

ibaAnalyzer

Working with ibaAnalyzer

Manual Part 2

Issue 8.4

Measurement Systems for Industry and Energy

www.iba-ag.com

Manufacturer

iba AG

Gebhardtstrasse 10-20

90762 Fürth

Germany

Main office: +49 911 97282-0

Email: info@iba-ag.com

Contacts

You can find the contact information for our regional offices or representatives on our website: www.iba-ag.com/en/contact

Unless explicitly stated to the contrary, it is not permitted to pass on or copy this document, nor to make use of its contents or disclose its contents. Infringements are liable for compensation.

© iba AG 2026, All rights reserved.

The content of this publication has been checked for compliance with the described hardware and software. Nevertheless, discrepancies cannot be ruled out, and we do not provide guarantee for complete conformity. However, the information furnished in this publication is updated regularly. Required corrections are contained in the following regulations or can be downloaded on the Internet.

The current version is available for download on our website www.iba-ag.com and can be found in the iba help center docs.iba-ag.com.

Version	Date	Revision	Author	Version SW
8.4	06-2026	HD event table with multiple stores, HD condition query comparing signal and signal	rm, mm	8.4.0

Windows® is a brand and registered trademark of Microsoft Corporation. Other product and company names mentioned in this manual can be labels or registered trademarks of the corresponding owners.

Contents

1 About this documentation 11

1.1 Target group 11

1.2 Notations 11

1.3 Used symbols 12

1.4 Documentation structure 13

2 The data file 14

2.1 Opening the data files 16

2.1.1 Opening a data file 16

2.1.2 Opening multiple data files 20

2.1.3 Defining groups of data files 21

2.1.4 Appending data files 23

2.1.5 Password protection for data files 26

2.1.6 Opening third-party file types 27

2.1.7 Opening archived data files 29

2.1.7.1 Supported formats and unpacking 28

2.1.7.2 Archives filter in the Open file dialog 29

2.1.7.3 PDC format 29

2.2 Advanced search for data files 30

2.3 Slide show 31

2.4 Trend query from file groups or time periods 32

2.5 Closing data files 33

2.6 Process-synchronous analysis 33

2.7 Time shift of data files 34

2.8 Export and import of file trees 37

3 The analysis 38

3.1 Creating a new analysis 39

3.2 Opening an analysis 39

3.3 Saving an analysis 41

3.4 Password protection for analyses 43

3.5 Default analysis file 44

3.6 Changing the connection to ODBC database 45

4 Quick access to analyses and more 47

4.1 Creating new analysis tree 48

4.2 Groups and subgroups in the analysis tree 49

4.3 Analysis (PDO files) in the analysis trees 51

4.4 Signal shortcuts 52

4.5 Expression shortcuts 54

4.6 Marker shortcuts 56

4.7 SQL query 58

4.8 SQL trend query 58

4.9 Export and import of analysis trees 59

5 Settings 60

5.1 X-axis 61

5.1.1 X-axis – Time 61

5.1.2 X-axis – Length 63

5.1.3 X-axis – Frequency 64

5.1.4 X-axis – 1/Length 66

5.2 Y-Axis 66

5.2.1 Y-axis in Preferences 66

5.2.2 Y-axis in Graph setup 68

5.3 Fast Fourier 70

5.4 2D View 72

5.5 3D View 75

5.6 Colors 79

5.7 Fonts 80

5.8 Hardcopy 81

5.9 Miscellaneous 82

5.10 Database 84

5.11 Signal tree 85

5.12 Signal grid 87

5.13 ibaCapture 89

5.14 Overview 92

5.15 Export/import settings 93

5.16 HD server 94
5.17 InSpectra 97
6 Displaying signals 98
6.1 Signal information in the signal tree 98
6.2 .Selecting and displaying signals 99
6.3 Moving signals 100
6.4 Searching for signals 101
6.5 Hiding signals 104
6.6 Removing signals 105
6.7 Moving graphs 106
6.8 Hiding graphs 107
6.9 Removing graphs 108
6.10 Scale signals 108
6.11 Y-axis 108
6.12 Shift scales 108
6.13 Compressing and stretching scales 109
6.14 Formatting the legend 110
6.15 Zoom in and out 110
6.16 Intervals in graphs 111
6.16.1 Showing digital signal intervals 112
6.16.2 Showing intervals between markers 113
6.16.3 Configuration of the intervals 113
6.17 Using the navigator 116
6.17.1 Navigator X-range 117
6.18 Autoscrolling 118
6.19 Markers 119
6.19.1 Interactive markers 120
6.19.2 Harmonic markers 120
6.19.3 X-axis markers (calculated markers) 123
6.19.4 Dynamic marker labels 128

6.20 X-axis modes (reference axes) 129
6.20.1 X-axis modes Time based and Length based 130
6.20.2 X-axis mode FFT 130
6.20.3 X-axis mode X - Y 133
6.21 Views 135
6.21.1 Standard view 135
6.21.2 3D view 137
6.21.2.1 3D wireframe 137
6.21.2.2 3D surface 138
6.21.2.3 2D top view 138
6.22 FFT view (ibaAnalyzer-InSpectra) 141
6.23 Orbit view (ibaAnalyzer-InSpectra) 141
6.24 Computation module 142
6.24.1 Creating computation module 143
6.24.2 Creating and configuring a profile 144
6.24.3 Linking inputs to the computation module 147
6.24.4 Placing and using a computation module 150
6.25 Audio player 151
6.25.1 Turning on the audio player 151
6.25.2 Audio player playback 151
6.25.3 Audio player volume 152
6.25.4 Audio player synchronization with video 152
6.26 PDA trend view 153
6.26.1 Functions of the PDA trend view 154
6.26.2 Properties of the PDA trend view 155
6.27 Map view 156
6.27.1 Configuring routes 158
6.27.2 Map area 160
6.27.3 Playback area of the map view 161
6.27.4 Map view properties 163
6.28 InCycle-Expert view (ibaAnalyzer-InCycle) 166
6.29 Event table 166
6.29.1 Use of the event table 167

7 Creating new signals 169

7.1 Adding signal in the signal table 169

7.2 Logical expressions 172

7.2.1 Logical expressions dialog window 173

7.2.2 Generating a simple signal 175

7.2.3 Creating vector signals (array) 177

7.2.4 Zone control for vector signals 181

7.2.5 Export and import of logical expressions 184

7.2.6 References of logical expressions 185

7.2.7 Groups for logical expressions 185

7.2.8 Renaming logical expressions 187

8 Macros 188

8.1 Macro designer 189

8.1.1 Area for input variables 191

8.1.2 Area for intermediate values 192

8.1.3 Example 1: Calculating the surface area within a hysteresis curve 194

8.1.4 Example 2: Calculating the head, fillet and tail of an aluminum strip 196

8.2 Applying macros in the expression builder 199

8.3 Export and import of macros 199

8.3.1 Exporting and importing global macros 200

8.3.2 Exporting and importing local macros 201

8.4 Password protection for macros 202

9 Filter editor 204

9.1 Filter editor dialog window 204

9.1.1 Signal selection 207

9.1.2 Editing the curve field and filters 208

9.1.3 Example: Create a bandstop filter for 50 Hz 208

9.2 Export and import of filters 213

9.2.1 Exporting and importing global filters 213

9.2.2 Exporting and importing local filters 214

10 Text signals 216

10.1 Display of text signals 216

10.2 Functions for text signals 216

10.3 Text signals in videos 217

11 Access to HD data with ibaAnalyzer 219

11.1 Opening the HD query dialog 219

11.2 Configuration of ibaHD-Server connection and HD queries 220

11.2.1 Setting up a connection to the ibaHD server 221

11.2.2 Time selection for the HD query 222

11.2.2.1 Time selection – signal tree 224

11.2.2.2 Time selection – preview of the HD trend graph 225

11.2.2.3 Selection of the preferred time base 226

11.2.3 Formulating signal conditions 228

11.2.3.1 Formulating conditions for events 230

11.2.3.2 Example: Formulating signal condition 231

11.2.4 Time period queries 233

11.2.4.1 Operation of the time period table 234

11.2.4.2 Configuring time period queries 236

11.2.5 Query of non-equidistant data from ibaHD-Server 240

11.3 HD query results 242

11.4 HD query results of an event-based HD store 243

11.5 Query results of the time periods 244

11.6 Updating the info fields of the time periods 245

11.7 Drill-down function 247

11.8 Subchannels min/max 249

11.9 Export and import of HD queries and time periods 251

11.10 Appending an HD Query 253

11.11 Replacing a file by HD query 254

11.12 Automation of HD analyses 255

12 Analysis with ibaCapture videos 256

12.1 Showing video recordings 256

12.2 Functions in the video window 260

12.3 Analysis of measurement signals based on video recording 263

12.4 Playing protected videos 264

12.5 Export of video data 266

12.6 Printing and reporting of video images 267

12.6.1 Print with video images 267

12.6.2 Report with video images 268

13 Print function (Hardcopy) 269

13.1 Requirements and setup 269

13.2 Print preview settings 269

14 Using views in other programs 272

14.1 Exchange of tables and graphs via the clipboard 272

14.2 Exchange of individual graphs as image files 275

15 Data export and extraction 276

15.1 Supported file formats 276

15.1.1 iba format (DAT file) 277

15.1.2 Text file (CSV) 278

15.1.3 COMTRADE 280

15.1.4 Apache Parquet 281

15.1.5 Matlab 284

15.2 Exporting data 286

15.2.1 Signal selection 287

15.2.2 Selection of time criteria 288

15.2.2.1 Time range 289

15.2.2.2 Time base 290

15.3 Extracting files 291

15.3.1 Data Extractor 292

15.3.1.1 Extractor output 294

15.3.1.2 Archive profile assignment 295

15.3.1.3 Info columns 300

15.3.1.4 Computed columns 303
15.3.1.5 Diagnosis and notification 305
15.3.1.6 Renumbering 307
15.3.2 File extraction via the command line 308
15.3.3 Automatic extraction using ibaDatCoordinator 309
15.4 Export of text signals to an ASCII file 310
15.5 Export of graph signals to text file 312
15.6 Export of graph signals to clipboard 313
16 Report generator 314
17 Support and contact 315

1 About this documentation

This documentation describes the function and application of the software

ibaAnalyzer.

1.1 Target group

This documentation addresses qualified professionals, who are familiar with handling electrical and electronic modules as well as communication and measurement technology. A person is regarded as trained specialist personnel if they are capable of assessing the work assigned to them and identifying possible risks based on their specialist training, knowledge, experience and awareness of the applicable regulations.

This documentation addresses in particular professionals who are in charge of analyzing measured data and process data. Because the data is supplied by other iba products the following knowledge is required or at least helpful when working with *ibaAnalyzer*:

- Operating system Windows
- *ibaPDA* (creation and structure of the measuring data files)

1.2 Notations

In this manual, the following notations are used:

Action	Notation
Menu command	Menu *Logic diagram*
Calling the menu command	*Step 1 – Step 2 – Step 3 – Step x* Example: Select the menu *Logic diagram – Add – New function block*.
Keys	<Key name> Example: <Alt>; <F1>
Press the keys simultaneously	<Key name> + <Key name> Example: <Alt> + <Ctrl>
Buttons	<Key name> Example: <OK>; <Cancel>
Filenames, paths	`Filename`, `Path` Example: `Test.docx`

1.3 Used symbols

If safety instructions or other notes are used in this manual, they mean:

Danger!



The non-observance of this safety information may result in an imminent risk of death or severe injury!

Observe the specified measures.

Warning!



The non-observance of this safety information may result in a potential risk of death or severe injury!

Observe the specified measures.

Caution!



The non-observance of this safety information may result in a potential risk of injury or material damage!

Observe the specified measures.

Note



A note specifies special requirements or actions to be observed.

Tip



Tip or example as a helpful note or insider tip to make the work a little bit easier.

Other documentation



Reference to additional documentation or further reading.

1.4 Documentation structure

This documentation describes the functionality of the *ibaAnalyzer* software in detail. It is designed both as a tutorial as well as a reference document.

In addition to this documentation, you can examine the version history in the main menu, *Help – Version history* (file `versions.htm`), for the latest information about the installed version of the program. This file not only lists the bugs that have been eliminated but also refers to extensions of the system in note form.

In addition, special "NewFeatures…" documentation comes with any software update that includes significant new features, which provides a more detailed description of the new features.

The state of the software to which the respective part of this documentation refers is listed in the revision table on page 2.

The *ibaAnalyzer* documentation (PDF version) is divided into six separate parts. Each part has its own section and page numbering beginning at 1 and is updated independently.

Part	Title	Content
Part 1	Introduction and Installation	General notes, licenses and add-ons Installation and program start User interface
Part 2	Working with *ibaAnalyzer*	Working with measurement files and analyses, presentation features, macro configuration, filter design, preferences, printing, export, file extract, interfaces to *ibaHD-Server*, and *ibaCapture*
Part 3	Expression editor	Directory of all calculation functions in the expression builder, including explanation
Part 4	Database interface	Working with data from databases, connecting to the database, writing iba measurement data to databases, extracting the data from the database and analyzing the data.
Part 5	Application examples	*In preparation*

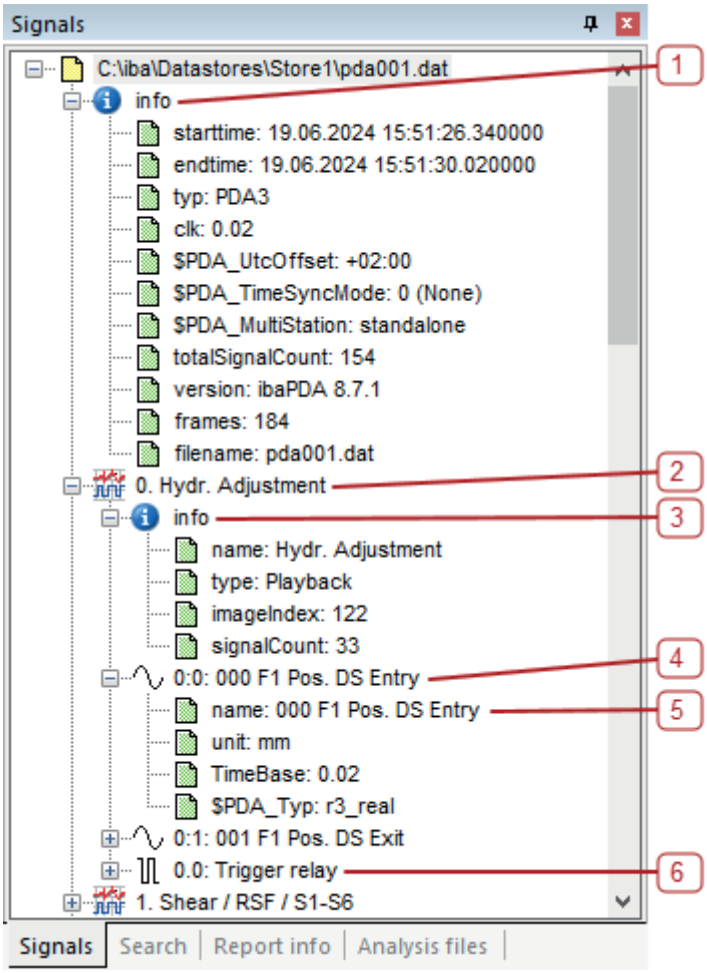
2 The data file

A data file for the purposes of *ibaAnalyzer* contains measured values and additional information generated by an iba data acquisition system. The data files have the extension .dat. *ibaAnalyzer* can only read data files, but not change them.

Windows Explorer marks these data files with the relevant icon .

The content of the data file is shown in the signal tree window. Alongside the actual measured values, process-synchronous acquisition systems such as *ibaPDA* also store additional information in the data file which can be displayed and evaluated in *ibaAnalyzer*.

The following areas are under each data file in the signal tree window:



1	File information	4	Analog signal
2	Module	5	Signal information
3	Module information	6	Digital signal

In line with the iba module concept, the signals are presented below the module level in a tree structure. Furthermore, additional information on the data files and signals is also available.

You can display these info fields from the data file like measurement signals in the graphs. You can use the features in the expression builder, the extraction dialogs and the report generator to process the information from practically all info fields.

The most important info fields in the info branch:

- clk: Acquisition timebase in seconds
- typ: File type
- starttime: Start of recording (date, time)
- frames: Number of measuring cycles
- starttrigger: Distance of the start trigger from the beginning of the file, expressed as a number of frames
- stoptrigger: Distance of the stop trigger from the beginning of the file, expressed as a number of frames

The data file may contain further information depending on the application and type.

If you have selected calculated signals, measurement signals or text signals in the data storage configuration for the *Files – Infofields* node in *ibaPDA*, they also appear in the *ibaAnalyzer*.

2.1 Opening the data files

You have various options for opening data files in *ibaAnalyzer*. There are also differences that lead to different display options for the data files. If you open several files at the same time, you can open the files next to each other or appended to each other. The following chapters describe the various procedures.

2.1.1 Opening a data file

ibaPDA, *ibaQDR* and *ibaLogic* store measured data in data files according to defined measurement specifications so that the measured data is available for analysis. The data files have the extension .dat. Windows Explorer marks these data files with the relevant icon .

You can open data files in different ways. If the data files were password-protected during creation with *ibaPDA*, then you have to enter the password for any of the following methods.

For more information about the password protection of data files, see ↗ *Password protection for data files, page 26*.

Opening a data file with Windows Explorer

You can open DAT files like other files by double-clicking in Windows Explorer. If *ibaAnalyzer* is installed, all DAT files are linked to *ibaAnalyzer* and are opened in *ibaAnalyzer*.

Alternatively, you can open the DAT files using drag & drop:

- Drag the file into the open *ibaAnalyzer* window.
- Drag the file onto the *ibaAnalyzer* desktop icon.

