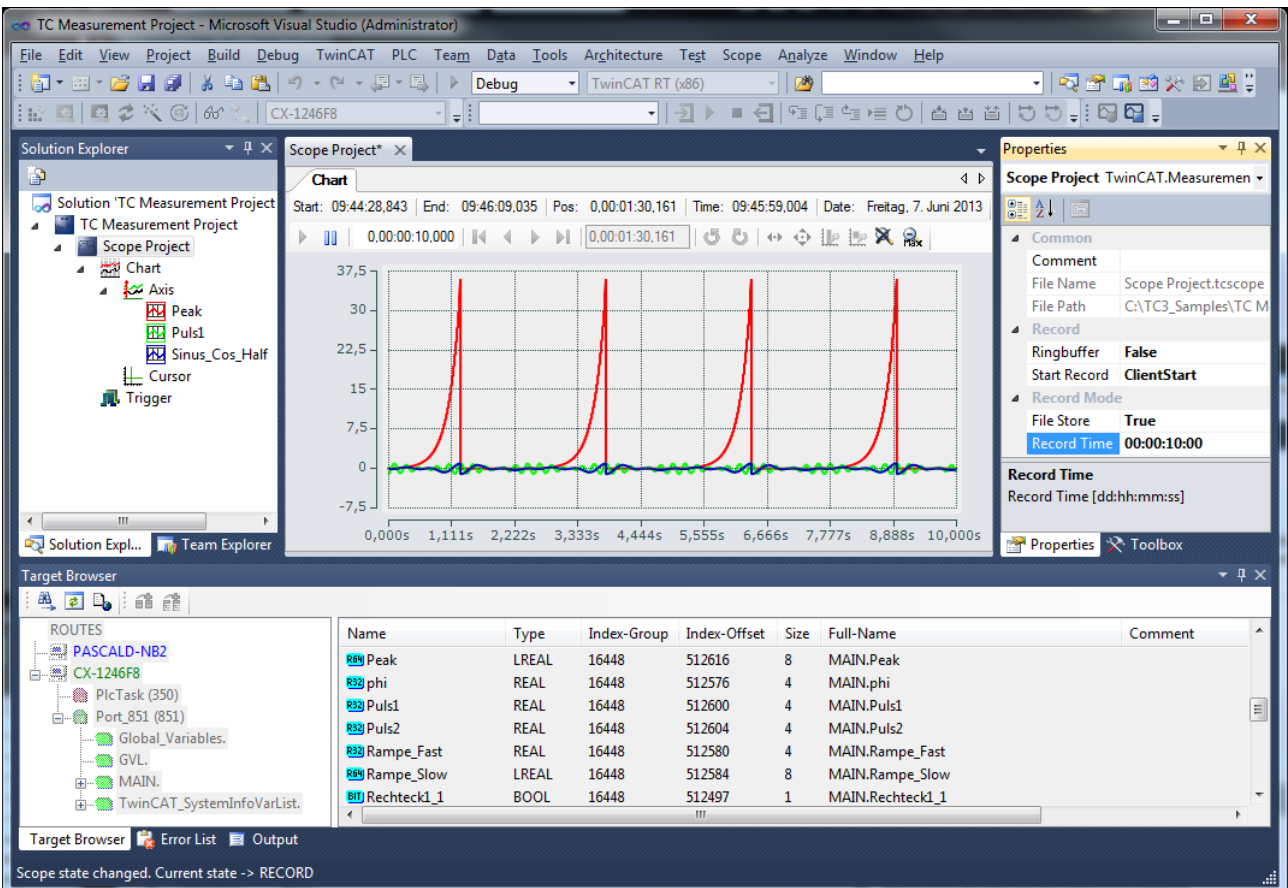
6. Changing the display

In order to improve the display of the channels in the View, charts and axes can be added even during the recording in TwinCAT 3 Scope View and channels can be shifted within the Scope.
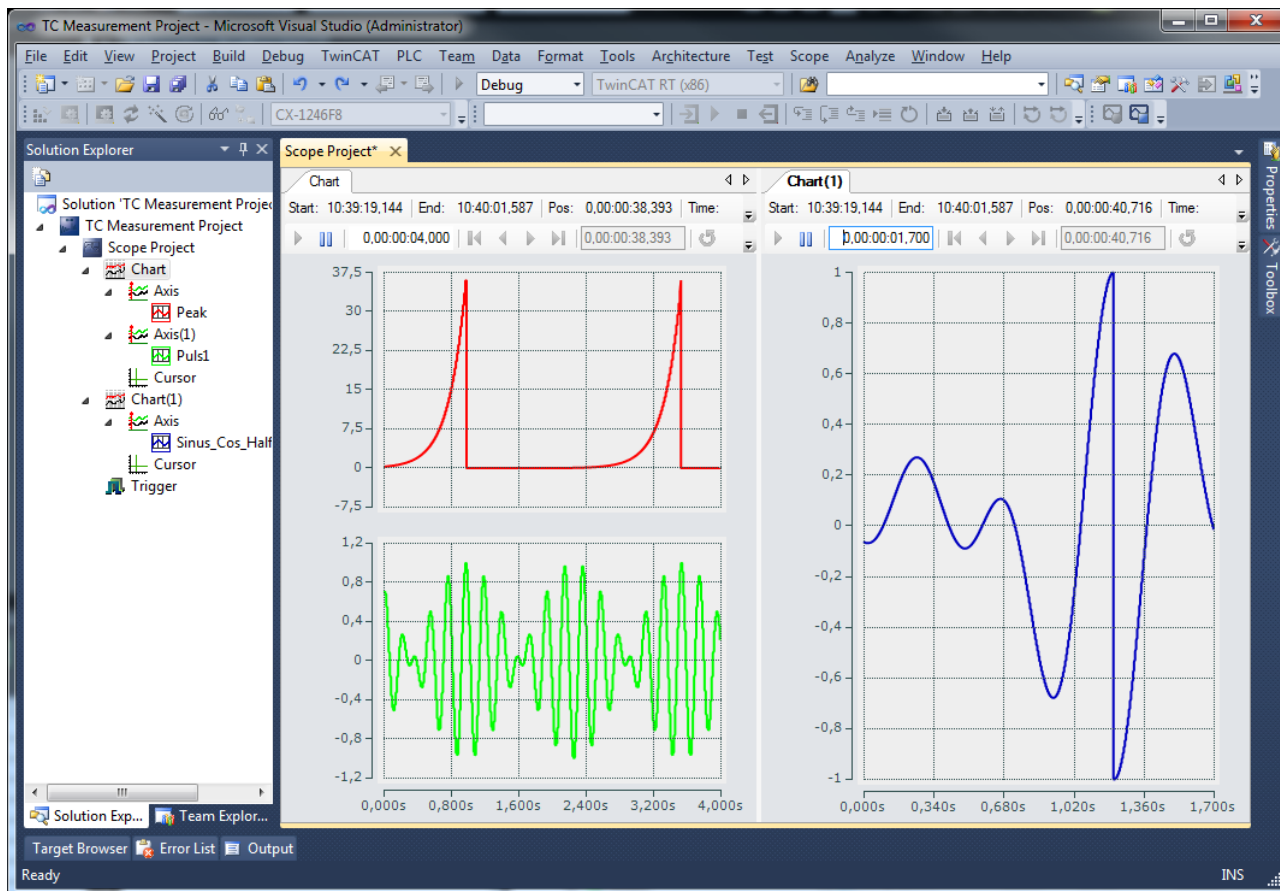
X and Y cursors can also be added and deleted at runtime.
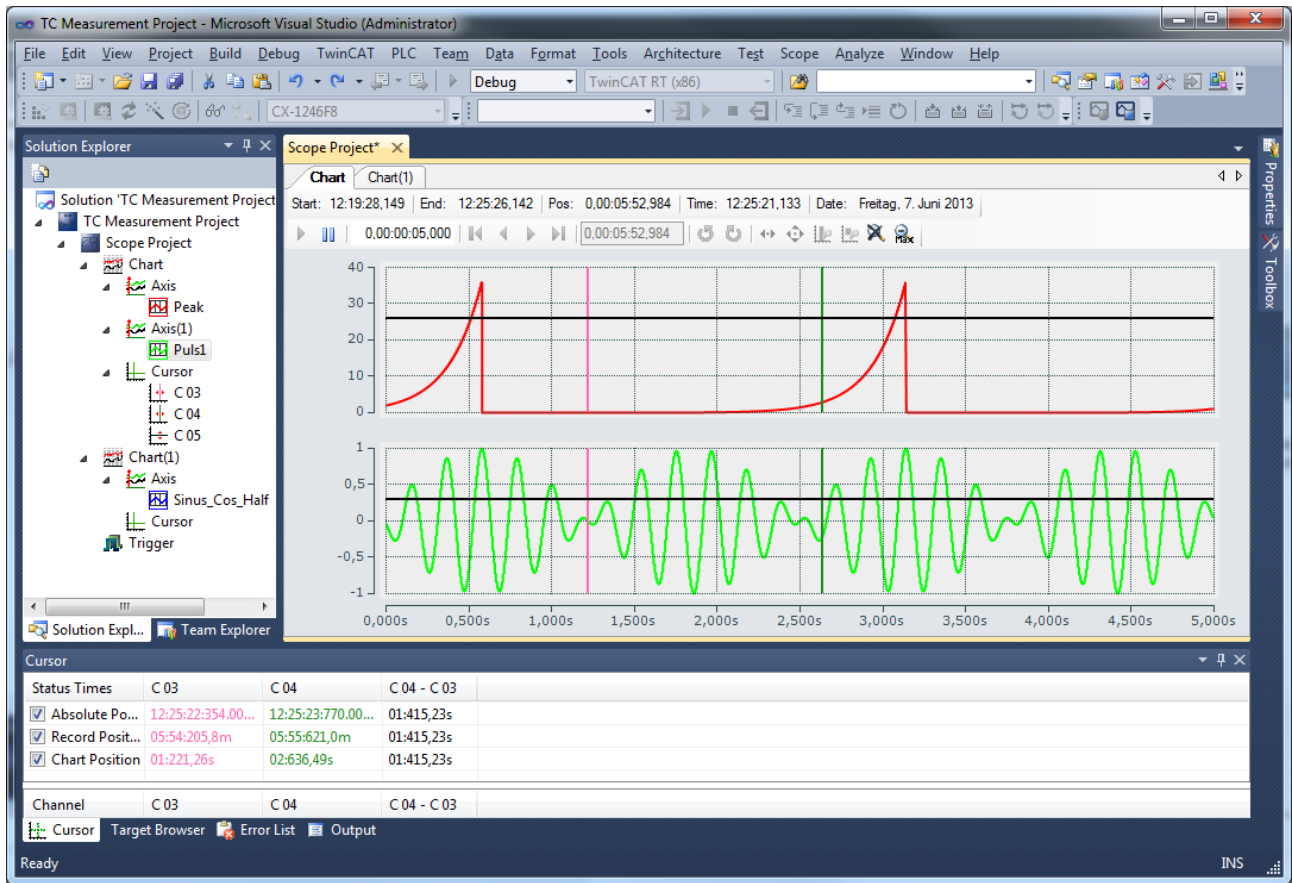
You can freeze the display of the current recording while the recording of the data continues to run in the background. Once the display has stopped you can zoom into the data and analyze the signal curve with the cursors.
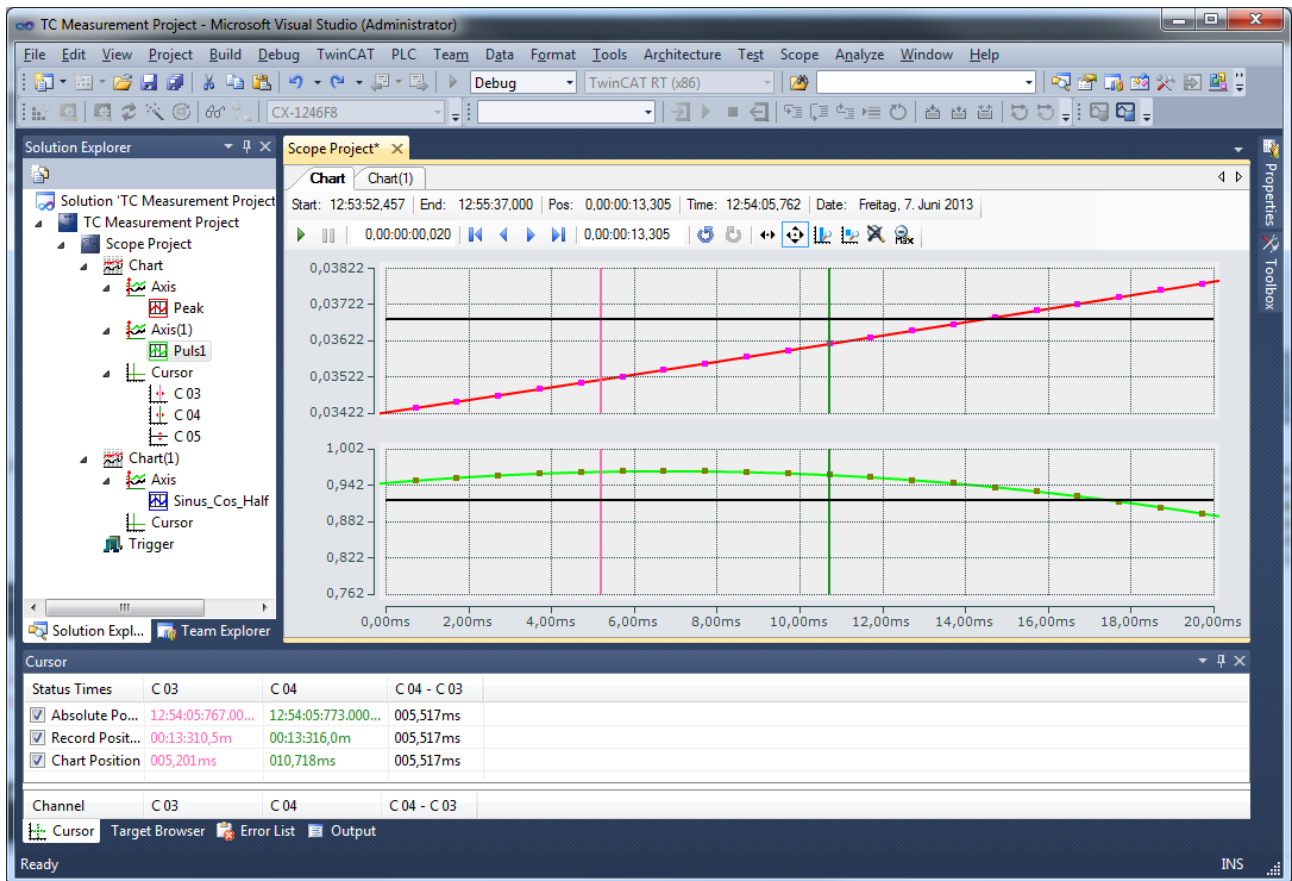
7. Stopping the recording and saving the data