If files are already open in *ibaAnalyzer*, the dropping location decides whether the new file is added or attached. If you drop the file over an existing file in the signal tree, the new file is appended, see ↗ *Appending data files, page 23*.

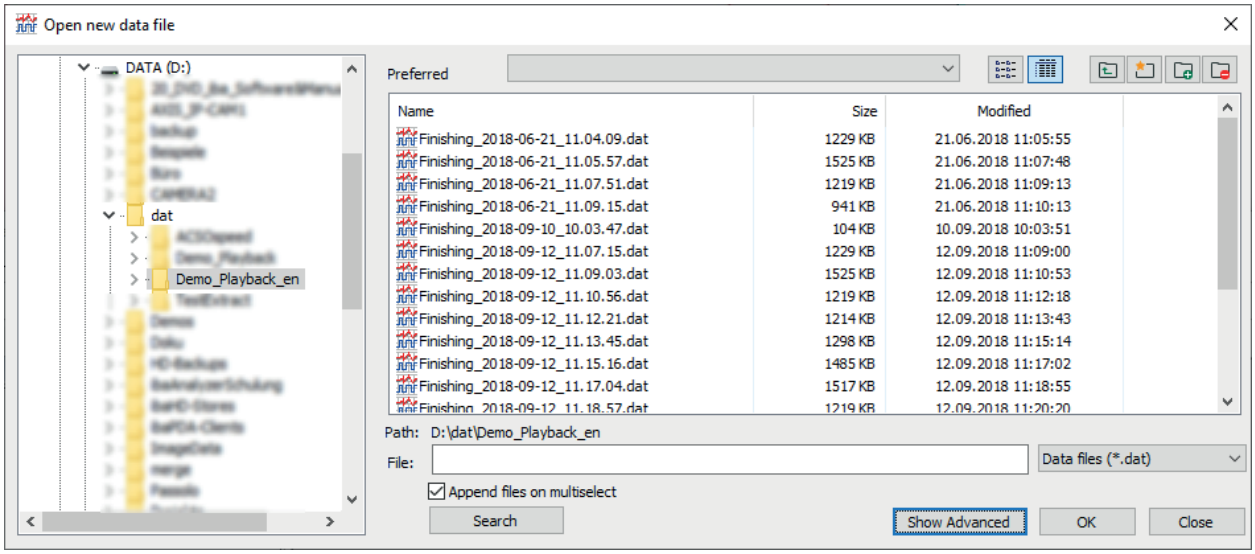
Opening a data file with ibaAnalyzer dialog

You can open data files via *File – Open data file* or the folder button in the toolbar.



In the dialog, select the folder and the desired file and confirm with <OK>.

If files are already open in *ibaAnalyzer*, they will be closed, and the new file will be opened in the signal tree.



Tip

If no data files are visible in the folder, this may be due to the following reasons:

- There are no data files in the folder.
- The incorrect *Open analysis* dialog is open. *ibaAnalyzer* suppresses the display of other file types.
- The wrong file type is set.

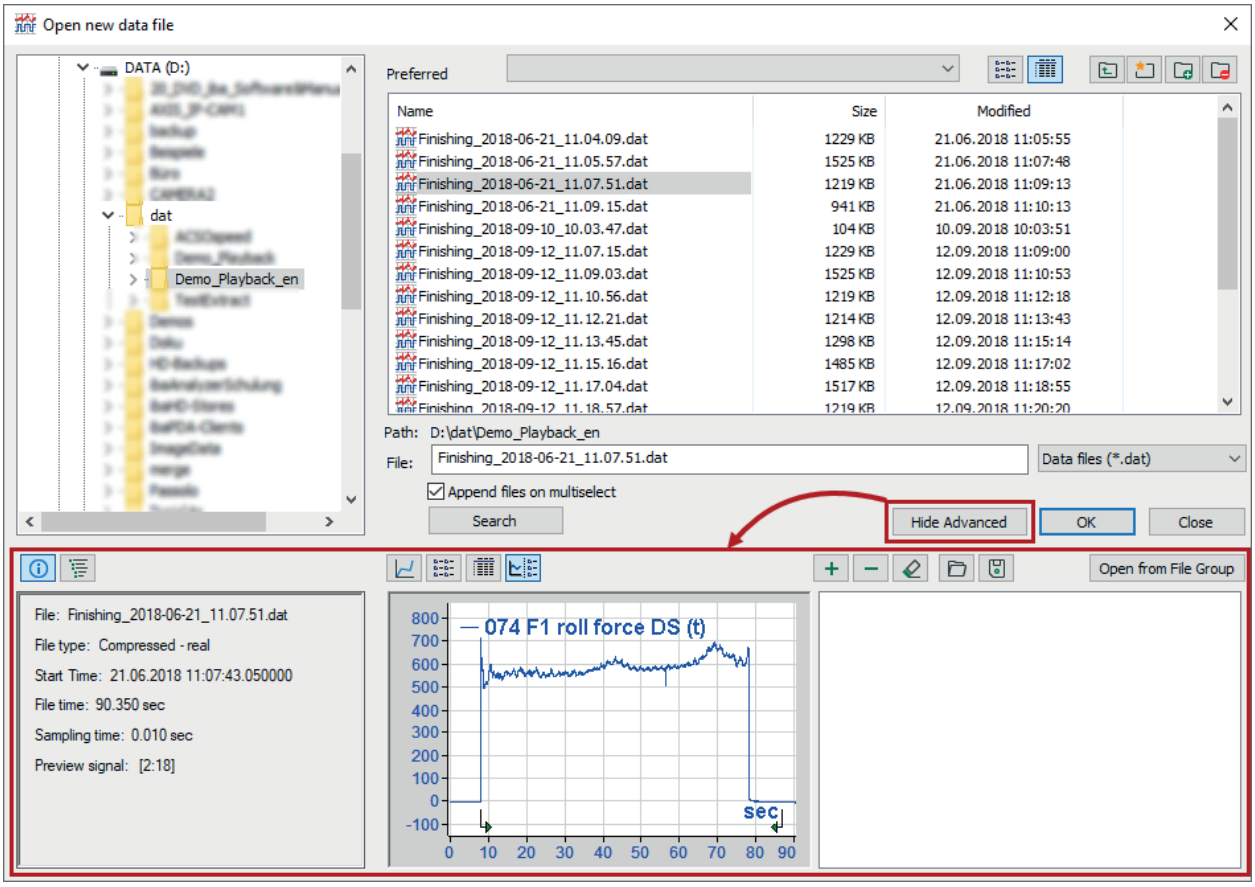
Preferred folder

If data files are saved in a specific directory (e.g. `D:\dat`), you can set this folder as the preferred folder. After selecting the folder in the dialog, click on the folder button with the plus sign at the top right. You can define several preferred folders and open them via the drop-down list.

<Show advanced>/<Hide advanced>

Use this button to switch the preview for a signal in the selected data file on or off. The additional information helps you to roughly assess the data file.

The preview function is only available for original iba data files.



Buttons in the preview area

Symbol	Function	Description
ⓘ	View file info	Shows the information of the selected data file.
	View signal tree	Shows the signal tree in the selected data file
	Preview of curve	Shows the curve of a signal that was marked in the signal tree on the left.
	Preview of file list	Shows the file list if several data files were selected, so that several files can be opened or a file group defined.
	Preview of file list with details	Shows the file list with detailed information if several data files were selected, so that several files can be opened or a file group defined.
	Preview of curve and file list	Shows the curve and the file list. The same signal is displayed in each case depending on which file is marked in the file list.
+	Add files to file group	Adds selected files from the upper window (browser) to the file group
−	Remove files from file group	Removes files that are marked in the file group from the file group.

Symbol	Function	Description
	Remove all files from file group	Removes all files from the file group in the preview area.
	Load saved file group (text file)	Opens a dialog to open a text file that contains a data file group. After selecting the text file and confirming with <OK> in this dialog, the data files specified in the text file appear in the file list.
	Save file group (text file)	Saves the current file group as a text file. Enter a name and path. You can also create these files manually according to the required pattern using an ASCII editor.

The preview with curve and file list is particularly helpful for making a good pre-selection of data files when you select a prominent signal for viewing.

You can also fill the file list in the preview area with files using drag & drop. For further information, see chapter ↗ *Defining groups of data files, page 21*.

2.1.2 Opening multiple data files

In *ibaAnalyzer*, you can open any number of data files simultaneously, which is useful for analyzing and comparing signals from different files.

Opening multiple data files with Windows Explorer

To open multiple data files from Windows Explorer, you have the following options:

- Drag individual files one by one into the open *ibaAnalyzer* window. If you drop the file over an existing file in the signal tree, the new file is appended, see ↗ *Appending data files, page 23*.
- Select multiple files (e.g. all files in the directory with <Ctrl>+<A>) and drag them all at once into the open *ibaAnalyzer* window.
- If multiple data files are packed in an archive file (*.pdc, *.zip, *.tar, etc.), you can drag the archive file into the signal tree window. The files contained within are opened on the same level, see ↗ *Opening archived data files, page 29*.

Note



If you select multiple files in Windows Explorer and press the <Enter> key, the files are opened in separate instances of *ibaAnalyzer* and not in the same window.

Opening multiple data files with ibaAnalyzer dialog

You can open multiple data files via *File – Open data file* or the folder button in the toolbar.

In the dialog, select the desired files in the browser area and confirm with <OK>.

If files are already open in *ibaAnalyzer*, they will be closed, and the new files will be opened in the signal tree.

You can also use wildcards (* and ?) in the *File* field to open multiple files at once.

Adding files

You can add one or more data files to already open files via *File – Add new data file* or through the folder button with a plus sign in the toolbar.



You can choose whether to open the files in parallel or append them in the signal tree, see ↗ *Appending data files, page 23*. Additionally, you can also create file groups, see ↗ *Defining groups of data files, page 21*.

2.1.3 Defining groups of data files

Using a group of data files is convenient if a series of similar data files, e.g. one file per manufactured product or per test run, are to be analyzed one after the other.

You also need data file groups for the following functions:

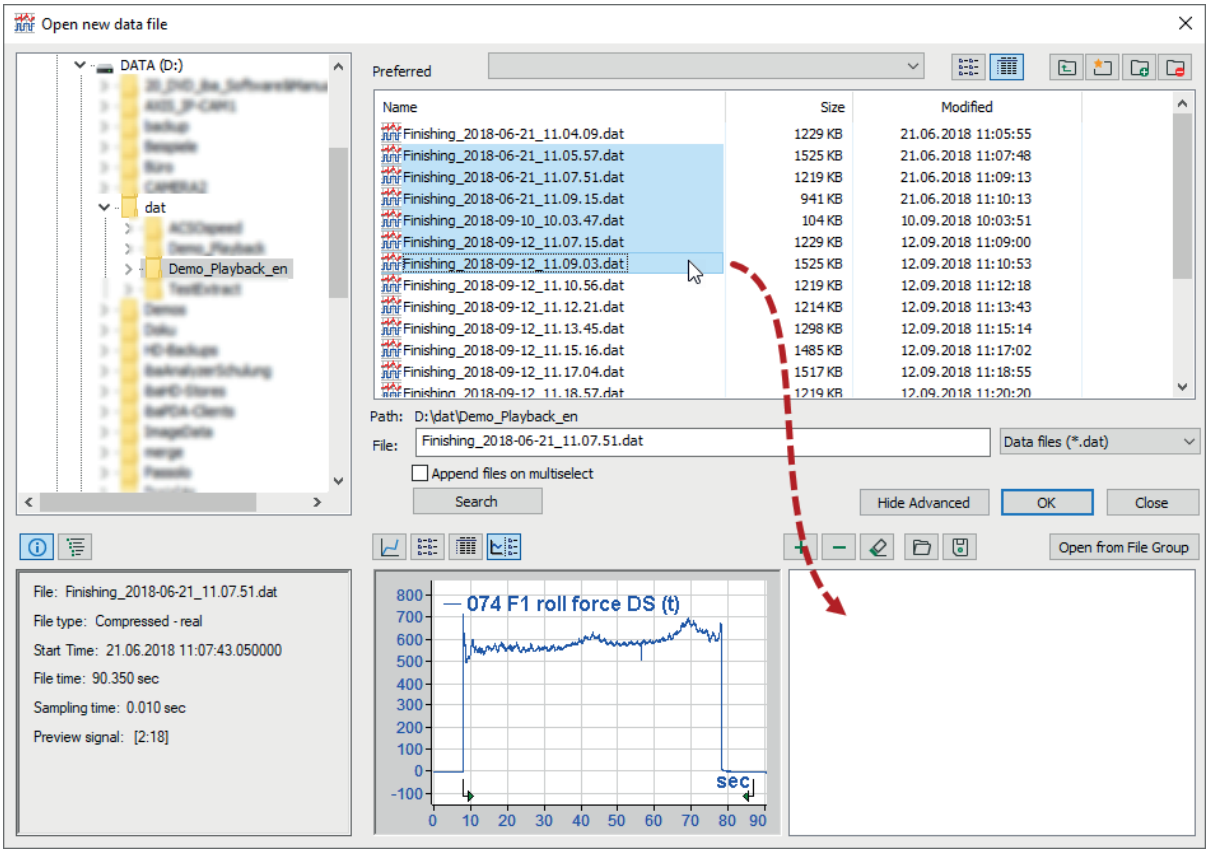
- Slide show of the data file group, see ↗ *Slide show, page 31*
- Querying the info fields in the data file group, see ↗ *Trend query from file groups or time periods, page 32*

Procedure

1. Open the *Open new data file* dialog via *File – Open data file* or the folder button in the toolbar.

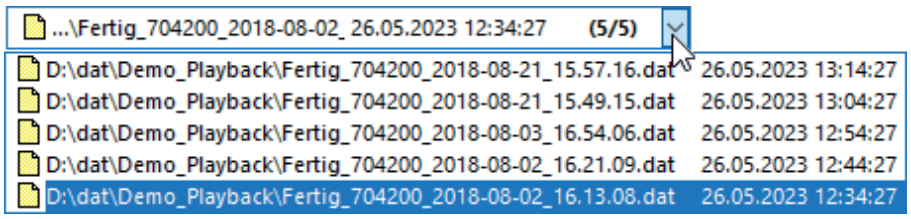


2. If not activated: Expand the dialog box via the <Show advanced> button and select the fourth view with graph preview and file group list.
3. Select the desired folder in the browser area.
4. Select several files. Drag & drop the files into the area of the file group list at the bottom.

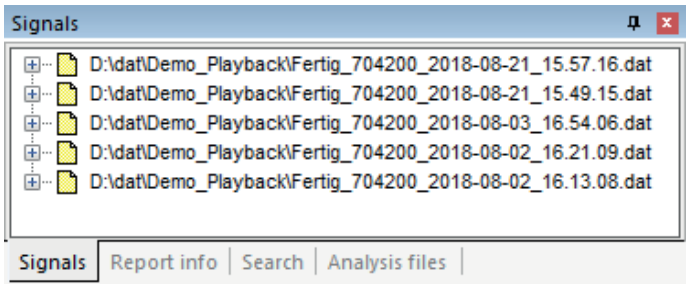


5. Select the files in the group and click <OK>.

→ The data files are now available in the selection list of the file group in the toolbar:



6. Optional: If you select the files in the group and click on <Open from file group> instead of <OK>, the data files are opened in parallel and they appear in the signal tree window.



7. Select the data file you want to analyze from the file group or the signal tree.

Information on the operating elements of the preview area can be found in chapter ↗ *Opening a data file, page 16*.

2.1.4 Appending data files

Cascading or appending data files is particularly useful when you want to view a signal trend across multiple data files. While process-synchronous recording systems like *ibaPDA* enable continuous recording, they often generate several smaller data files, each covering a manageable period, such as 10 minutes.

If you want to view an overview of the signal trend for an hour, you can append the 6 files, each 10 minutes long, in *ibaAnalyzer* and display the signal trends as a whole.

Appending data files with Windows Explorer

To append data files from the Windows Explorer, you must already have a data file open in *ibaAnalyzer*.

Drag and drop additional data files one by one into the signal tree. If you drop the file onto an existing file in the signal tree, the new file is appended.

Note

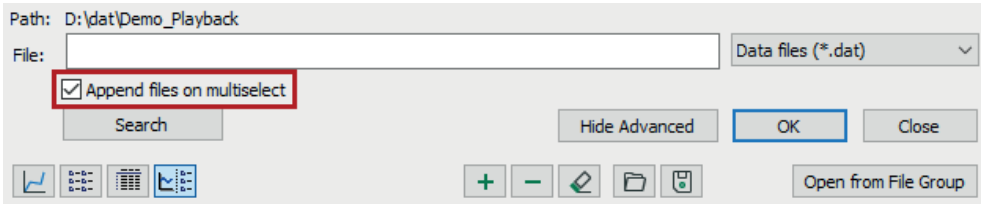
Pay attention the correct chronological order of the data files so that the displayed curve also corresponds to the real chronological sequence.

iba AG recommends activating the option *Synchronize files on recording time* in the X-axis settings to avoid errors in the sequence. *ibaAnalyzer* then inserts the data file at the correct position. If this option is not activated, a new file is inserted directly after the file to on which it was dropped.

Appending data files with the ibaAnalyzer dialog

You can append files directly to one another when you open several files or create file groups.

1. Open the *Open new data file* dialog via *File – Open data file* or the folder button in the toolbar.
2. Select the option *Append files on multiselect*.



3. Select the desired files in the browser area.
4. Optional for file groups:
   a) Expand the dialog box via the <Show advanced> button and select the fourth view with graph preview and file group list.
   b) Drag & drop the files into the area of the file group list at the bottom.
   c) Click on <Open from File Group>.
   → The files are opened in the signal tree as concatenated files and as a file group.
5. Confirm with <OK>.

→ The files are opened in the signal tree as concatenated files.