In this case the started recording is stopped automatically on expiry of the recording time of ten minutes. Alternatively, you can end the recording at any time via the **Stop** button in the TwinCAT Measurement toolbar. The recorded data are only consistent if you save them as an .svd file via the **Save Data** command in the **Scope** menu.
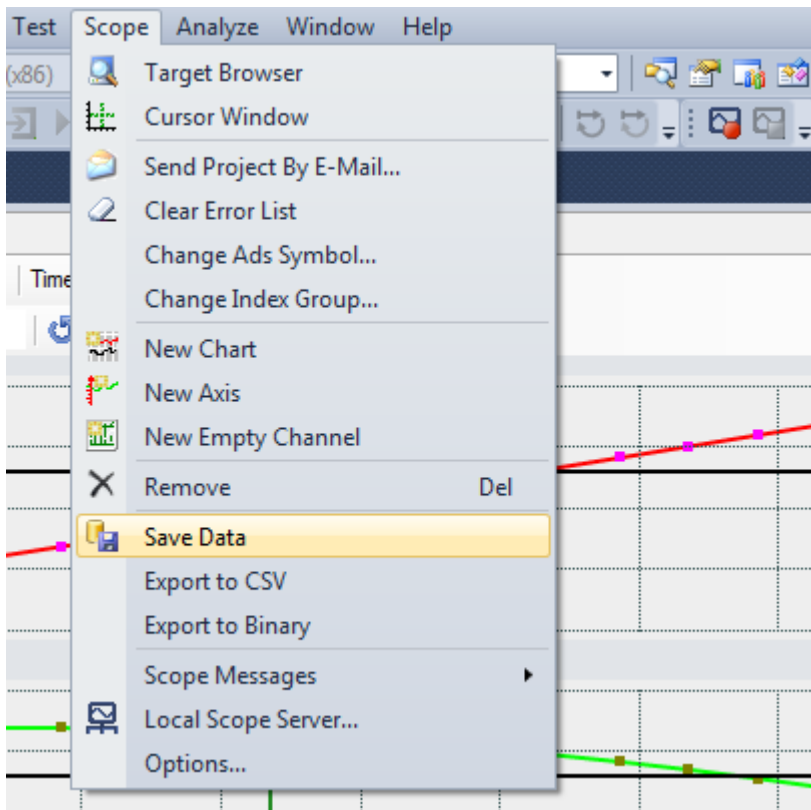
8 Appendix

8.1 How To

8.1.1 Changing common settings at the same time

Multiple selection – changing group-wise

To change the settings of several elements at the same time, mark the elements in the **Solution Explorer** and edit the settings of all the marked elements in the **Properties** window. Data that differ between the selected elements are represented as empty fields.

To make multiple selections, keep the [Ctrl] key pressed while selecting the channels. In the case of successive elements, you can select the first element, keep the [Shift] key pressed and then select the last element in order to mark all elements in between.

To select all channels of a scope irrespective of the configuration structure, keep the [Alt] key pressed and double-click on the first channel in the Scope configuration.

All elements of the same type can be selected.

Multiple selection is implemented for:

- Scopes
- Charts
- Axes
- Channels
- Cursor
- Trigger Groups
- Trigger Sets

## 8.1.2 Oversampling recordings with the TwinCAT 3 Scope

With the TwinCAT 3 Scope it is possible to represent oversampling values in a single variable. Since n values (n = oversampling factor) are recorded for each cycle when oversampling, the TwinCAT System Manager generates an ADS symbol containing a time stamp for each individual value. If a connection is made between the ADS symbol and the Scope, the Scope will accept all other settings to display the n values in the correct sequence.

So that the ADS symbol is created and can be directly scoped by an EtherCAT terminal (e.g. EL3702 or EL3632), some settings need to be made in the TwinCAT System Manager configuration:

In the EtherCAT process image the **ADS** tab of the ADS servers must be enabled, and the option **Create symbols** must be switched on. The assigned ADS port has to be specified later in the Scope in order to find the ADS symbols.
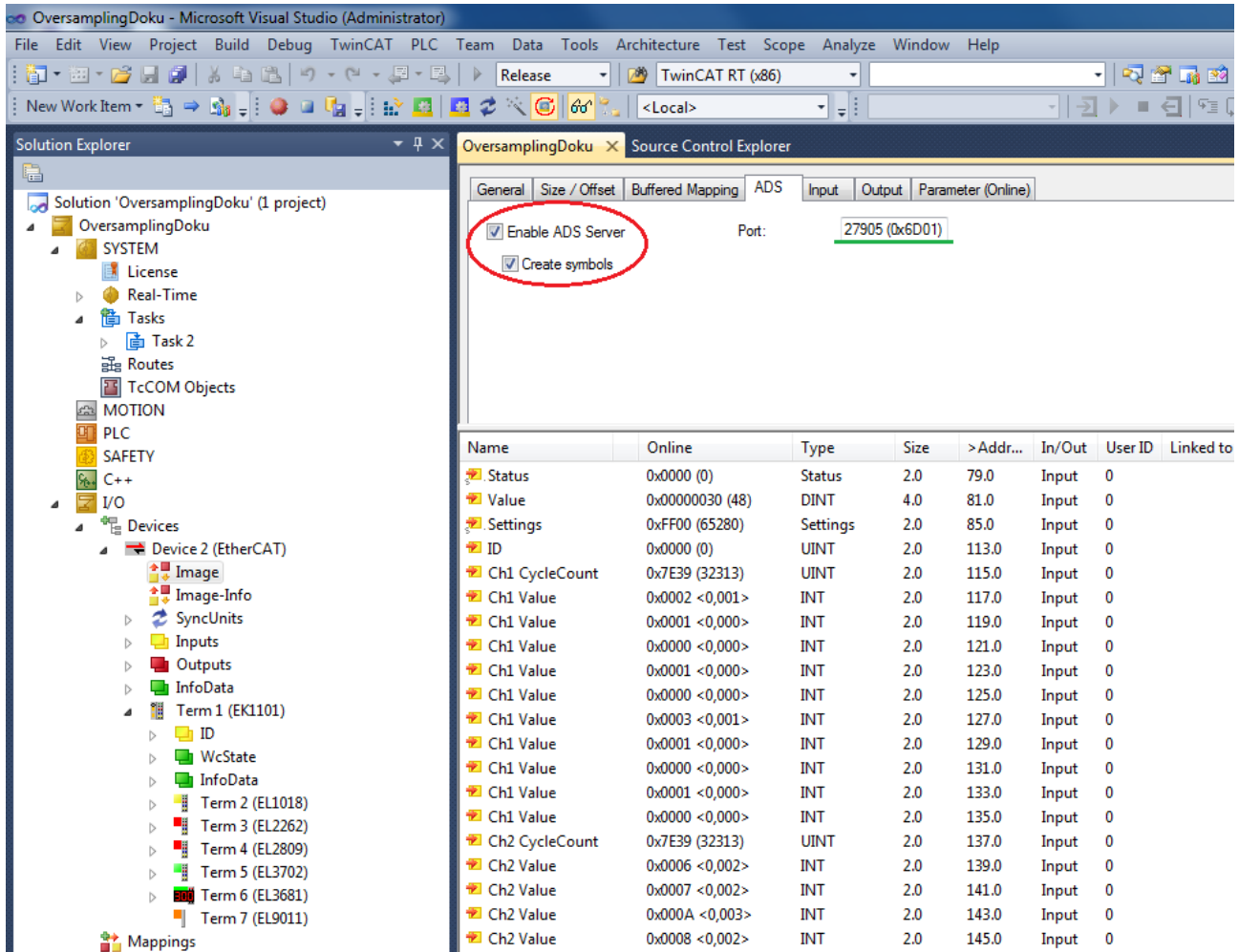