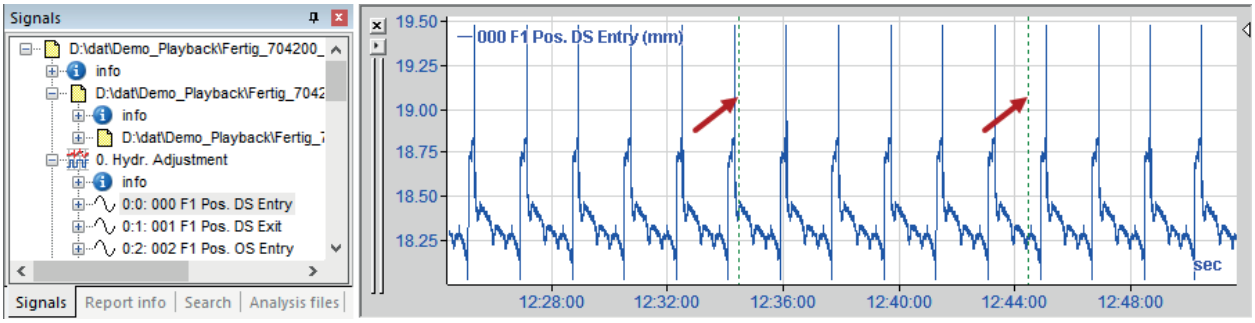
You can append one or more files to an already open file at any time.

1. Through *File – Append new data file* or the context menu in the signal tree, you can open the dialog to append data files.
2. In the dialog, select the desired files in the browser area and confirm with OK.

→ The new files are appended to the already opened file.

Displaying appended data files

The appended files are displayed as follows:



The file boundaries are shown as vertical green dashed lines. If the end time of a data file is not available, e.g. when appending database query results, the lines are displayed in magenta.

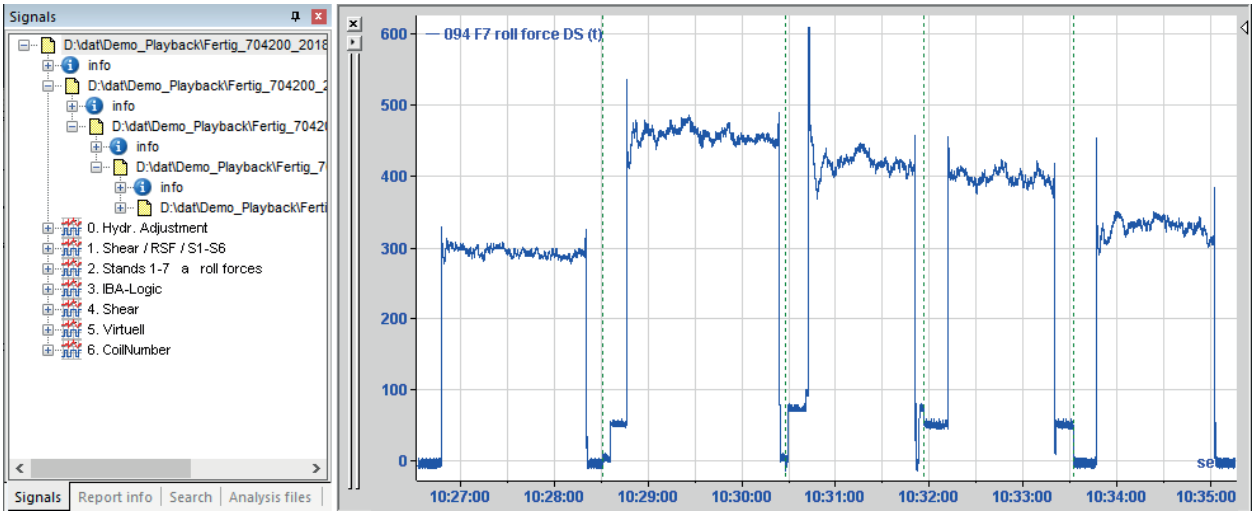
If no lines are displayed, you can enable the display in the preferences or graph setup. Go to *2D View* and enable the option *Show triggers and file separators*.

Displaying with real recording time

When a series of data files is recorded continuously in sequence, the alignment of values on the time axis is accurate, as shown in the image above.

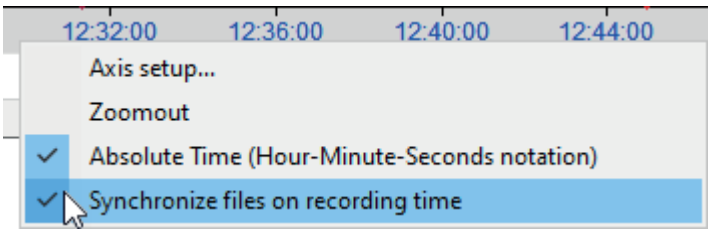
However, if the data files were not recorded exactly one after the other, the time axis information is only correct for the first file. *ibaAnalyzer* does not evaluate the timestamp of the data file by default and therefore appends the files seamlessly.

In the image below, the signal trend lasts only about 8 minutes, but the data files were recorded with large time gaps.

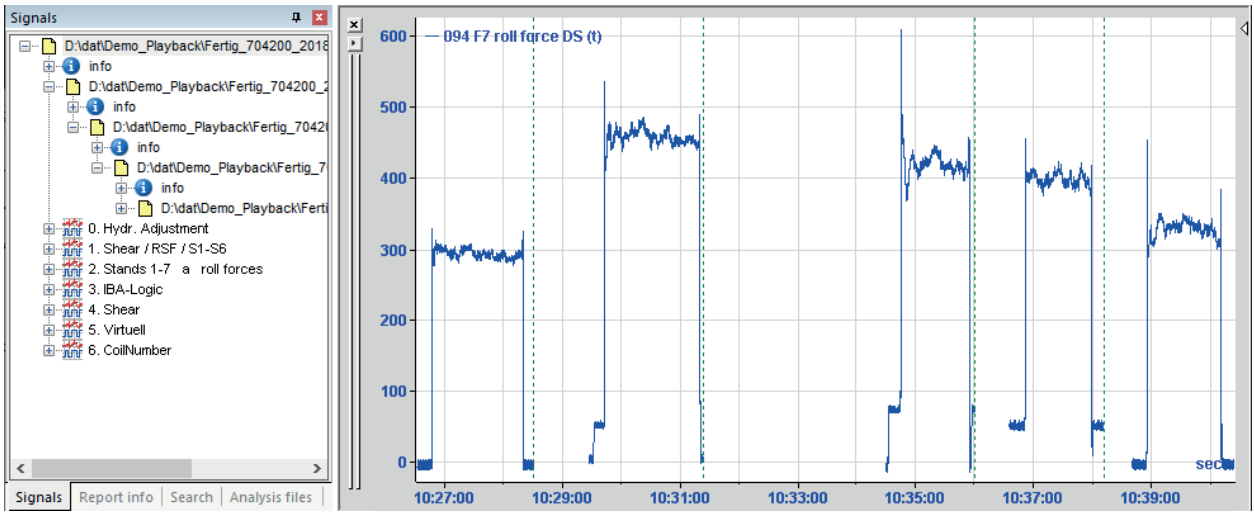


To display appended data files in their real positions on the X-axis, you need to synchronize the display with the actual recording time. You have 2 possibilities for this:

- In the *Graph setup*, under *X-Axis – Time*, enable the options *Absolute Time (Hours - Minutes - Seconds notation)* and *Synchronize files on recording time*.
- Open the context menu on the X-axis and enable the options *Absolute Time (Hours - Minutes - Seconds notation)* and *Synchronize files on recording time*.



The display with synchronized recording time is shown in the image below.



Tip

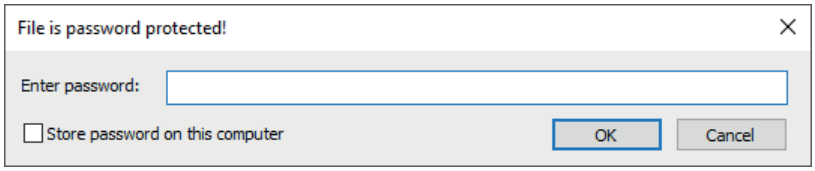
For long periods, you can also display the days. In the settings, additionally enable the option *Always show date*.

2.1.5 Password protection for data files

Both *ibaAnalyzer* and *ibaPDA* support the creation of password-protected data files.

Opening password-protected data files

If you open a password-protected file with *ibaAnalyzer*, you must first enter the password.



After entering the password once, the password is saved for the duration of the *ibaAnalyzer* session so that you do not have to enter it again to open further data files.

Saving the password on the computer

You can permanently save the password by enabling the option *Store password on this computer* in the password dialog. This way it is no longer necessary to repeatedly input the password across several sessions.

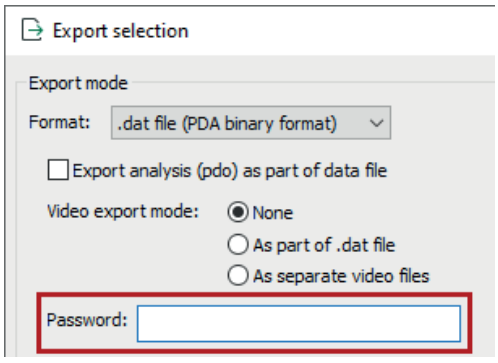
If you want to remove the saved password from your computer, use the command *Clear data file password* in the *File* menu.

Creating password-protected data files

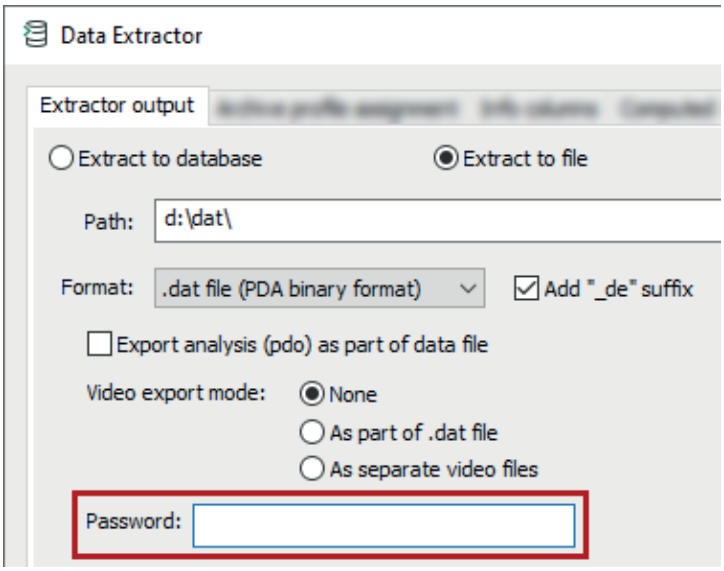
If you create a new data file through export or extraction using *ibaAnalyzer*, you can protect this file with a password.

To do so, enter a password in the corresponding field.

Password entry for export



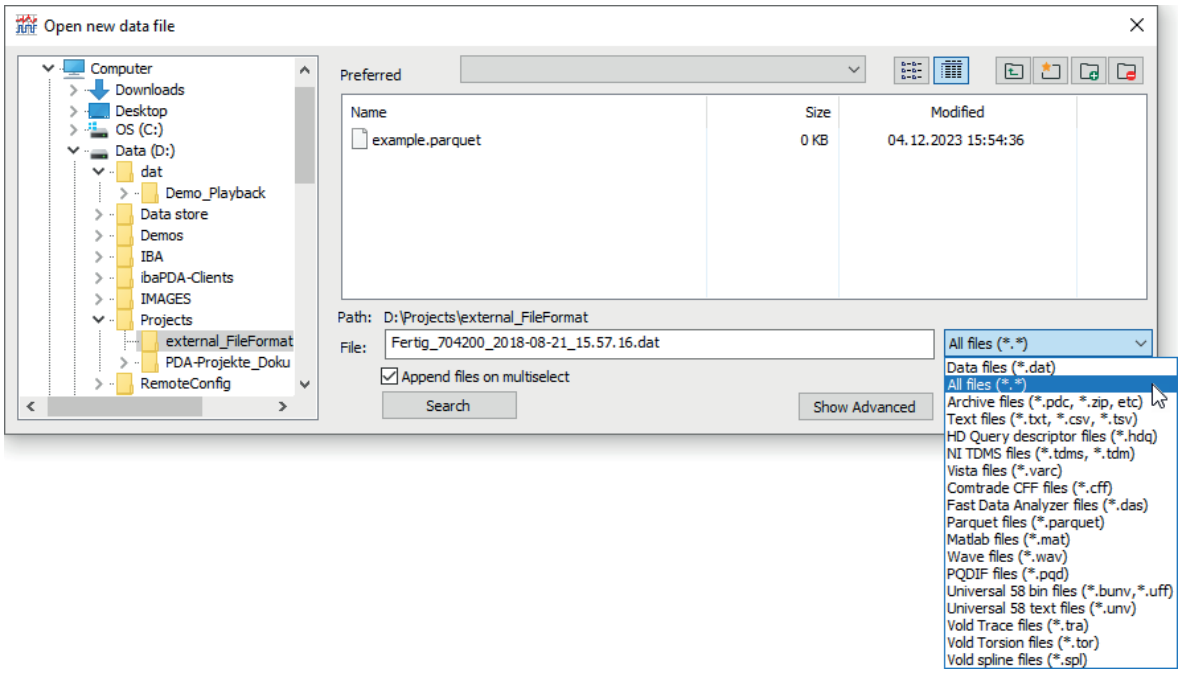
Password entry for extraction



If you leave the password field empty, a data file without a password will be created regardless of whether the original file had a password.

2.1.6 Opening third-party file types

If you have an *ibaAnalyzer-E-Dat* license, you can also open data files in other file formats.



The directory window only shows files of the preset type. The available options are:

File type	Description	ibaAnalyzer-E-Dat license required
.dat	iba data file format	-
.pdc, .zip …	Archive files, see ↗ *Opening archived data files, page 29*	-
.hdq	HD query files that contain the rule for an HD query (pseudo data file)	-
.txt., .csv	ASCII text files, CSV files	Yes
.cff	Comtrade CFF files, Comtrade 2013 files	Yes
.das	Fast Data Analyzer files, data files from the Danieli FDA system	Yes and *ibaAnalyzer* as 32-bit version
.mat	Matlab files	Yes
.parquet	Parquet files	Yes
.pqd	Power Quality Data Interchange Format (PQDIF), binary format in IEEE standard 1159.3 from the power quality field	Yes and *ibaAnalyzer* as 32-bit version
.tmds, .tdm	National Instruments TDMS files	Yes
.tra, .tor, .spl	Vold Trace, Vold Torsion, Vold Spline	Yes
.bunv, .uff, .unv	Universal 58 binary files and universal 58 text files	Yes
.varc	Vista Controls Vlogger files	Yes and *ibaAnalyzer* as 32-bit version
.wav	Only uncompressed PCM wave files, e.g. from Siemens SM1281	Yes

Thanks to Unicode support (UTF-8, UTF-16), the characters that can be used in the files are not restricted.

Other documentation



You can find more information about handling third-party file formats in the *ibaAnalyzer-E-Dat* product manual.

2.1.7 Opening archived data files

Besides iba data files (*.dat), *ibaAnalyzer* can open data files of other supported file types (*.csv, *.parquet etc.) even when they are zipped in a common archive file.

Moreover, analysis files (*.pdo) and report layout files (*.lst) can be part of the archive and loaded automatically.

All files must be stored in the root directory of the archive. Folders will be ignored.

2.1.7.1 Supported formats and unpacking

ibaAnalyzer supports the following formats:

- .zip
- .rar
- .7z
- .xz
- .tar
- .bz2
- .gz
- .gzip

ibaAnalyzer opens the archive file and unpacks the contents into the temporary standard path of Windows (usually `C:\Users\[Username]\AppData\Local\Temp`). Afterwards, the files are opened normally. When you close *ibaAnalyzer*, the temporary files will be deleted.

Note

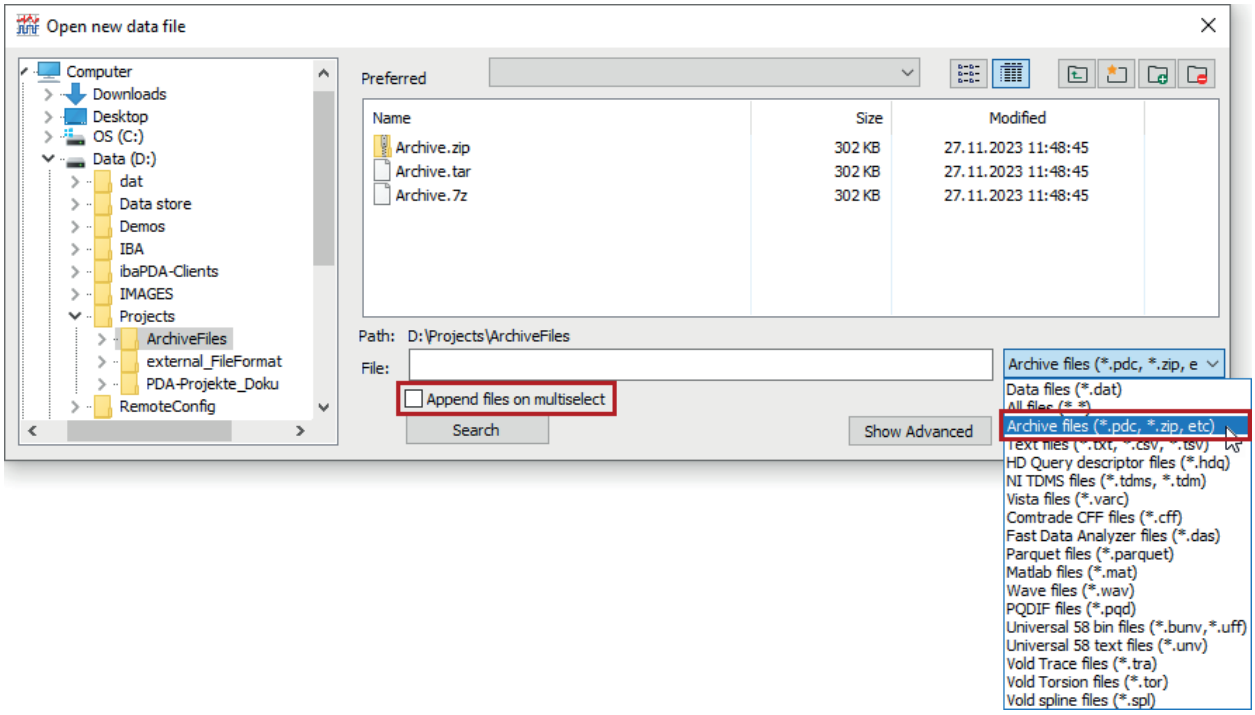


You may edit the open files. However, changes will not be stored in the files in the archive.