The desired oversampling factor should be set in the TwinCAT System Manager for the respective oversampling terminal before the configuration is activated.
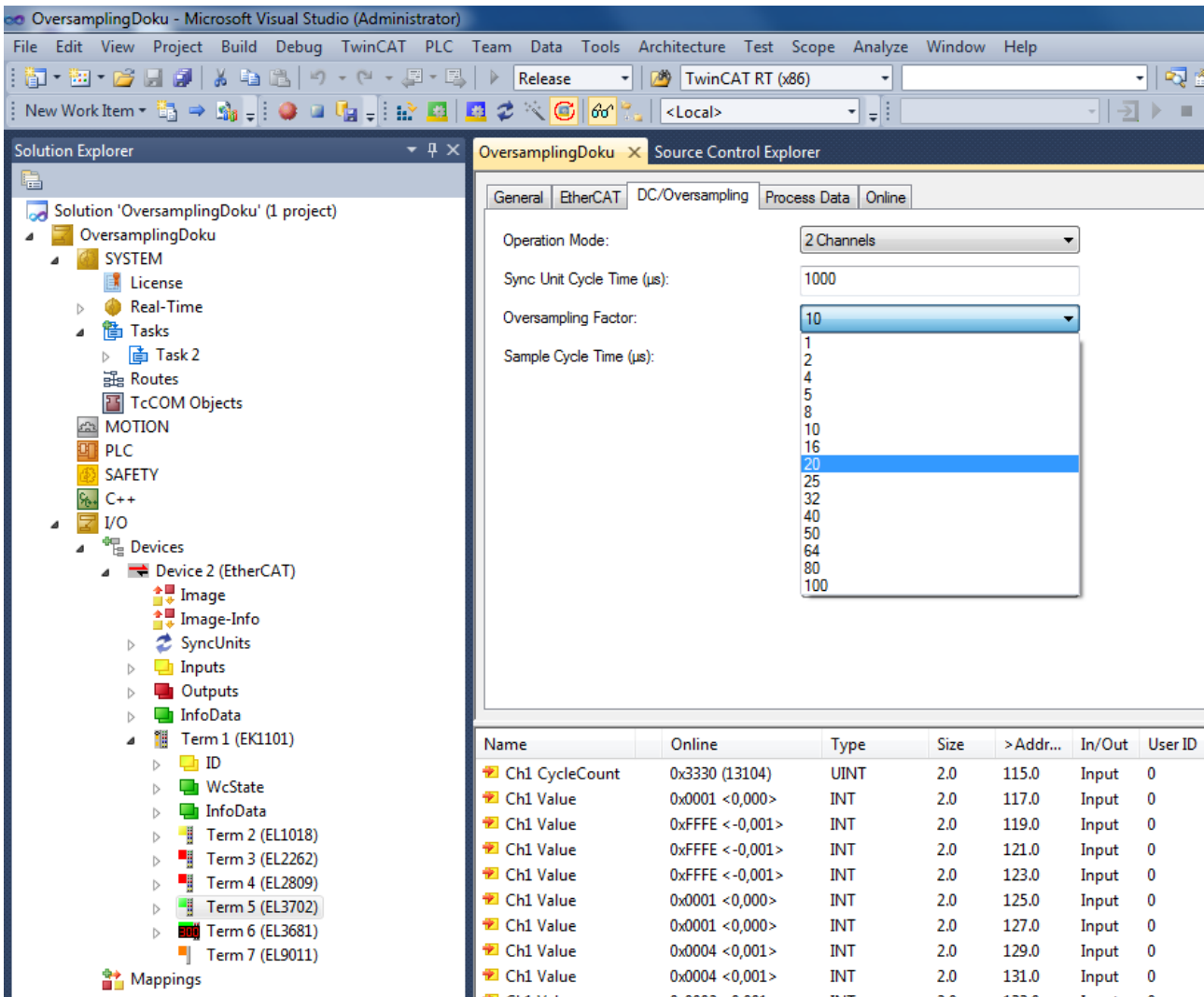
If the configuration was loaded successfully and TwinCAT is in Run mode, the TwinCAT 3 Scope can be called. The server settings must be defined in the Target Browser of the Scope. The ADS port number assigned by the TwinCAT System Manager must be entered in the corresponding dialog and added with the Add, provided the port has not already been entered.
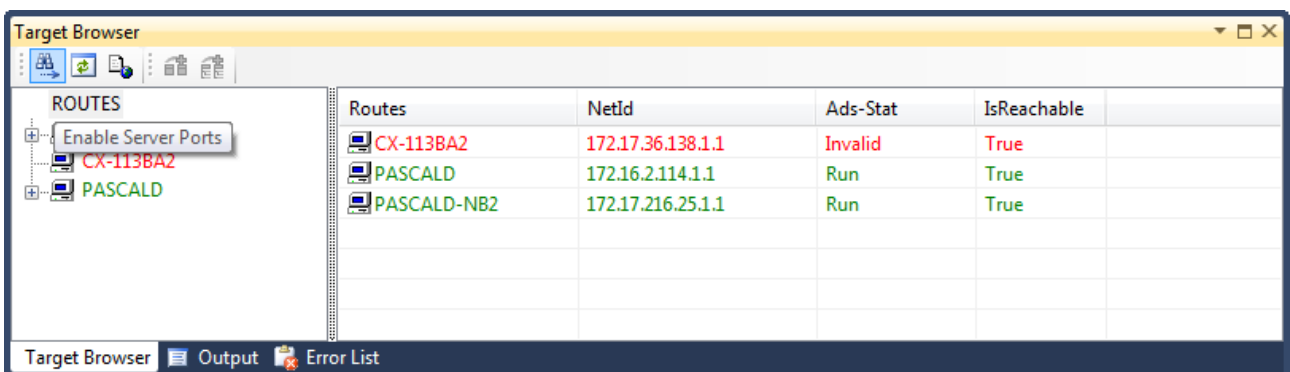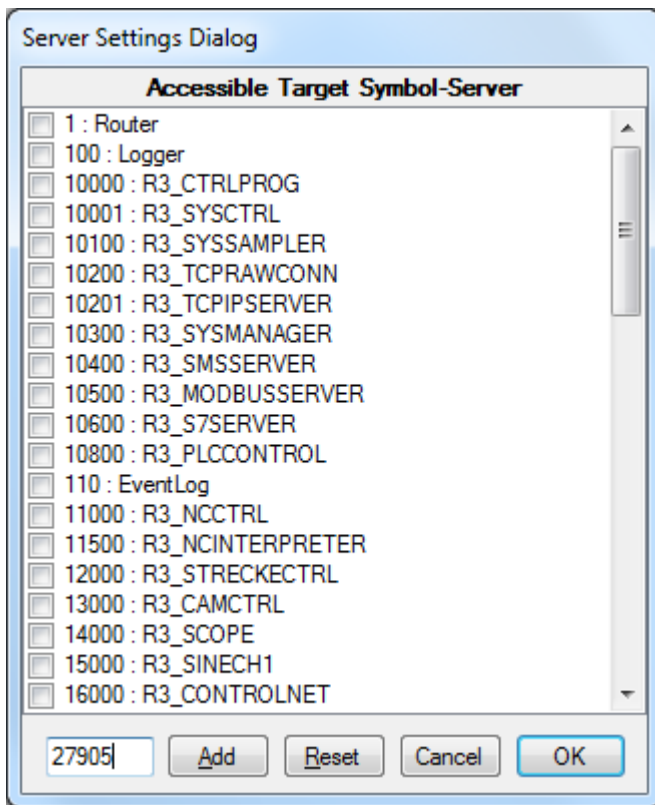