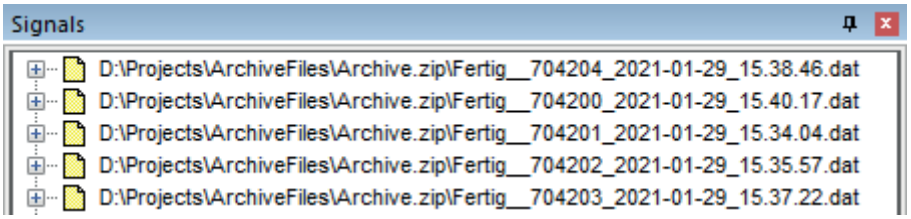
2.1.7.2 Archives filter in the Open file dialog

The dialog *Open new data file* offers in the drop-down list for file types a filter option *Archive files (*.pdc, *.zip, etc.)* for showing supported archive files only.

The option *Append files on multiselect* also works for archive files if more than one data file is included. Correspondingly, you can use the switch "/append" when opening by command line.



After opening the data files appear together with the archive name in the signal tree window.



2.1.7.3 PDC format

Beside the standard archive formats, a special format *.pdc* (Process Data Container) is available.

These files are normal ZIP archives with a different suffix and have the same functionality as the standard archives.

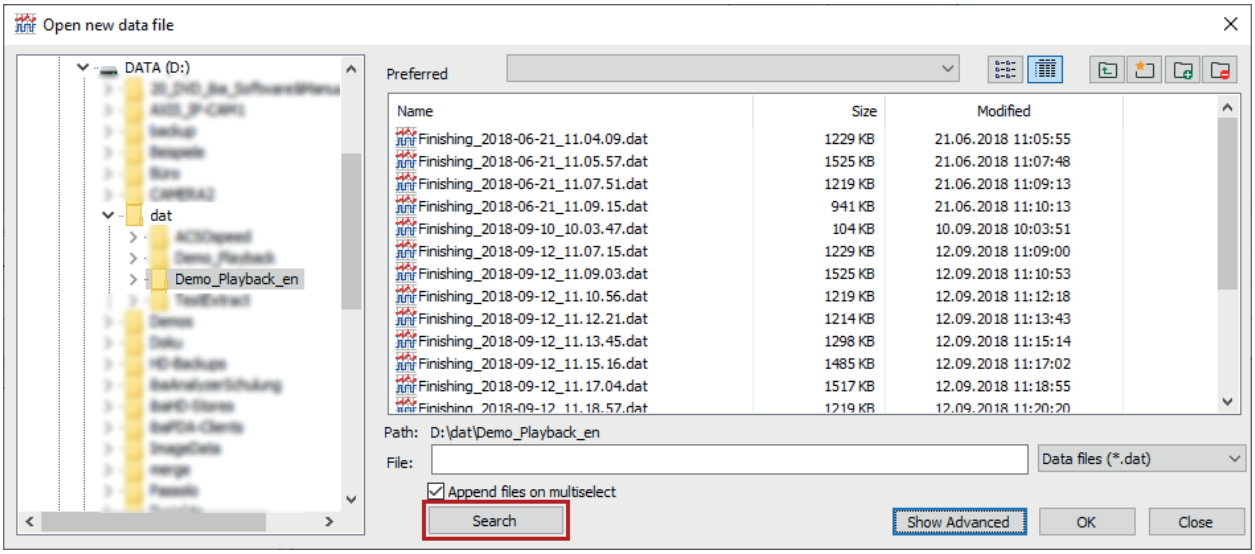
The advantage is, though, that *ibaAnalyzer* registers the file ending *.pdc*, and thus you can open these files using double-click in the Windows Explorer. Furthermore, you can assign *ibaAnalyzer* as default application to these files, e.g. for downloads from *ibaDaVIS*.

The files have a separate icon:

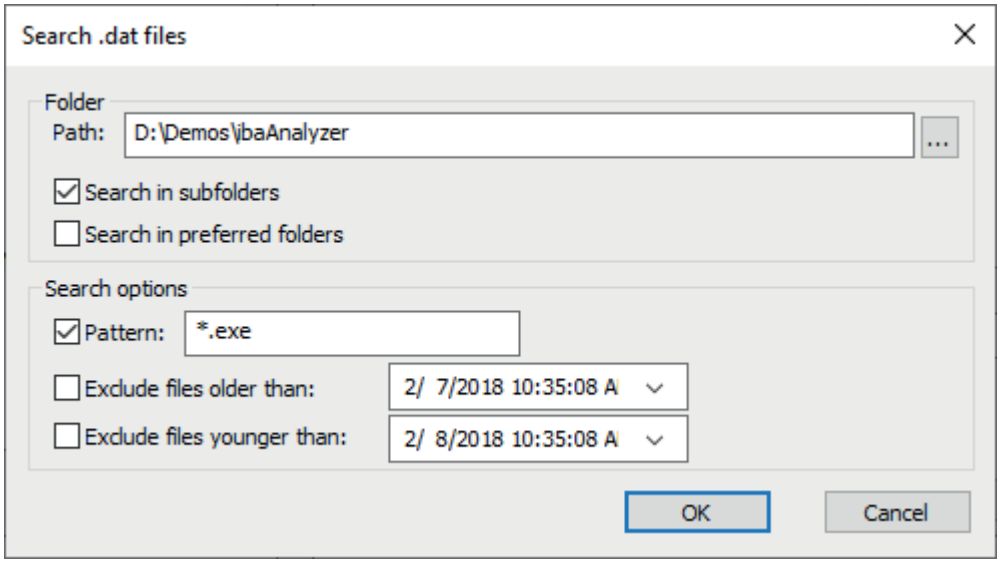


2.2 Advanced search for data files

The *Open new data file* dialog window includes a <Search> button. This allows you to search specifically for data files or other file types supported by *ibaAnalyzer* (.dat, .txt, .csv, .hdq etc.).



When you click on the <Search> button, the following dialog opens up:



There are various options for specifying and restricting your search.

Path
Set the initial path for the search.

Search subfolders
If you enable this option, the search includes the subfolders in the specified path.

Search preferred folders
If you enable this option, the search includes your saved preferred folders.

Pattern
Enter a search pattern to find the files you want, e.g. "Product_*.dat". You can also search for other file types. The search is limited to iba data files and file types supported by the *ibaAnalyzer-E-Dat* license however.

Exclude/include data files …
Specify a time limit for your search.

Search result

When the search is complete, the first file found is shown in the *Open new data file* dialog. The corresponding folder is marked on the left in the directory structure. The preview window shows all the other files found as a group.

2.3 Slide show

The so-called slide show function enables the consecutive opening of the data files of a group in *ibaAnalyzer*, so you can view these files for a editable time span. In this way, you can view many files one after the other with the same analysis to get an overview.

In the toolbar for the data file group list, you can select the files of a group in the drop-down list, see ↗ *Defining groups of data files, page 21*.



You can use the two arrow buttons to switch between the data files in the group.

Start the slide show with the button .

ibaAnalyzer now opens the data files of the group one after the other. You can set the time interval under *Preferences – Miscellaneous – Slide show timer*.

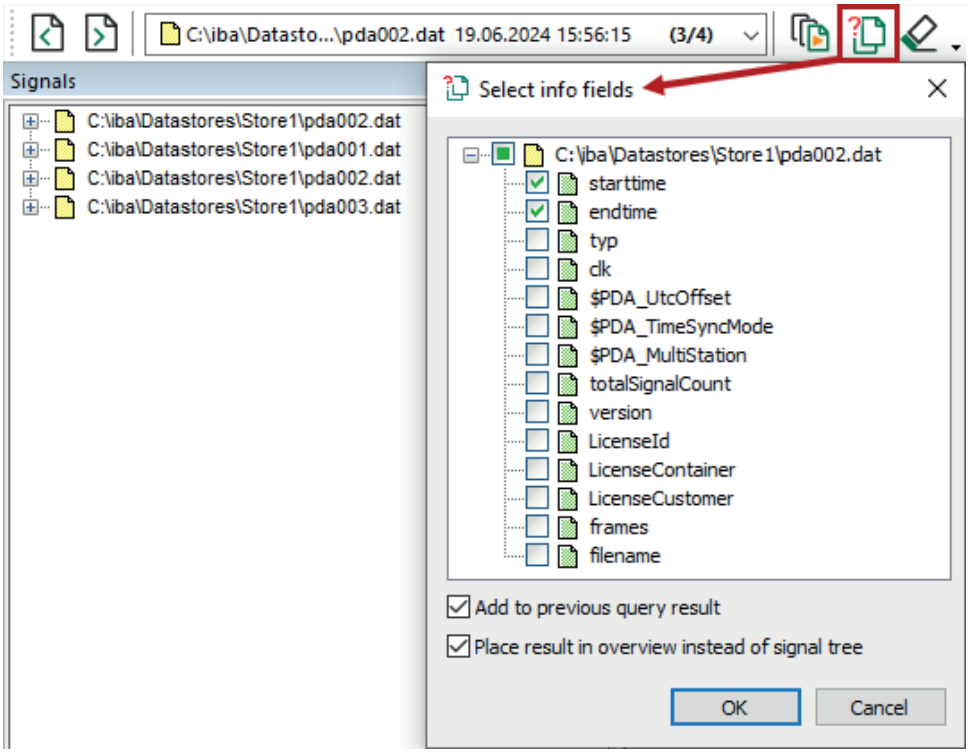
Click the slide show button again to end the slide show.

2.4 Trend query from file groups or time periods

With the trend query from file groups, you can query the info fields of the individual files in the group. You can use data files or HD queries or time periods that you open as a group. Depending on the selection, the results of the query appear in the signal tree or in the *Overview* tab.

Note

The group trend query only works reliably if all elements in the group have this info field. This is typically the case for data files from the same recording or for time periods from the same time period store.



If you click on the button to query from groups, a dialog opens in which you can select the info fields. The file displayed in the file group is used as an example. However, the query includes all files or time periods in the file group.

You can configure the display of the query results as follows:

Add to previous query result
If you deactivate this option (default), the previous query result is overwritten if you have previously carried out a query. Otherwise, the current query result is added to the other query result in the *Trend query result*s node in the signal tree.

Place result in overview instead of signal tree
If you enable this option, the trend query results appear in the *Overview* tab instead of in the signal tree.

2.5 Closing data files

Similar to opening, you have various options for closing a DAT file:

- When opening a new DAT file (not adding or appending), the current file is automatically closed.
- If several DAT files are open, select one or more files in the signal tree that you want to close. Select the *Close data file* command in the context menu of the signal tree window or in the *File* menu.
- To close all open DAT files at the same time, select *Close all data files* in the context menu of the signal tree window or in the *File* menu.

2.6 Process-synchronous analysis

With *ibaAnalyzer*, you can open data files that are being generated by *ibaPDA*, for example, and analyze the data recorded up to that point. With the appropriate default settings, *ibaAnalyzer* reloads the relevant data file at cyclical intervals so that the analysis is completed step by step. For more information on the preferences, see ↗ *Miscellaneous, page 82*.

Requirements

You have specified the path to which the data files are written, e.g. from *ibaPDA*, in the *Preferences* of *ibaAnalyzer* under *Miscellaneous – Autoload data files*.

Procedure

1. In *ibaAnalyzer*, click the button .

→ Wait until a data file appears in the signal tree window.

2. Switch on the cyclical reload function via the button .

→ For a continuous process-synchronous analysis, both buttons must remain activated.



3. Option of manual reload:

   To reload the data file manually, deactivate the automatic reload function (see step 2).

   Reload the data files manually using the button .

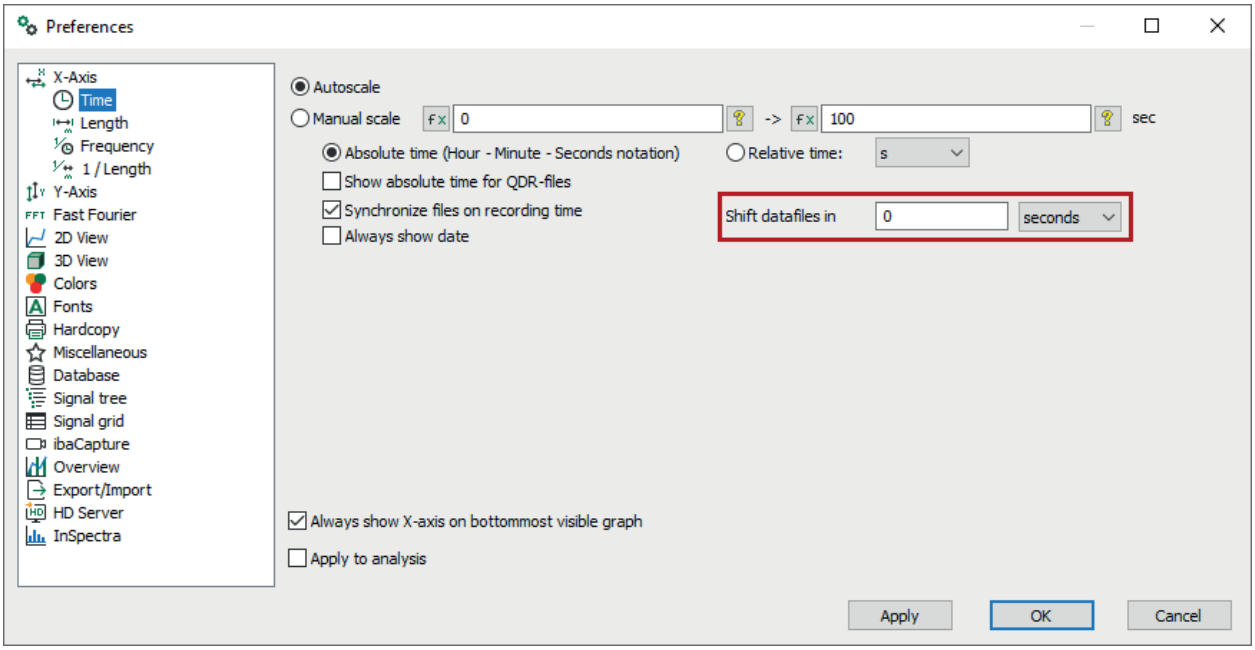
2.7 Time shift of data files

If you open several data files at the same time, you can overlay signal curves from different measurements and compare them with each other. With untriggered recordings, characteristic signal events are often not at the same position on the time axis. You can therefore move the data files along the time axis.

Collective time shift

If required, you can specify a collective time shift in the preferences or in the graph setup in the *X-axis – Time* node. The time shift then applies to all open data files.

Enter a time value. A positive value shifts the curve to the right, a negative value to the left.



Note

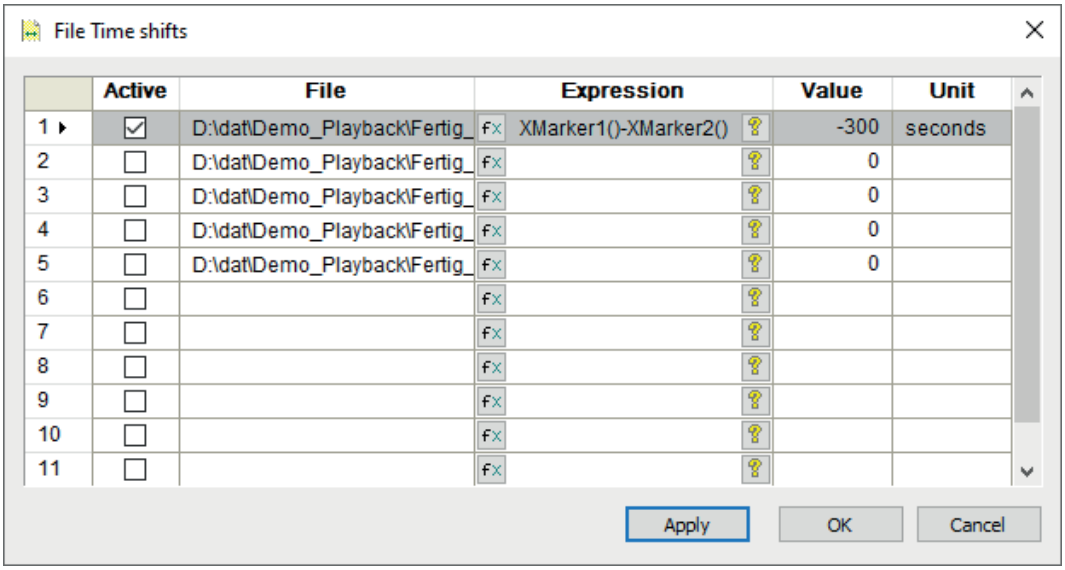
If you apply a collective time shift to data files, the collective time shift overwrites the existing individual shifts. Therefore, specify the collective time shift first and then the individual time shift.

The time shift setting is stored in the analysis file. Thus, a time shift is immediately applied to an open data file as soon as the analysis with time shift has been opened.

File-specific time shift

1. Use the *Time shift data file* command in the context menu of the signal tree to open the *File time shift* dialog.

→ The dialog shows a table with all currently open data files. You can define an individual time shift for each file.



2. In the *Active* column, tick the file you want to shift.
3. In the *Expression* column, enter a positive or negative time value. A positive value shifts the curve to the right, a negative time value to the left.

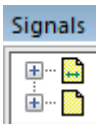
   Alternatively, you can also open the Expression builder using the <fx> button and enter a formula to calculate the shift.

Tip

Use markers to determine the required time value. See also the example below.

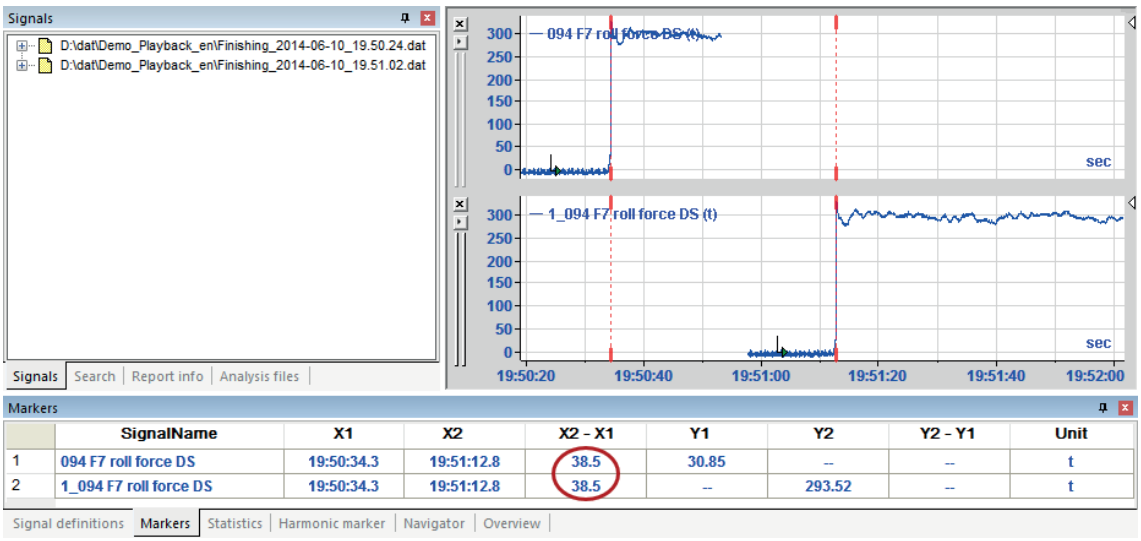
4. Select the unit for the value in the *Unit* column.
5. To display the result of the calculation in the Value column, click <Apply>.

→ The file with the time shift is marked with a green double arrow in the signal tree.



Example: Alignment of two signal trends at marker position

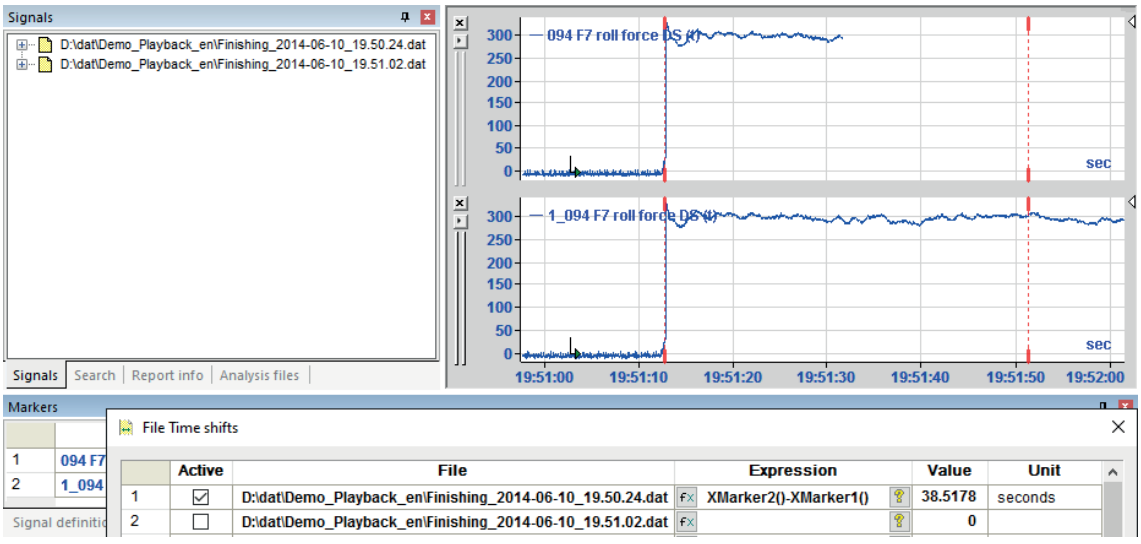
The trends of two rolling force signals from two consecutive data files give the curves shown below:



To better compare the two signal curves, the rising edges of the signal are to be overlaid. The markers are each positioned on a similar event in the graphs.

The position indicator in the *Markers* tab shows a difference of 1 min 48.06 s between the markers (X2-X1).

Therefore, the upper curve must be shifted to the left by exactly this value so that the signal curves overlap. The difference of the marker positions is used as expression for the time shift.



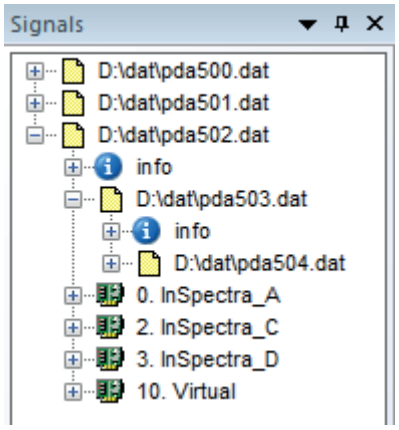
To compare the curves exactly, you can also place the curves together in a graph in the shifted state.

2.8 Export and import of file trees

This command can export the file tree to a text file or import it from a text file. When you use the correct syntax (see example in the table), you can create the file with any text editor.

The table shows the file tree as TXT file with parallel and appended data files.

Signal tree	Content of the export file (*.txt)	Description

	`[0]`
	`D:\dat\pda500.dat`
	`[1]`
	`D:\dat\pda501.dat`
	`[2]`
	`D:\dat\pda502.dat`
	`D:\dat\pda503.dat`
	`D:\dat\pda504.dat`
		1. Data file index
		Path and file name
		2. Data file index
		Path and file name
		3. Data file index
		Path and file name
		Path and file name
		Path and file name

Tip



To start *ibaAnalyzer* via the command line and specify a file tree at the same time, you can use the */filetree* switch.

Example:

```
C:\Program files\iba\ibaAnalyzer\ibaAnalyzer.exe /filet-
ree: MyFileTree.txt
```

3 The analysis

An analysis includes the complete configuration of the user interface as well as all analysis-related add-ons, such as expressions or virtual signals. An analysis can be applied to any number of data files.

Analyses are saved as a file with the extension .pdo. These files allow you to save individual settings and use them at any time.

An analysis file includes the following items:

- Number, sequence and size of graphs
- Signal composition (module number and signal number)
- Graph setup, e.g. axis scaling, appearance and colors.
- Mathematical and logical calculations (expressions)
- Logical expressions (virtual signals)
- Settings for log printing, including additional text fields
- Settings for the report generator
- When using the database interface: All settings for data extraction (archive profiles, computed columns, etc.) and reading data.
- All other settings made under *Graph Setup*.
- The selected tab in the signal tree

Note



The signals in an analysis are referenced via the module number and signal number. This means you can also apply an analysis to data files that do not match the analysis exactly, as long as the data files contain signals with the same module numbers and signal numbers. The values are displayed without an error message.

You can open PDO files like other files by double-clicking them in Windows Explorer. All PDO files are linked to *ibaAnalyzer* and are opened in *ibaAnalyzer*.

ibaAnalyzer then starts with the settings saved in the analysis file. Data files are only opened if the name of a data file is stored in the analysis (see ↗ *Saving an analysis , page 41*).

3.1 Creating a new analysis

When you start *ibaAnalyzer* directly or via a data file, the signal table (signal definitions) and the recorder window are empty.

Using a data file containing the data you want to analyze, you can build up an analysis step by step as you need:

- Open graphs in the recorder window.
- Create calculations and expressions.
- Create virtual signals.
- Configure reports.
- etc.

Save the analysis via *File – Save analysis* or the button .

To start a new analysis and discard the current settings, go to *File – New analysis*, or click the button .

Loaded data files stay in the signal tree window. The new analysis does not yet have a name and is untitled.

3.2 Opening an analysis

You can open analysis files in various ways. If the analysis is password-protected, you must enter the password when using each of the following methods.

For more information on password protection of analysis, see ↗ *Password protection for analyses, page 43*.

Open analysis with Windows Explorer

You can open PDO files like other files by double-clicking them in Windows Explorer.

Alternatively, you can open the PDO files using drag & drop:

- Drag the file into the open *ibaAnalyzer* window. Any already open analysis will be closed.
- Drag the file onto the *ibaAnalyzer* desktop icon.

Open analysis with ibaAnalyzer dialog

You can open an analysis via *File – Open analysis* or the toolbar PDO folder button.



In the dialog, select the folder and the desired file and confirm with <OK>.

If analysis are already open in *ibaAnalyzer*, they will be closed and the new file opened.

Tip

If no analysis files are visible in the folder, this may be for the following reasons:

- There are no PDO files in the folder.
- The wrong *Open new data* file dialog has been opened. *ibaAnalyzer* is hiding other file types.

Preferred folders

If analysis are saved in a specific directory (e.g. `D:\pdo`), you can specify it as the preferred folder. After selecting the folder in the dialog, click on the folder button with the plus sign at the top right. You can define multiple preferred folders and open them via the drop-down list.

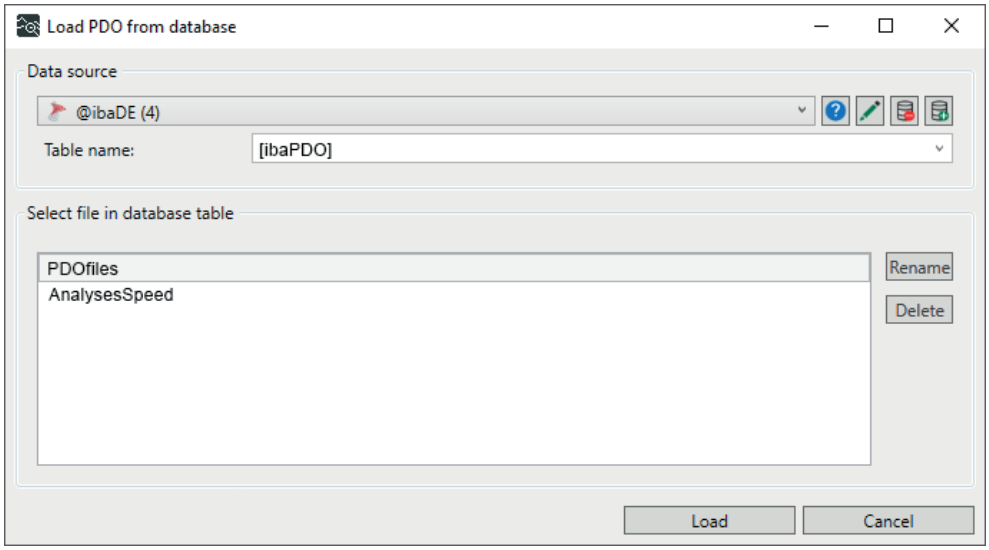
Retrieve analysis from database

In *ibaAnalyzer*, you can save analysis files not only within the file system but also in a database, see ↗ *Save analysis in a database, page 41*. You can load those analyses from the database and open them in *ibaAnalyzer*.

To do this, you must have configured a database connection in *ibaAnalyzer*. For more information on database connections, see *ibaAnalyzer* manual part 4, chapter *Configuring the database connection*.

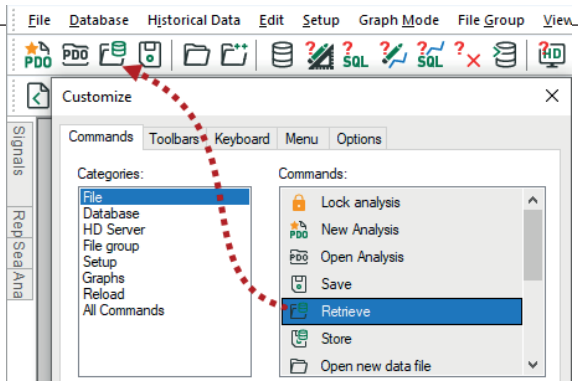
Procedure

1. Go to *File – Retrieve analysis from database*.
2. In the dialog, select the database as the data source, the table, and then the desired file.



3. To open the selected file, click <Load>.

button on the toolbar. To do this, in the *View – Toolbar Setup* menu simply drag the relevant button from the dialog into the toolbar.



3.3 Saving an analysis

In the analysis file, you can save your settings in *ibaAnalyzer*, analysis-related add-ons, etc.

If the analysis is password-protected, you must enter the password each time you save. This prevents unauthorized persons from modifying the analysis. Password protection for an analysis is set in a different dialog, see ↗ *Password protection for analyses, page 43*.

Save analysis with ibaAnalyzer dialog

You can save an analysis via *File – Save analysis as* or the toolbar Save button.



In the dialog, select the folder and enter a name for the file.

Optionally, you can save the names of the data files that are open at the time of saving in the analysis. This is helpful if you have created an analysis specifically for a data file. Note that only the file name is saved, not the file itself. If you want to use the analysis with the measurement data on another computer, you must also copy or transfer the relevant data file.

Preferred folders

If analysis files are saved in a specific directory (e.g. `D:\pdo`), you can specify it as the preferred folder. After selecting the folder in the dialog, click on the folder button with the plus sign at the top right. You can define multiple preferred folders and open them via the drop-down list.

Save analysis in a database

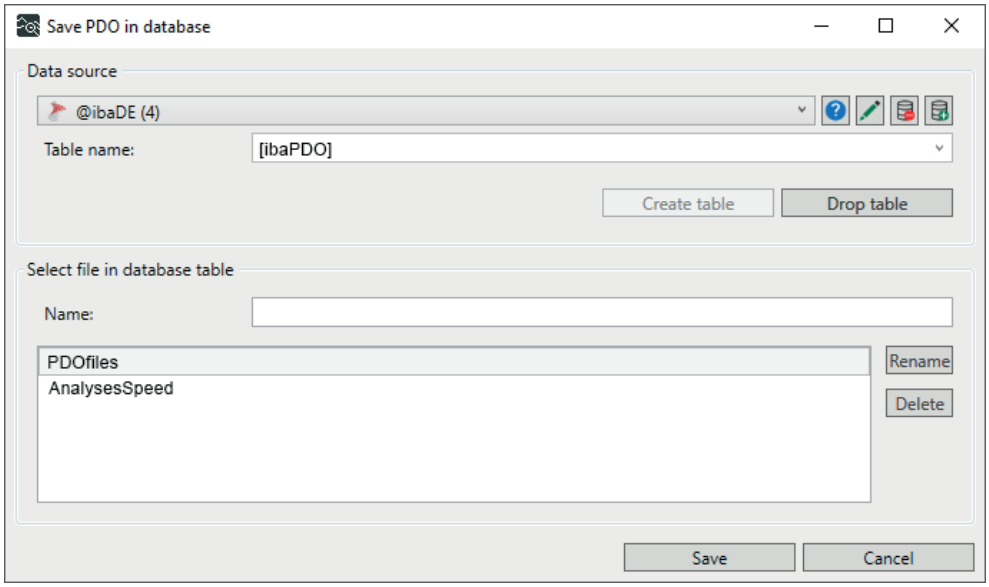
In *ibaAnalyzer*, you can save analysis files not only within the file system but also in a database. You can load those analyses from the database and open them in *ibaAnalyzer*, see ↗ *Opening an analysis, page 39*.

Prerequisite: You have configured a database connection in *ibaAnalyzer*.

For more information on database connections, see *ibaAnalyzer* manual part 4, chapter *Configuring the database connection*.

Procedure

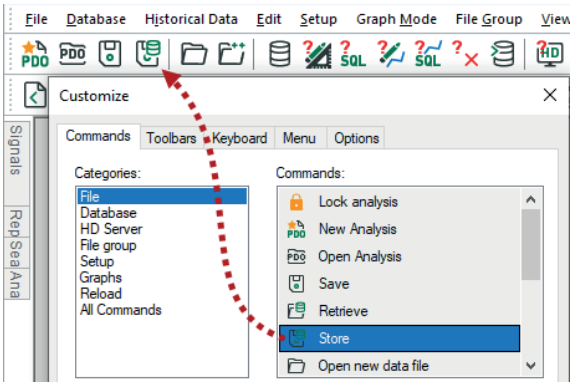
1. Go to *File – Store analysis in database*.
2. In the dialog, select the database as the data source and the table.



3. Enter a name for the file.
4. To save the selected file, click <Save>.

Tip

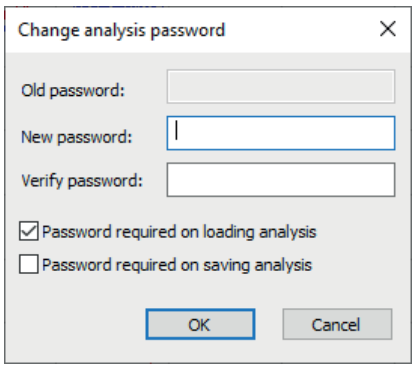
You can place the menu command for saving the analysis into a database as a button on the toolbar. To do this, in the *View – Toolbar Setup* menu simply drag the relevant button from the dialog into the toolbar.