*AdsPort of Image X (27905)* appears in the Scope Target Browser. With this new entry, it is possible to browse the EtherCAT Terminals including the desired oversampling terminal. In the oversampling terminal, the variable shown in red with the supplement [T20] should be selected for the Scope recording. In this case 20 stands for 20 times oversampling. This number may vary according to the oversampling factor.
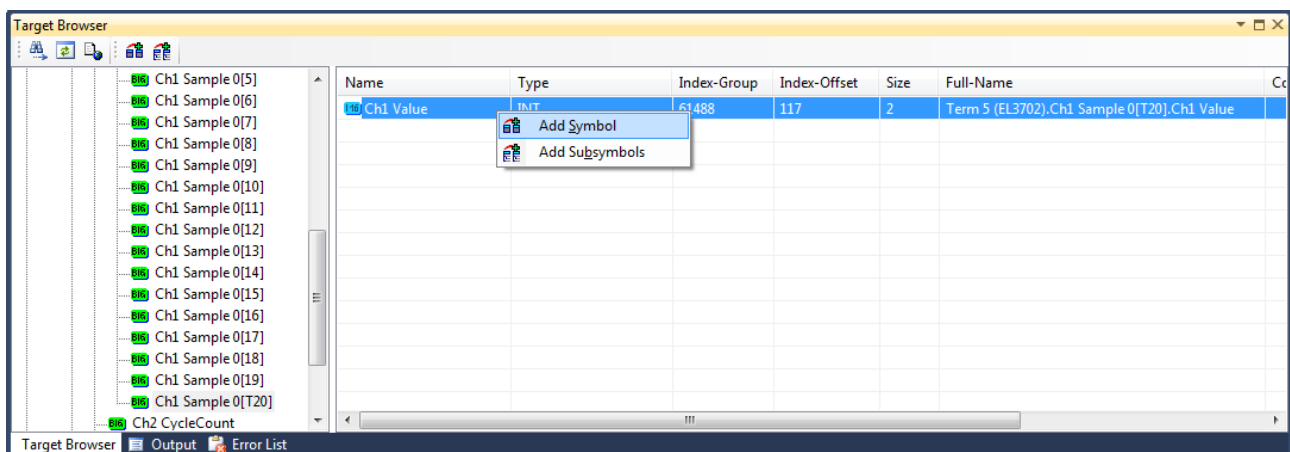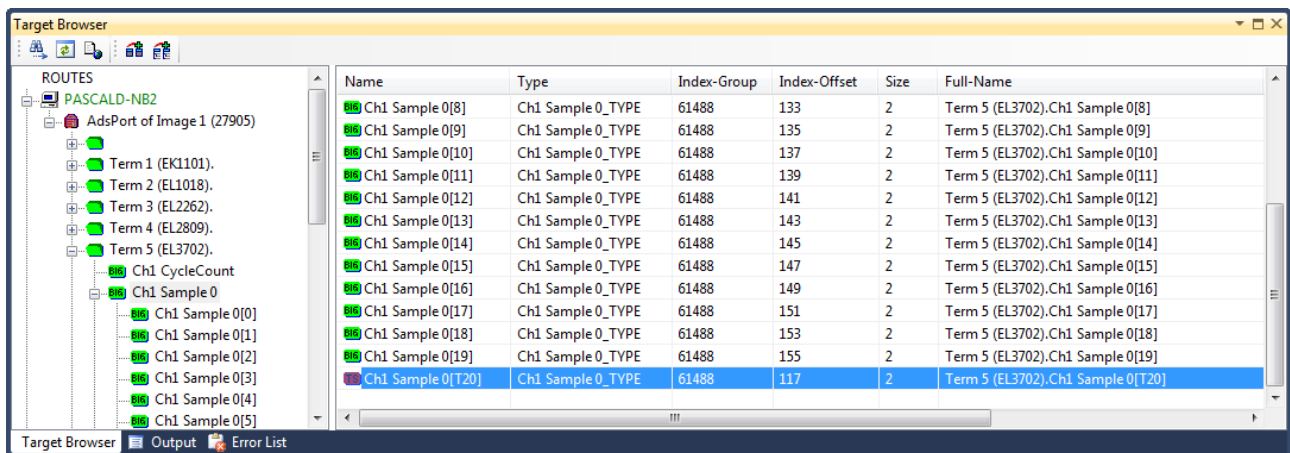
The INT16 variable can be recorded directly in TwinCAT Scope. In the following screenshot 20-fold oversampling with a cycle time of 1 ms.
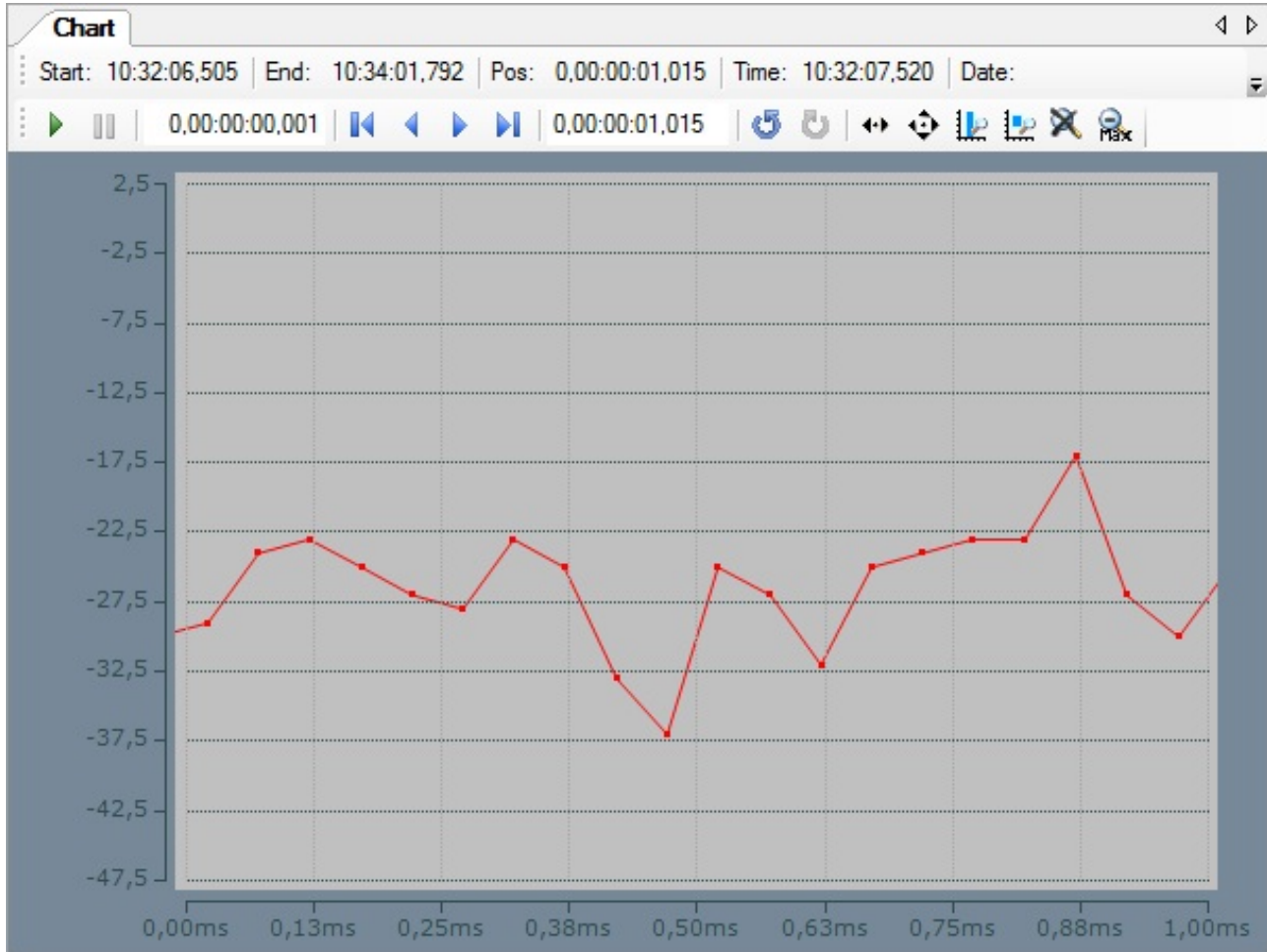
8.1.3   Opening .svd files from a network directory