3.4 Password protection for analyses

You can protect analyses with a password for various purposes. You can prevent unauthorized access to the analysis, or you can protect against saving of the analysis so as to prevent accidental or unauthorized changes.

Open the password input dialog via the *File – Analysis password protection* menu.



The dialog contains the following items:

Old password
If a password has already been assigned, enter it here in order to change it. If you do not enter the old password, the dialog closes when you click <OK>, without applying any change.

New password
Enter the new password here if you are assigning one for the first time or want to change an existing one. If you leave the field empty, but an old password exists, the password will be deleted and password protection is thus disabled.

Verify password
Enter the exact text here as in the *New password* field. If the content of both fields does not match, an error is displayed when closing the dialog, and the dialog remains open.

Password required on loading analysis
If you enable this option, the password is requested to open an analysis.

Password required on saving analysis
If you enable this option, the password is requested before the analysis can be saved.

At least one of the two options must be enabled for you to close the dialog with <OK>.

Procedures

First time password entry

1. Enter a new password in the *New password* box.
2. Enter it again in the new password in the *Verify password* box.
3. Confirm with <OK>.

Changing a password

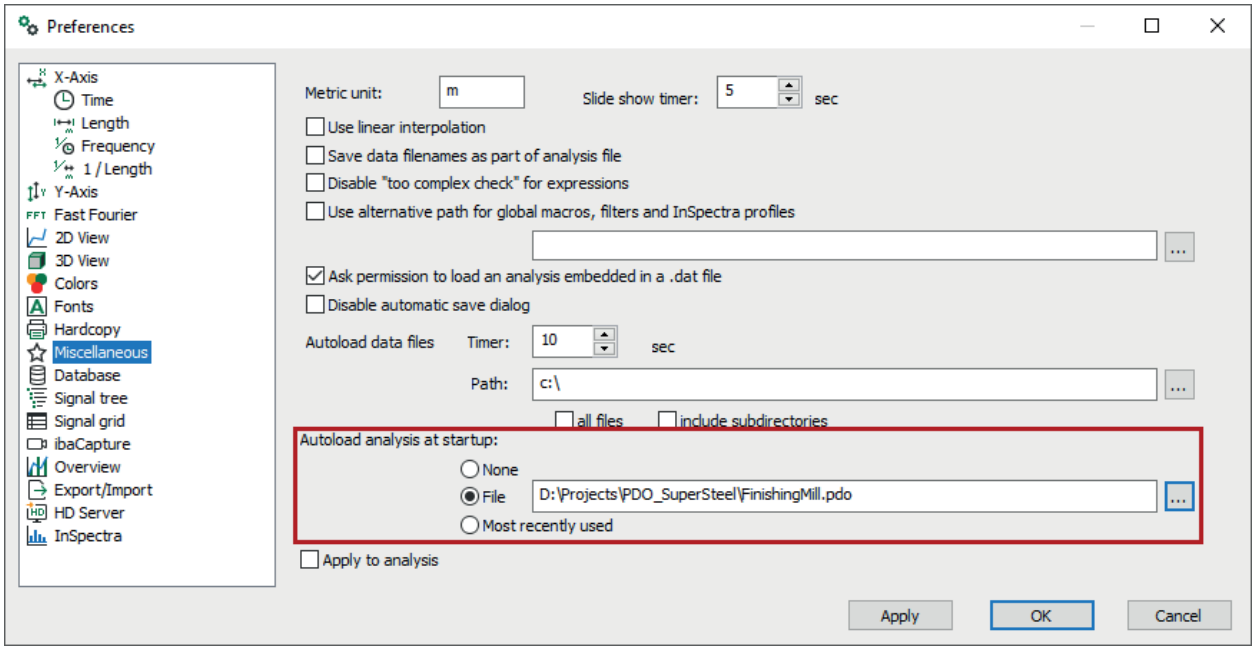
1. Enter the old password in the *Old password* box.
2. Enter a new password in the *New password* box.
3. Enter it again in the new password in the *Verify password* box.
4. Confirm with <OK>.

Removing a password

1. Enter the old password in the *Old password* box.
2. Leave the *New password* and *Verify password* boxes empty.
3. Confirm with <OK>.

3.5 Default analysis file

In order to open automatically a particular analysis file when starting *ibaAnalyzer*, enter the analysis file name in the *Preferences* dialog under *Miscellaneous*.



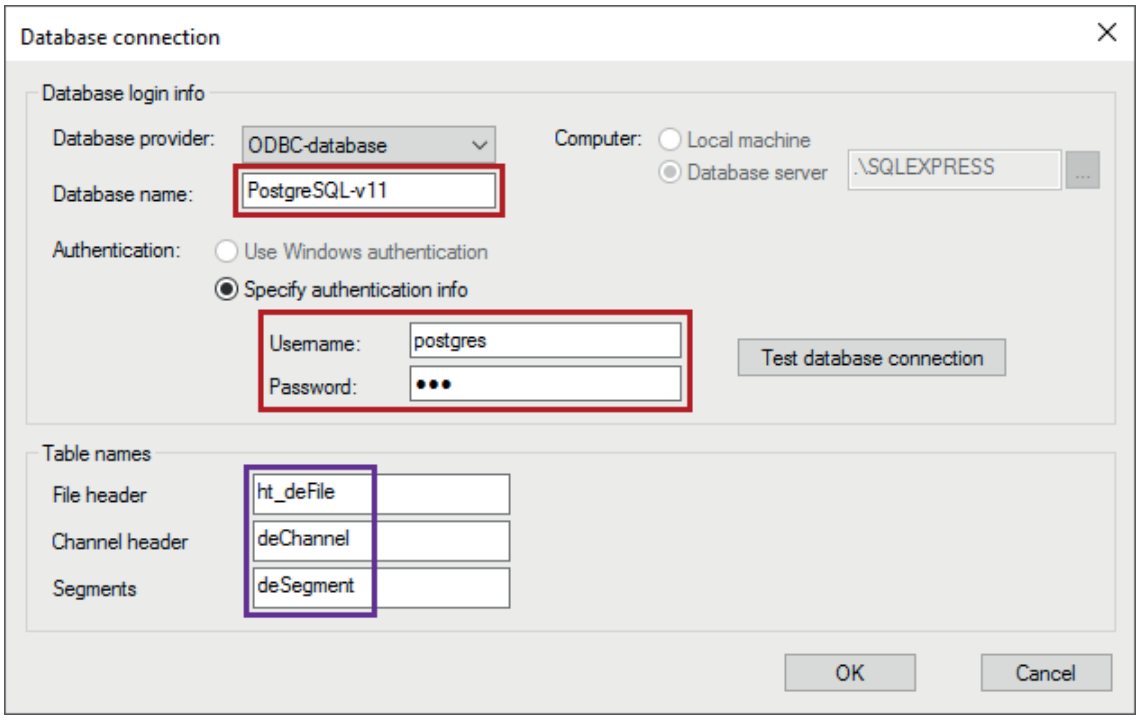
3.6 Changing the connection to ODBC database

Note

ibaAnalyzer v8.0.0 or higher no longer supports any database connections via ODBC. Databases that have been configured in older PDO files via ODBC must be converted to the actual database provider in *ibaAnalyzer* and directly connected to the database.

If you open an older PDO file of *ibaAnalyzer-v7*, in which an ODBC database connection has been configured, a message is shown that this connection is no longer supported. Therefore, you have to change the database connection in the PDO file. You make these changes for each PDO file with an ODBC database connection.

Database settings via ODBC in *ibaAnalyzer-v7* for comparison.

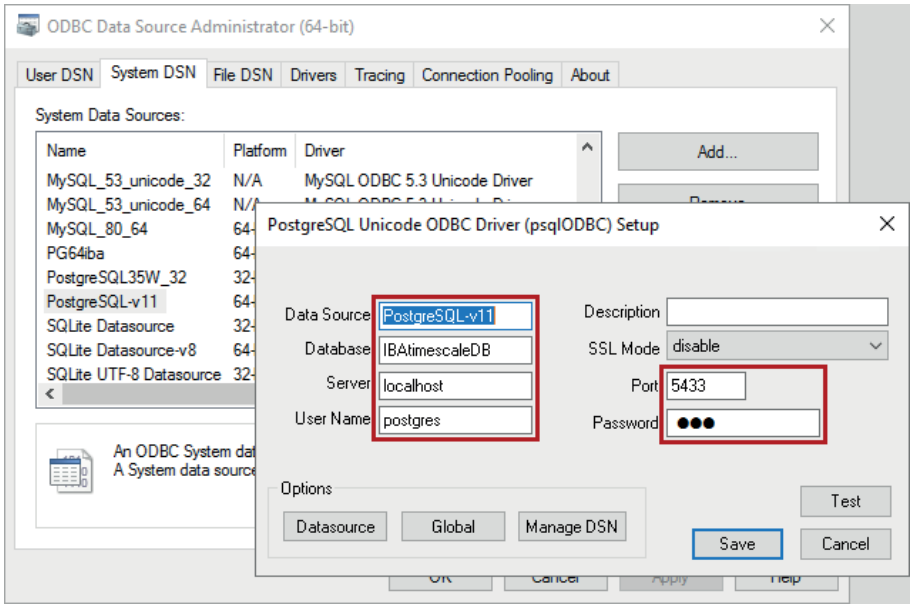


Customizing the database connection of PDO files with ODBC database connections

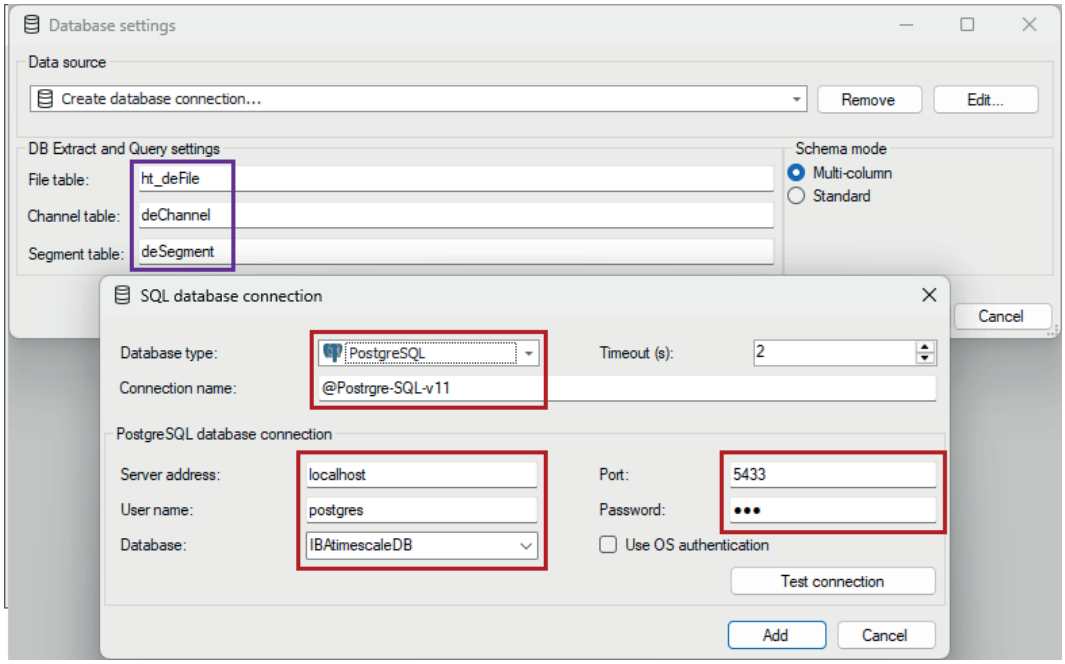
1. Open the PDO file from *ibaAnalyzer-v7* in *ibaAnalyzer* v8.1.0 or higher.
2. Open the *Database settings*.
3. From the *Data source* drop-down list, select *Create new connection*.

4. Configure the new connection with the settings for the original database.

→ To view the specific settings of the connection, you can open the ODBC settings under Windows.



→ *ibaAnalyzer* automatically applies the table names.



5. Apply the database settings and save the PDO file.

Further information on the settings for the specific databases can be found in the respective chapters in part 4 of the *ibaAnalyzer* manual:

- MySQL/MariaDB
- PostgreSQL
- SQLite

4 Quick access to analyses and more

In the *Analysis files* tab of the signal tree window, you can configure a tree structure with any number of analysis files. You can apply these analysis files on the currently loaded data file by double-clicking. In this way, you can analyze a data set under different aspects without having to constantly open and close analysis files.

In addition, you can include links (shortcuts) to signals, expressions and X-axis markers in the tree. You can even assign one or more data files to a group.

The following chapters describe the procedure for setting up the tree elements and the various elements.

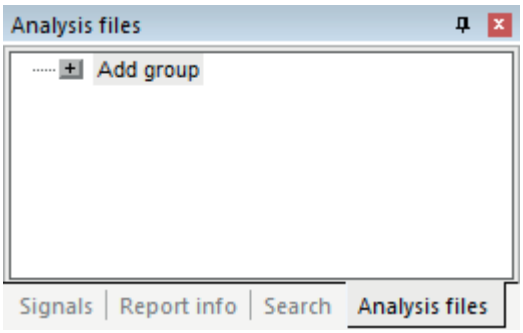
Note



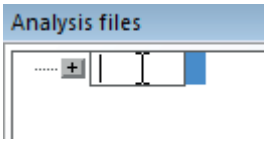
The analysis tree is NOT stored in an analysis file but in the registry (like the preferences). Therefore, the analysis tree is also available if *ibaAnalyzer* is started without an analysis. To empty the analysis tree, you have to delete the elements it contains.

4.1 Creating new analysis tree

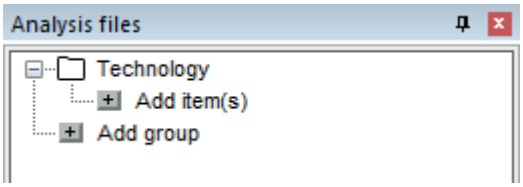
1. Create and save the analysis (PDO files) you are using.
2. Select the *Analysis files* tab in the signal tree window.



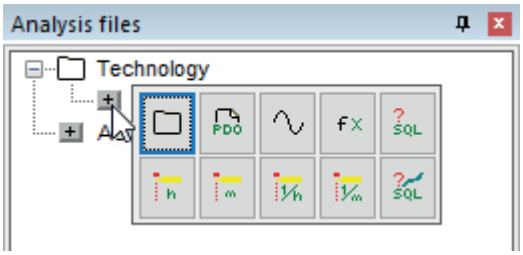
3. Add a group using the <+> button, and enter a name for the group.



→ The group is included in the analysis tree.
You can add items to the group, or create more groups.



4. Select the desired items using the <+> button in front of *Add item(s)*.



→ You can add the following items. More information can be found in the relevant chapters.

Item	Description	More information
	Subgroup	↗ *Groups and subgroups in the analysis tree, page 49*
	Analysis (PDO file)	↗ *Analysis (PDO files) in the analysis trees, page 51*
	Signal	↗ *Signal shortcuts, page 52*
	Expression	↗ *Expression shortcuts, page 54*

Item	Description	More information
	X-axis marker time based	↗ *Marker shortcuts, page 56*
	X-axis marker length based	
	X-axis marker frequency based	
	X-axis marker 1/length based	
	SQL query	↗ *SQL query, page 58*
	SQL trend query	↗ *SQL trend query, page 58*

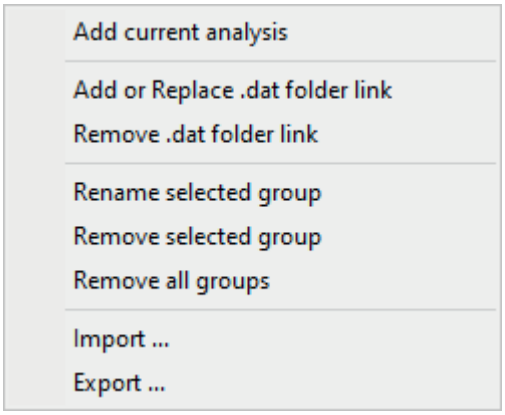
4.2 Groups and subgroups in the analysis tree

As well as adding items to a group, you can also add subgroups, which in turn contain items. The number of groups is unlimited.

Once you have created a group, add a subgroup to the main group, see ↗ *Creating new analysis tree, page 48*.

You can treat subgroups the same way as all groups. You can move the groups and subgroups within the analysis tree using drag & drop to arrange them differently.

Using the context menu that opens when you right-click on a group, you can rename or remove a selected group, remove all groups, or assign the current open analysis to that group. You can also link to a .dat folder.



Groups with .dat folder link

If you add a .dat folder link to a group, you can access and analyze the data files in the folder directly via the group. If new data files are added to the folder, these are also available.

To do this, select *Add or replace .dat folder link* from the context menu.

Then a browser window opens for selecting the desired path.

This window has the following functions:

Also search for new .dat files in subdirectories
If you enable this option, *ibaAnalyzer* regularly checks the selected path and its subdirectories for new data files.

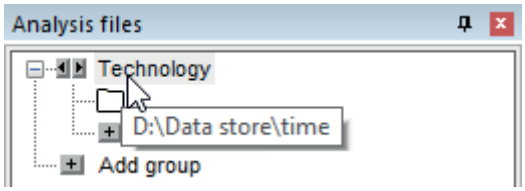
Preferred folder list
If you have already defined preferred folders for storing the DAT files in the *Open data file* dialog in *ibaAnalyzer*, you can select a preferred folder here.

<Make new folder>
Create a new folder if necessary.

Confirm your settings with <OK>.

Using the .dat folder link

If a group in the analysis tree is linked to a .dat folder, the group icon is replaced by two gray arrows. When you move the mouse over the group name, the tooltip shows the folder path.



When you click on one of the arrows for the first time, *ibaAnalyzer* opens the most recent data file in the relevant directory, if there is a data file in it.

Use the arrows to switch between the data files: left to older files, right to newer files. The files are replaced, and not attached or grouped.

If you open a data file in a different way (e.g. with *Open data file*), clicking on one of the arrows will take you back to the most recent file.

To append new files to the currently open file, hold down the <Alt> key while clicking on the arrow buttons. A click on the right arrow button will append a younger data file to the end of the current file and thus at the end of the chain of appended files. A click on the left arrow button appends the file which is older than the current first file in the chain before that file, i.e. it puts the new file at first position of the chain. This guarantees that the data files are displayed in correct chronological order.

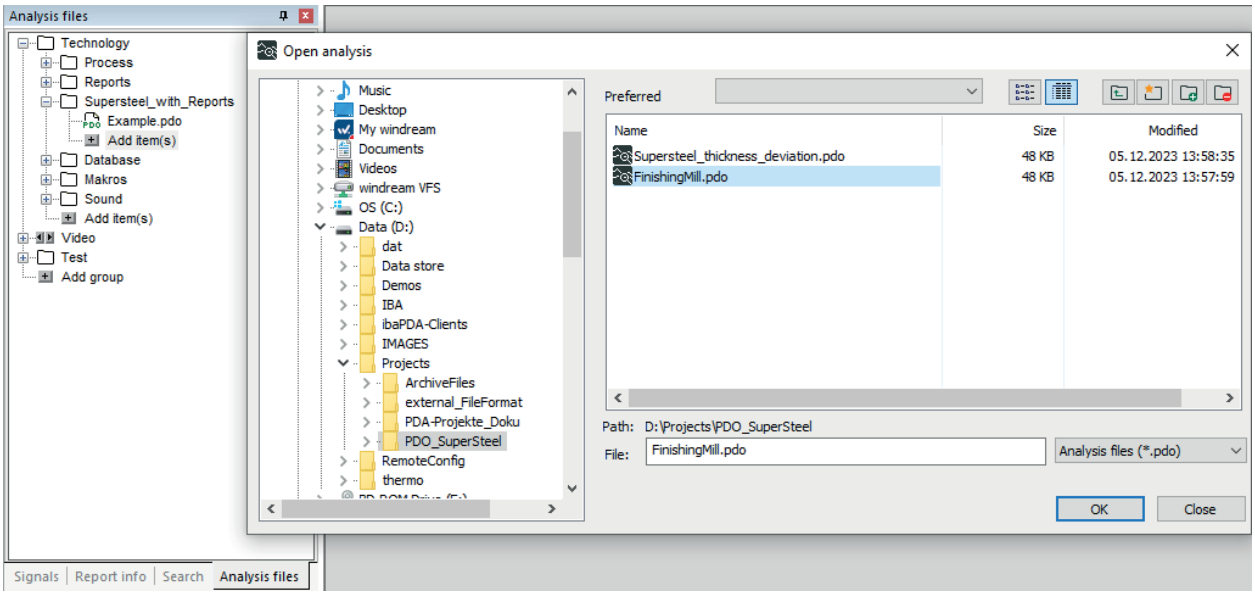
Instead of using the mouse for these operations, you can use the arrow keys together with the <Ctrl> key, provided the group node is marked. The appending of files works respectively with the key combinations <Alt>+<Ctrl>+ <left>/<right>.

4.3 Analysis (PDO files) in the analysis trees

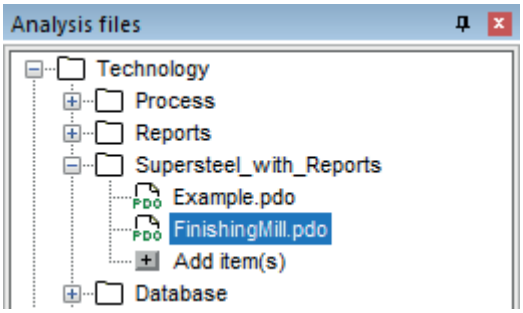
If you store multiple analyses in the analysis tree, you can analyze a data set in different aspects without continually opening and closing analyses.

Once you have created a group, add an analysis to the group as an item, see ↗ *Creating new analysis tree, page 48*.

Select one or more analysis files in the file browser.



The selected analysis files appear in the tree right beneath the group or subgroup.



To open the analysis, double-click on it in the analysis tree.

You can use the context menu for removing an analysis file from the tree (*Remove selected file*).

You can move the analysis within the analysis tree using drag & drop to assign it to other groups.

Ideally, group your analyses in a way that suits best your requirements. These are some popular grouping methods:

- Technical grouping according to the process or plant unit (e.g. infeed section, cleaning, furnace, skin pass mill, outfeed section)
- Higher-level grouping (e.g. technology, production, statistics, maintenance, etc.)
- Personal analysis preferences where multiple people are using the same computer for different analyses.

Note

The analysis tree is NOT stored in an analysis file but in the registry (like the preferences). Therefore, the analysis tree is also available if *ibaAnalyzer* is started without an analysis.

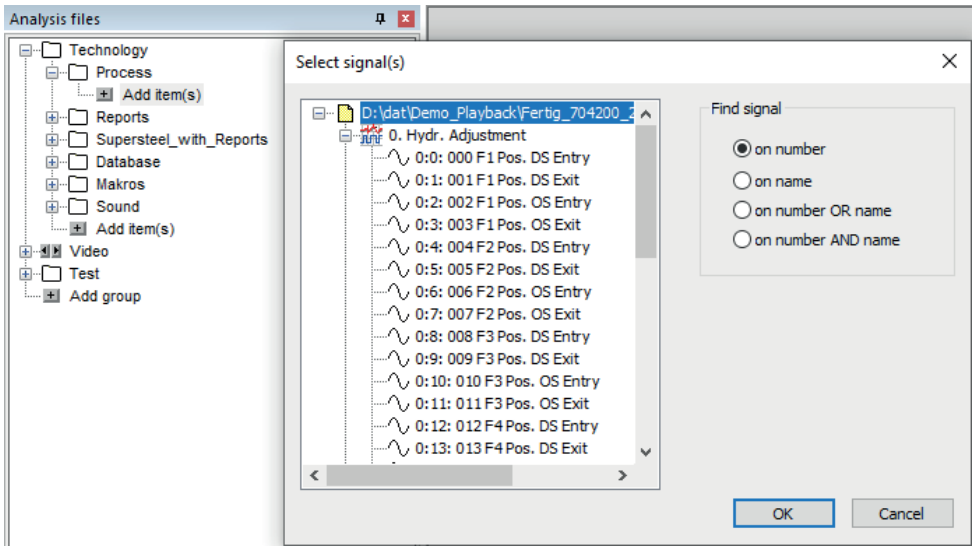
4.4 Signal shortcuts

You can create shortcut links to preferred signals in each group or subgroup. Using those shortcuts, you can then drag and drop the signals into the recorder window as in the normal signal tree or open them by double-clicking. Signal shortcuts make switching between the signal and analysis tree unnecessary.

To be able to add signal shortcuts, a corresponding data file must be open.

Once you have created a group, add a signal to the group as an item, see ↗ *Creating new analysis tree, page 48*.

Select one or more signals in the dialog.



The following options are available for finding the signals:

on number
A signal of a currently open data file can always be displayed via this signal shortcut if it has the same number (i.e., file number, module number and signal number).

on name
A signal of a currently open data file can always be displayed via this signal shortcut if it has the same name. The first signal with this name is displayed (if multiple signals have the same name).

on number OR name
A signal of a currently open data file can always be displayed via this signal shortcut if it has either the same number or the same name. The first signal matching one of these conditions will be displayed.

on number AND name
A signal of a currently open data file can only be displayed via this signal shortcut if it has both the same number and the same name.

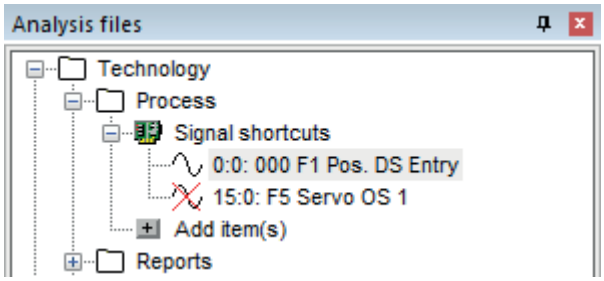
Confirm your settings with <OK>.

Another way to add a signal shortcut is to drag & drop a signal from the recorder window onto a group or a *Signal shortcut* node in the analysis tree.

Signal shortcuts in the analysis tree

Signal shortcuts have been created in the analysis tree.

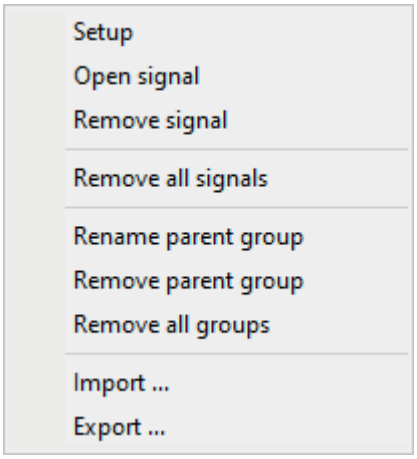
If a signal the shortcut refers to is not available in the data file, it will be indicated by a red cross on the shortcut icon. This occurs, for example, if the currently open data file does not contain the same signals as the file with which the signal shortcut was created.



You can move the signal shortcuts within the analysis tree using drag & drop to assign them to other groups.

Signal shortcuts context menu

Right-clicking on a signal shortcut opens a context menu.



Setup
This command opens the signal browser just like when adding a signal shortcut. You can select a different signal or change the option for finding and opening a signal.

Open signal
This command displays the signal in the recorder window.

Remove signal
This command removes the signal shortcut from the group. If it is the only signal, the *Signal shortcuts* node will also be removed from the group.

Remove all signals
This command removes all signal shortcuts and the *Signal shortcuts* node from the group.

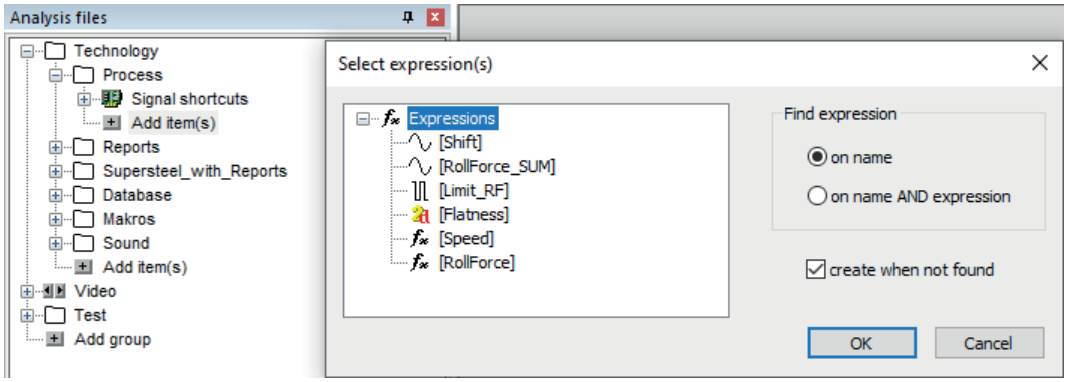
4.5 Expression shortcuts

You can create shortcut links to preferred expressions in each group or subgroup. An expression shortcut can link to either an expression that was created in the signal table or a logical expression.

To be able to add expression shortcuts, the opened analysis must contain logical expressions. For more information, see ↗ *Logical expressions, page 172*.

Once you have created a group, add an expression to the group as an item, see ↗ *Creating new analysis tree, page 48*.

Select one or more expressions in the dialog.



The following options are available for finding the expressions:

on name
An expression in a currently open analysis file can always be displayed via this shortcut if it has the same name. The first expression with this name is displayed (if multiple expressions have the same name).

on name AND expression
An expression in a currently open analysis file can only be displayed using the shortcut if both its name and the expression itself match.

create when not found
If you enable this option, an expression already contained in the analysis tree will be automatically generated in a current analysis file (in the signal table). This can occur, for example, if another analysis file has been loaded that does not yet contain the expression in question. This ensures that an expression shortcut can always be opened.

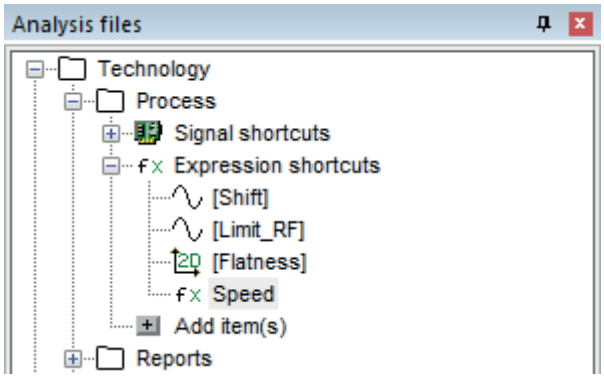
Confirm your settings with <OK>.

Another way to add an expression shortcut is to drag & drop an expression from the recorder window onto a group or an *Expression shortcuts* node in the analysis tree.

Expression shortcuts in the analysis tree

Expression shortcuts have been created in the analysis tree.

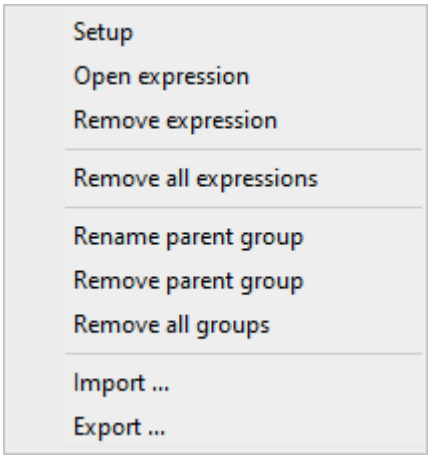
If an expression the shortcut refers to is not available in the analysis file, it will be indicated by a red cross on the shortcut icon. This occurs, for example, if the currently open analysis file does not contain the expressions for which shortcuts already exist.



You can move the expression shortcuts within the analysis tree using drag & drop to assign them to other groups.

Expression shortcuts context menu

Right-clicking on an expression shortcut opens a context menu.



Setup

This command opens the expression browser just like when adding an expression shortcut. You can then select a different expression or change the option for finding and opening an expression.

Open expression

This command displays the expression in the recorder window. Opening an expression will not necessarily open a new graph in the recorder window. It may be displayed in a graph with other signals, depending on where it was configured.

Remove expression

This command removes the expression shortcut from the group. If it is the only expression, the *Expression shortcuts* node will also be removed from the group.

Remove all expressions

This command removes all expression shortcuts and the *Expression shortcuts* node from the group.

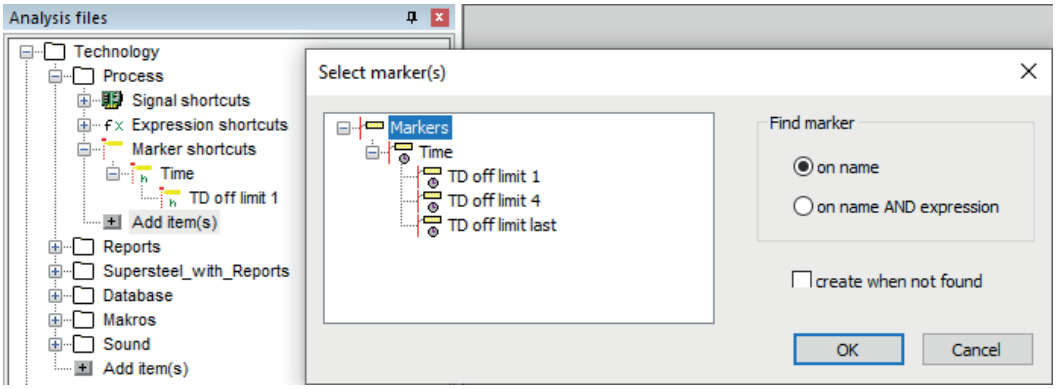
4.6 Marker shortcuts

You can create shortcut links to preferred markers in each group or subgroup. Using those shortcuts, you can then drag and drop the markers into the recorder window as in the normal signal tree or open them by double-clicking.

To be able to add marker shortcuts, the opened analysis must contain markers. For more information, see ↗ *Markers, page 119*.

Once you have created a group, add a marker (time, length, frequency or inverse length) to the group as items, see ↗ *Creating new analysis tree, page 48*.

Select one or more markers in the dialog.



The following options are available for finding the markers:

on name
A marker in a currently open analysis file can always be displayed via this shortcut if it has the same name. The first marker with this name is displayed (if multiple markers have the same name).

on name AND expression
A marker in a currently open analysis file can only be displayed using the shortcut if both its name and the expression that defines the marker match.

create when not found
If you enable this option, a marker already contained in the analysis tree will be automatically generated in a current analysis file (in the marker table). This can occur, for example, if another analysis file has been loaded that does not yet contain the marker in question. This ensures that a marker shortcut can always be opened.

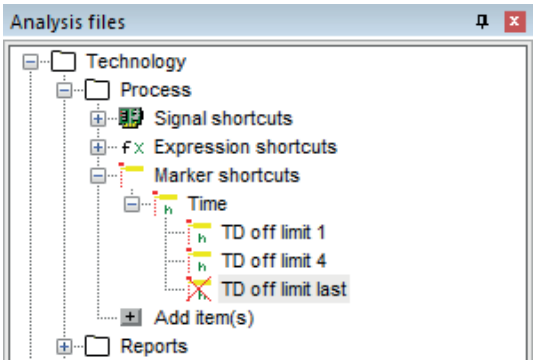
Confirm your settings with <OK>.

Another way to add a marker shortcut is to drag & drop a marker from the recorder window onto a group or an *Expression shortcuts* node in the analysis tree.

Markers in the analysis tree

Marker shortcuts have been created in the analysis tree. The marker shortcuts are automatically grouped by type.