If a .svd file from the network is to be merged into the solution, the following error message will probably appear if appropriate preparations are not made.

An unknown Error occured. Error handling .svd file within scope server. Enable and check scope server tracing.

One of the following entries usually appears in the Server Tracing log:

- "ScopeServerTraceSource",Error,8442,"Access to the path '\\remote-PC\SVDs\test.svd' is denied.",,368,,"16","2014-04-28T07:26:59.9720667Z",,

or

- "ScopeServerTraceSource",Error,8442," Could not find file '\\remote-PC\SVDs\test.svd '.",,3376,,"16","2014-04-28T07:26:59.9720667Z",,

Both errors indicate that the TwinCAT Scope server service has no rights of access to the network directory.

Proceed as follows to assign the required rights to the TwinCAT Scope server (example for Windows 7)**:**

1. Right-click on the directory to be enabled, in order to open the context menu. Select the command **Properties**, and in the dialog that opens select the command **Share** in the **Sharing** tab.

⇨ The dialog **File Sharing** opens**,**

2. Enter the users who are to gain access to the directory. Permission level "Read" is sufficient for Scope server access and for displaying the .svd file.
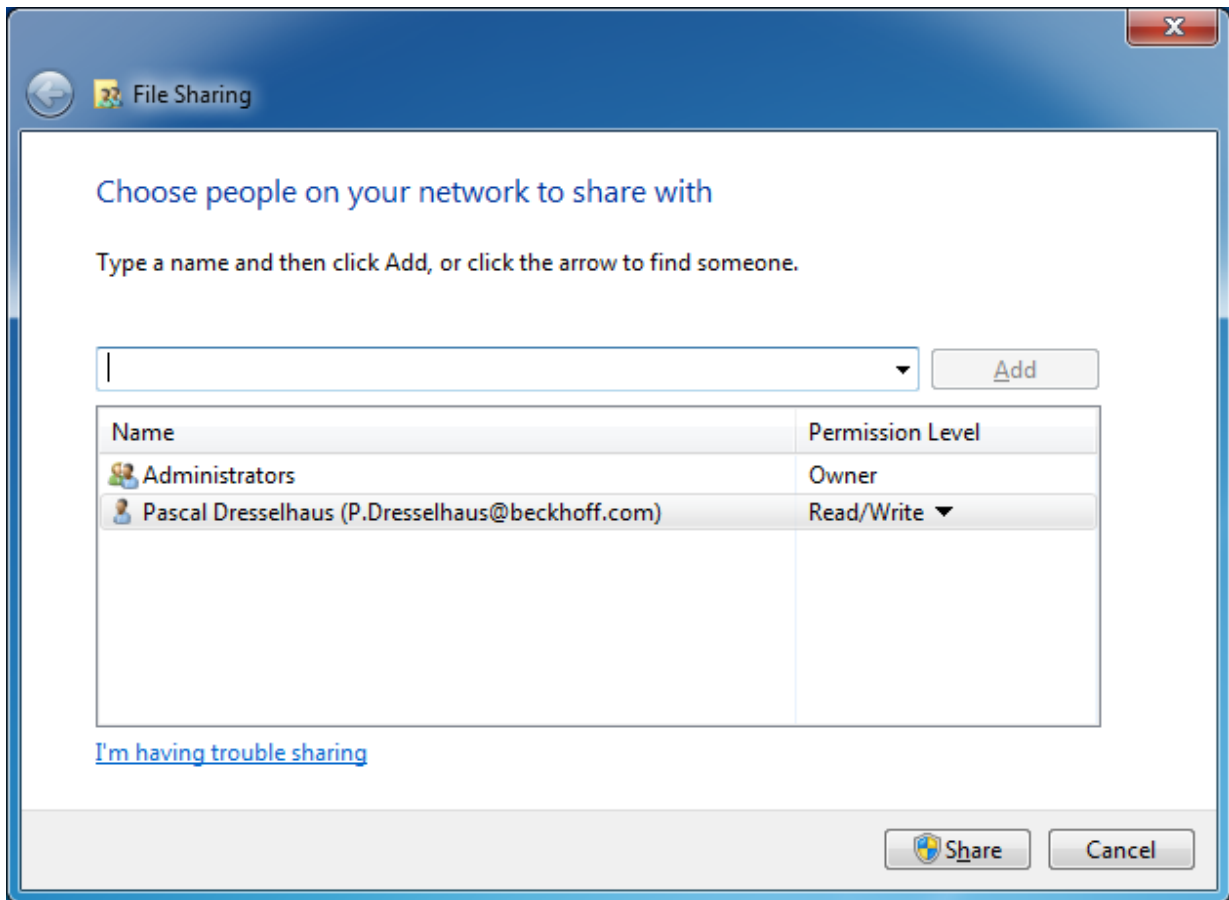
3. Press **[Windows] + [R]** and enter "services.msc". In the **TwinCAT3 Scope Server Properties** dialog select the **Log On** tab.

4. Enter the account with which you have rights of access to the network directory.
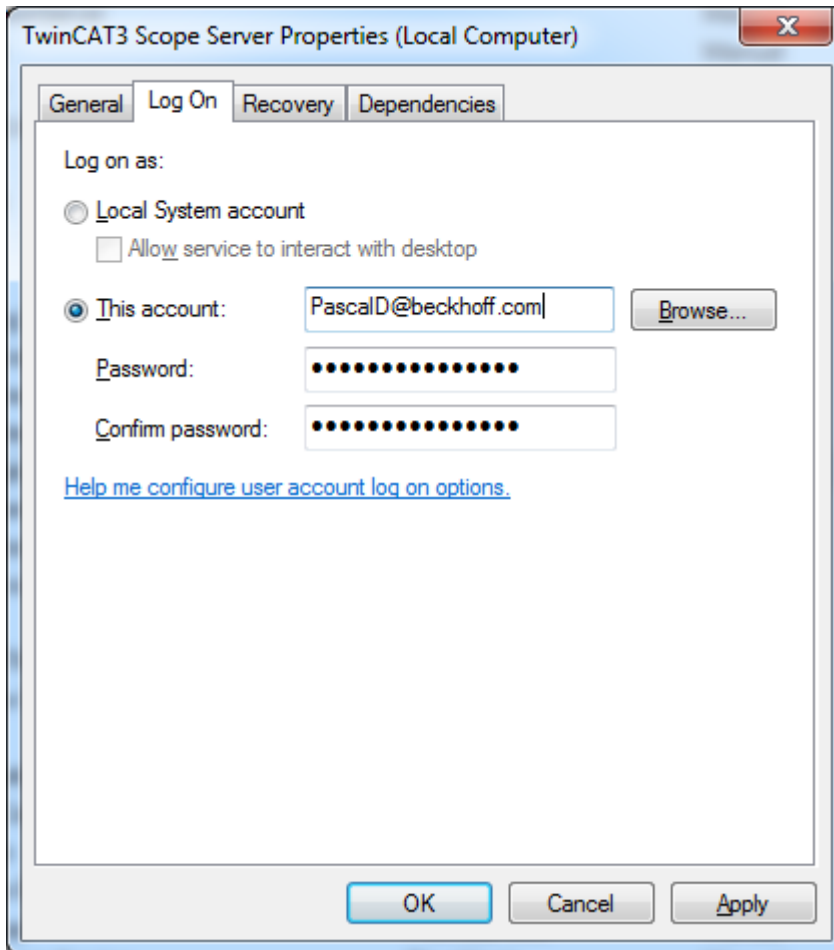
Pay attention to the correct use of your server domain.

5. After confirming, restart the service in order to accept the new user data.

⇨ Subsequently, .svd files from the configured network directory can be opened in the measurement project on this system.

8.1.4 Using old configuration files

The TwinCAT Scope is a very dynamic product. Nevertheless, old configuration files and data files can still be opened in the latest version. If an old configuration file (.sv2 from TwinCAT 2 Scope 2, .tcscope and .tcscopex from TwinCAT 3) opens with a brand new version in which there have been internal changes due to new functions, the Error List in Visual Studio displays the following message:
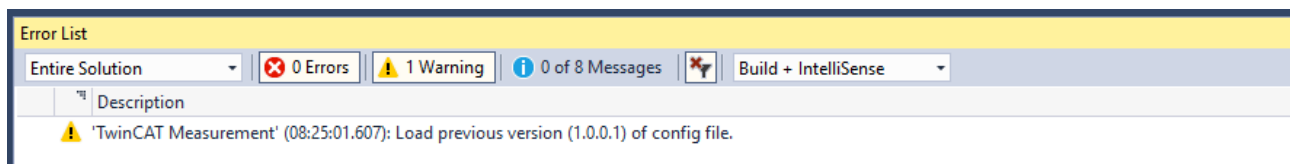
On saving the opened configuration, an updated configuration file will be saved in the project folder. Furthermore, the user will be asked whether he wishes to update the original file. If the new configuration file from the project folder or the updated original file is opened again, the warning does not appear in the Error List. If the original file is not to be updated and the warning no longer displayed, the warning for the Scope can be completely deactivated in the Options [▶ 141] (not recommended!).

8.2   Support and Service

Beckhoff and their partners around the world offer comprehensive support and service, making available fast and competent assistance with all questions related to Beckhoff products and system solutions.

Beckhoff's branch offices and representatives

Please contact your Beckhoff branch office or representative for local support and service on Beckhoff products!

The addresses of Beckhoff's branch offices and representatives round the world can be found on her internet pages:
http://www.beckhoff.com

You will also find further documentation for Beckhoff components there.

Beckhoff Headquarters

Beckhoff Automation GmbH & Co. KG

Huelshorstweg 20
33415 Verl
Germany

Phone:	+49(0)5246/963-0
Fax:	+49(0)5246/963-198
e-mail:	info@beckhoff.com

Beckhoff Support

Support offers you comprehensive technical assistance, helping you not only with the application of individual Beckhoff products, but also with other, wide-ranging services:

- support
- design, programming and commissioning of complex automation systems
- and extensive training program for Beckhoff system components

Hotline:	+49(0)5246/963-157
Fax:	+49(0)5246/963-9157
e-mail:	support@beckhoff.com

Beckhoff Service

The Beckhoff Service Center supports you in all matters of after-sales service:

- on-site service
- repair service
- spare parts service
- hotline service

Hotline:	+49(0)5246/963-460
Fax:	+49(0)5246/963-479
e-mail:	service@beckhoff.com

8.3 FAQ

In this section frequently asked questions are answered in order to make your work with TwinCAT 3 Scope easier. If you have further questions, please contact our support (-157)

1. Is it true that "TC3 Scope" in the TwinCAT 3 world is not a product, but a generic term for various products? [▶ 170]
2. Why are there different product levels for the TC3 Scope? [▶ 170]
3. Can the Scope View Professional be operated with the Scope Server Base? What are the restrictions? [▶ 170]
4. What do you need a TF3300 Scope Server for, when you can also switch on the "Use local server" option? [▶ 170]
5. Can the TwinCAT Scope View be integrated into its own visualization? [▶ 170]
6. Is it possible to print out recorded Scope Charts? [▶ 170]
7. Is multimedia documentation available for TwinCAT 3 Scope? [▶ 170]
8. My Scope .svd files are on a network drive. Why can't I open them? [▶ 170]

Is it true that "TC3 Scope" in the TwinCAT 3 world is not a product, but a generic term for various products?

Yes, "TC3 Scope" is a generic term. The TC3 Scope is subdivided into the Scope View and Scope Server products. This means that a TC3 Scope always consists of the View and Server products. Beyond that there are also different product levels.

Why are there different product levels for the TC3 Scope?

The Scope is a constantly growing tool for data logging and analysis; however, not every Scope user requires the full range of functions for his purposes. Therefore, we offer product levels tailored as far as possible to the application. The currently available product levels are Base and Professional. Base is free of license costs and is an outstanding tool for machine commissioning. Professional is very well suited for process monitoring in addition to machine commissioning.

Can the Scope View Professional be operated with the Scope Server Base? What are the restrictions?

Yes, Professional View can also be operated with the Base Server. Only the limitations of the Base Server version need to be observed. It is also possible, for example, to scope a remote device with the local Base Server. To do this the **Use local server** option must be set in the channel settings.

What do you need a TF3300 Scope Server for, when you can also switch on the "Use local server" option?

The TwinCAT TF3300 Scope Server function is required as soon as a Scope Server has to run autonomously on a remote device. The server can thus be controlled, for example from the PLC, without a View being connected.

Can the TwinCAT Scope View be integrated into its own visualization?

Yes, you can integrate Scope Control into your own .NET-based visualization application. The license model remains identical.

Is it possible to print out recorded Scope Charts?

The currently displayed chart can be printed out by inserting a report in a Scope project. Beyond that, fringe data shown by the cursor, such as channel name, axis name or values, are displayed and added to the printout.

Is multimedia documentation available for TwinCAT 3 Scope?

Yes, there is a recording of a TwinCAT 3 Scope Webinar on the Beckhoff-Homepage.

My Scope .svd files are on a network drive. Why can't I open them?

That is probably because the TwinCAT 3 Scope server service has no rights of access to the network directory. In order to be able to open .svd files, you either need to copy them to the local system or grant the Scope server service the appropriate rights. (See section)

8.4 TwinCAT Scope Glossary

Term	Description
Scope	The Scope forms the highest hierarchical element in a Scope configuration and manages all recording settings.
Scope Server	The Scope Server is the logger program of the TwinCAT Scope. A Scope connects itself to a server in order to record new data or to read an existing file (.svd).
Scope View	The Scope View is the front end of the TwinCAT Scope. From here configurations are created or loaded, the display adjusted and recordings controlled.
YT Chart	The chart provides a graphic area with time axis. All connected channels are scaled to the same time segment.
Overview Chart	An Overview chart can be displayed within a YT chart. It provides an overview of the whole recording period with absolute times.
XY Chart	The chart provides a drawing area for a channel that is calculated from two variables. One variable is shown on the X-axis, the other one on the Y-axis.
Array Bar Chart	The chart provides the drawing interface for an array data type, the elements of which are assigned to a bar line or a bar.
Axis	A (Y-) axis represents the value scaling for connected channels.
Channel	The channel forms the connection of system variable and graph. Therefore, there is one setting window for the acquisition and another one for the general settings (colors, marks etc.).
Data Picker	This is a mouse pointer tool that is automatically activated in the chart when the cursor runs over a data point. Clicking brings up a tool tip showing the values of the X- and Y-axes, the channel name and the absolute time for the selected data point.
Target Browser	All connected systems and their devices can be simply scanned for system variables with the target browser.
Solution Explorer	The Solution Explorer is a kind of project manager, among other things for the measurement projects, which are necessary for a Scope recording.
Measurement Project	The measurement project can contain several Scope projects in which the actual recording configurations are created.
Cursor	The cursor module serves to display graph and axis values and their differences.
Marks	Marks are the actual data points of recorded variables. They can be adjusted via the channel properties.
Trigger	Various actions can be triggered with freely configurable trigger groups.
Quick View Chart	The Quick View chart is available in the Target Browser. It can be used to provide a quick overview of the current value status before the configuration is complete.

Glossary

Array Bar Chart

The chart provides the drawing interface for an array data type, the elements of which are assigned to a bar line or a bar.

Axis

A (Y-) axis represents the value scaling for connected channels.

Channel

The channel forms the connection of system variable and graph. Therefore, there is one setting window for the acquisition and another one for the general settings (colors, marks etc.).

Cursor

The cursor module serves to display graph and axis values and their differences.

Data Picker

Mouse pointer tool that is automatically activated in the chart when the cursor moves over a data point. Clicking brings up a tool tip showing the values of the X- and Y-axes, the channel name and the absolute time for the selected data point.

Marks

Marks are the actual data points of recorded variables. They can be adjusted via the channel properties.

Measurement project

The Measurement project may contain several Scope projects, in which the actual recording configurations are created.

Overview Chart

An Overview chart can be displayed within a YT chart. It provides an overview of the whole recording period with absolute times.

Quick View Chart

The Quick View chart is available in the Target Browser. It quickly and clearly indicates whether values are currently supplied.

Scope

The Scope forms the highest hierarchical element in a Scope configuration and manages all recording settings.

Scope Server

The Scope Server is the logger program of the TwinCAT Scope. A Scope connects itself to a server in order to record new data or to read an existing file (.svd).

Scope View

The Scope View is the front end of the TwinCAT Scope. From here configurations are created or loaded, the display adjusted and recordings controlled.

Solution Explorer

The Solution Explorer is a kind of project manager, which can also be used for the measurement projects that are required for a Scope recording.

Target Browser

All connected systems and their devices can be simply scanned for system variables with the target browser.

Trigger

Various actions can be triggered with freely configurable trigger groups.

XY Chart

The chart provides a drawing area for a channel that is calculated from two variables. One variable is shown on the X-axis, the other one on the Y-axis.

YT Chart

The chart provides a graphic area with time axis. All connected channels are scaled to the same time segment.