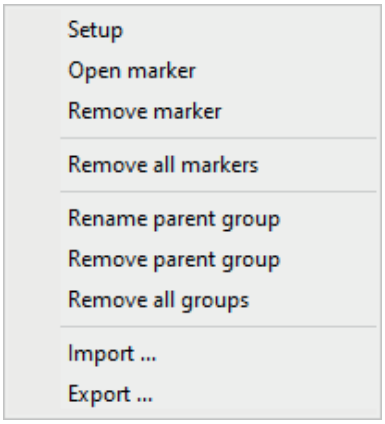
If a marker the shortcut refers to is not available in the analysis file, it will be indicated by a red cross on the shortcut icon. This occurs, for example, if the currently open analysis file does not contain the markers for which shortcuts already exist.



You can move the marker shortcuts within the analysis tree using drag & drop to assign them to other groups.

Marker shortcuts context menu

Right-clicking on a marker shortcut opens a context menu.



Setup
This command opens the marker browser just like when adding a marker shortcut. You can then select a different marker or change the option for finding and opening a marker.

Open marker
This command displays the marker in the recorder window. Markers are not only opened in a new graph, but are also displayed in the graphs of other signals (e.g. frequency markers in FFT views). Therefore, it is recommended to drag and drop the desired marker from the analysis tree into an appropriate graph.

Remove marker
This command removes the marker shortcut from the group. If it is the only marker, the *Marker shortcuts* node will also be removed from the group.

Remove all markers
This command removes all marker shortcuts and the *Marker shortcuts* node from the group.

4.7 SQL query

An SQL query is designed for retrieving and showing measurement and analysis data previously extracted to a database.

You can use these grouping elements to easily execute different SQL queries you previously created.

Once you have created a group, add an SQL query to the group as an item, see ↗ *Creating new analysis tree, page 48*.

For the configuration, the same dialog opens as from the database toolbar. Here you can either enter an existing SQL query file or enter the SQL statements directly.

Confirm your settings with <OK>. The SQL query has then been created in the analysis tree.

To execute the SQL query, double-click on it in the analysis tree.

For detailed information on database queries, see *ibaAnalyzer* manual part 4, chapter *SQL queries*.

4.8 SQL trend query

An SQL trend query is designed for finding corresponding database entries based on certain conditions and showing selected characteristic values of those database entries as trends in the *Overview* tab (Signal grid area). From this graph, the complete data extractions (measurement and analysis data) can be specifically retrieved and displayed.

You can use these grouping elements to easily execute different SQL trend queries you previously created.

Once you have created a group, add an SQL trend query to the group as an item, see ↗ *Creating new analysis tree, page 48*.

For the configuration, the same dialog opens as from the database toolbar. Here you can either enter an existing SQL trend query file or enter the SQL statements directly.

Confirm your settings with <OK>. The SQL trend query has then been created in the analysis tree.

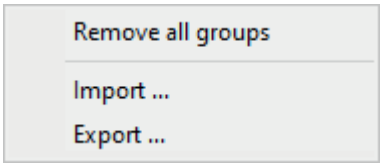
To execute the SQL trend query, double-click on it in the analysis tree.

For detailed information on database queries, see *ibaAnalyzer* manual part 4, chapter *SQL trend queries*.

4.9 Export and import of analysis trees

You can export and re-import the analysis trees you create. For the purpose, *ibaAnalyzer* generates a text file, which you can also edit using text editors or MS Excel.

For export or import, open the context menu in the analysis tree and select *Import* or *Export*. These functions are always available, even if the analysis tree is still empty.



The import and export functions are upward and downward compatible. You can import files exported from an older *ibaAnalyzer* version into a newer version at any time.

When importing analysis trees created with a newer version, the following restrictions apply:

- Versions <5.8: Subgroups are ignored.
- Versions <5.1: Links to signals, expressions and markers are ignored.

5 Settings

There is generally little difference between the dialog window for the preferences and for the graph setup. They differ with respect to the generally valid and specific settings.

Preferences
You can open the dialog for the preferences via the menu *Setup – Preferences* or using the gears button in the toolbar.



The preferences determine the form of presentation when a new analysis is created or when a new graph is opened.

The preferences are not saved in the analysis, but in an initialization file of *ibaAnalyzer* and are therefore independent of an analysis file.

Note



A change in the preferences only affects the currently displayed graphs if you enable the *Apply to analysis* option (in the dialog in the lower left corner) before applying the change.

Graph setup
Open the dialog for the graph setup via the menu *Setup – Graph setup* or in the context menu of a graph under *Setup*.

In contrast to the preferences, the graph setup only applies to the trend currently active graph (i.e. the selected graph) or to the graph in which the context menu was opened. The *Graph set-up* dialog offers only the possibilities of the preferences that are relevant for the active graph. The dialog boxes differ between *Preferences* and *Graph setup*. Thus, e.g. the X-axis settings of the graph in the *Time*, *Length*, *Frequency* and *1/length* nodes also show the markers which is not the case with the preferences.

To apply a change directly to the active graph, click the <Apply> button. A change in the graph setup only affects the preferences if you enable the *Apply to preferences* option (in the dialog in the lower left corner) before applying the change.

The graph setup are saved in the analysis.

Note



The regional and language settings under Windows determine the formatting of numerical data such as time, date, etc., for example on the time axis, in tables or in the export dialog.

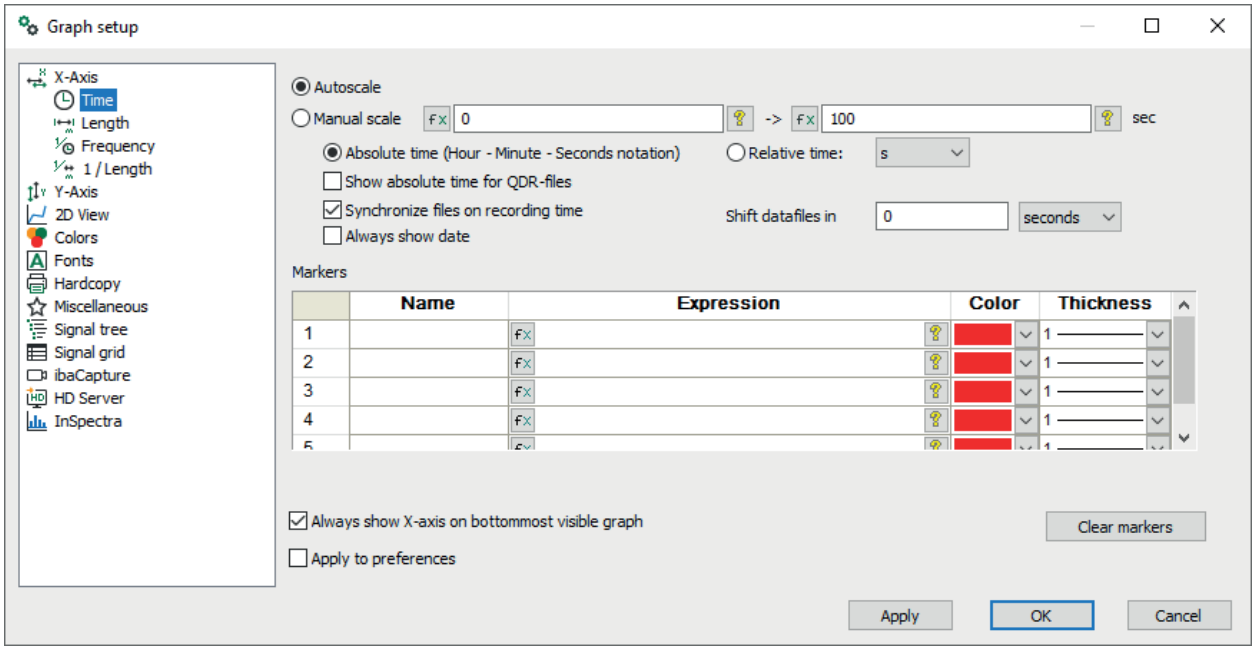
5.1 X-axis

The X-axis settings depend on the X-axis mode. You can change the X-axis mode if, for example, you want to analyze length-based signals or a frequency; see ↗ *X-axis modes (reference axes), page 129*. By default, the X-axis is time-based.

You can also access the X-axis settings via the X-axis context menu.

5.1.1 X-axis – Time

If the X-axis is in *Time* mode, the following settings are available.



Autoscale
Selected by default. Autoscaling automatically adjusts the X-axis to the duration of the data file. If an additional signal with a longer recording time is opened, the time scale is adapted to that longer signal. In an analysis, there is only one common time axis for all time-based signals.

Manual scale
With manual scaling, you can set fixed start values and end values for the scale. You can also use expressions to define these values dynamically, for example depending on certain process parameters. The <fx> button opens the expression builder. Regardless of the signal duration, only the defined section is displayed. Manual scaling is indicated by a hand symbol at the origin of the scale.



Absolute time (hours-minutes-seconds)
This option switches the scale labeling. The time values on the scale are displayed in hours:minutes:seconds.

Show absolute time for QDR files
Only applicable to data files from *ibaQDR* systems. Time and length references are stored in these files. Normally, the signals in the time based view are scaled to the entire X-axis, and so "stretched" to the total runtime of the strip in the plant unit. So a time trend is visible, but the assignment of the Y-values to the time scale is not correct.

This option corrects the signal trend over time on the X-axis. This makes it easy to see when, and for how long, a measurement signal was recorded for the strip in question.

Synchronize files on recording time
This option is important for representing appended data files. Only if you enable this option will the signals be arranged on the time axis according to the recording date of the data file. See ↗ *Appending data files*, page 23.

Always show date
The *Absolute Time* option enables you to additionally display the date on the scale. This is useful for data files that span multiple days or exceed a date limit.

Relative time
This option switches the scale labeling. The scale starts at 0, and shows the other scale values as a distance from it. For the relative time reference, you can select the display in seconds, minutes:seconds or hours:minutes:seconds.

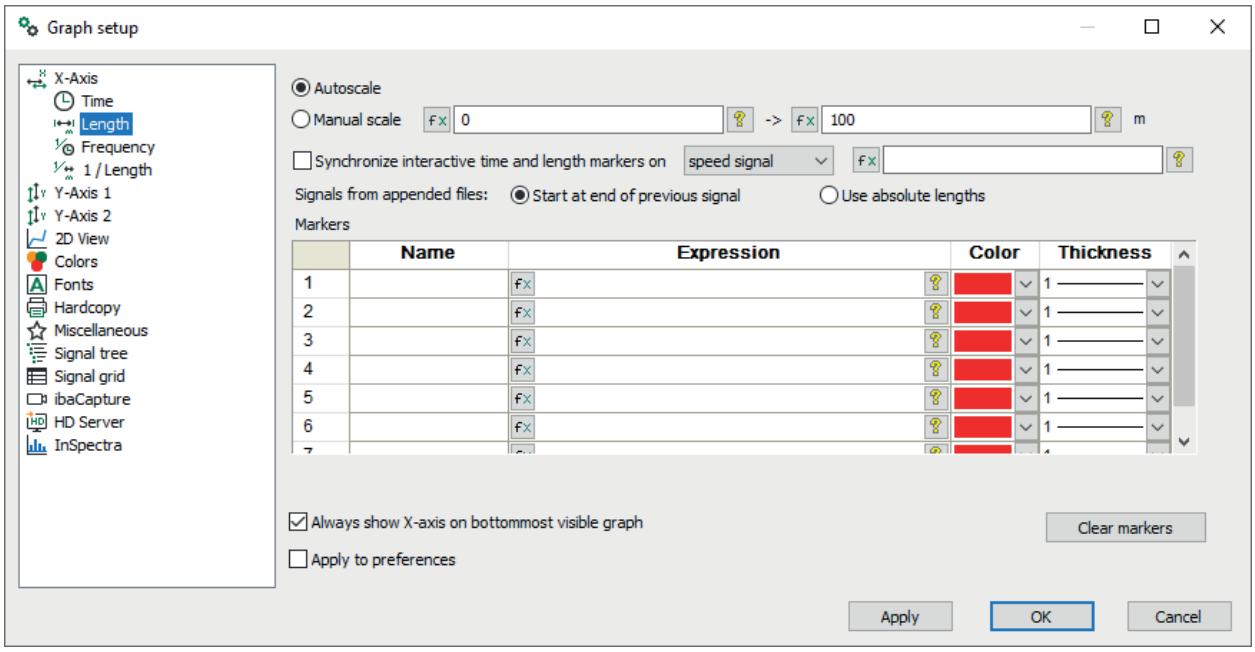
Shift data files in … time
You can specify a collective time shift. The shift then affects all open data files equally. See ↗ *Time shift of data files*, page 34.

Markers table
The *Markers* table shows all currently defined X-axis markers. You can define or delete markers here. This table contains the same markers that you can also access by selecting *Markers* in the context menu of a graph. You decide subsequently whether a marker is displayed in the graph in question by dragging it from the signal tree into the graph. See ↗ *X-axis markers (calculated markers), page 123*.

Always show X-axis on bottom-most visible graph
If more graphs are open than can fit in the display area, a scroll bar is shown at the right-hand margin. If you do not enable this option, the scale (X-axis) remains on the bottom-most graph and is no longer visible when you scroll the view upward. If you enable this option, the scale is always automatically positioned below the lower graph, which is still fully visible in the recorder window.

5.1.2 X-axis – Length

If the X-axis is in *Length-based* mode, the following settings are available.



Autoscale
Selected by default. Autoscaling automatically adjusts the X-axis to the length of the data file. If an additional signal with a greater recording length is opened, the scale is adapted to that longer signal. In an analysis, there is only one common length axis for all length-based signals.

Manual scale
With manual scaling, you can set fixed start values and end values for the scale. You can also use expressions to define these values dynamically, for example depending on certain process parameters. The <fx> button opens the expression builder. Regardless of the signal duration, only the defined section is displayed. Manual scaling is indicated by a hand symbol at the origin of the scale.



Synchronize interactive time and length markers on …
If you enable this option, the markers of time-based and length based signals will be synchronized. This is useful if you want to display time-based and length based signals simultaneously and determine the corresponding length value at a specific point in time – or vice versa. This may be necessary, for example, when analyzing video signals in order to obtain both the time reference and the length reference for certain events. The length-time reference is created for the marker depending on which graph is active.
You must select either a speed signal or a position signal as the synchronization signal from the drop-down list.
If you select *Position signal*, enter a position signal that delivers a measured length value in the adjacent field. If you do not have a position signal, select *Speed signal* and enter the name of a speed signal. *ibaAnalyzer* automatically converts time and length, similar to the *TimeToLength* or *TimeToLengthL* functions, and positions the markers accordingly.
Negative speed signals and invalid or unassigned position signals are ignored.

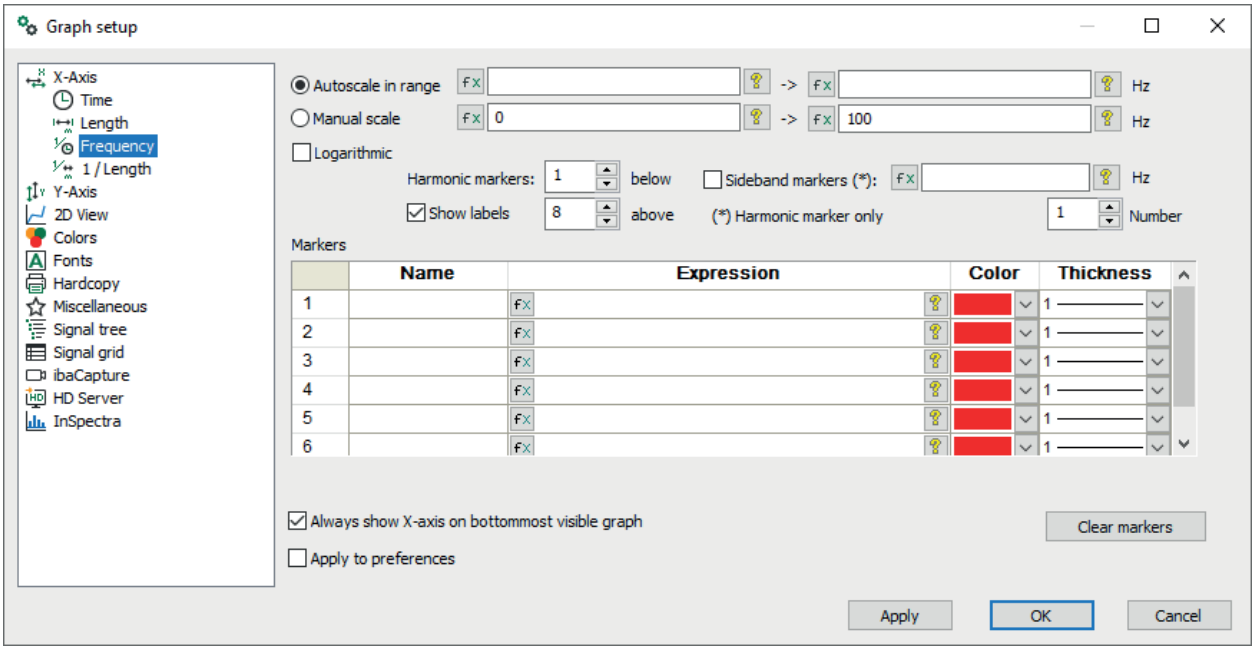
Signals from appended files …
Select whether, when data files are appended together, the length based measured values in the trend graphs are displayed directly one after the other or according to the length value in the trend graph. The second option corresponds to the *Synchronize files on recording time* setting for time-based signals.

Markers table
The *Markers* table shows all currently defined X-axis markers. You can define or delete markers here. This table contains the same markers that you can also access by selecting *Markers* in the context menu of a graph. You decide subsequently whether a marker is displayed in the graph in question by dragging it from the signal tree into the graph. See ↗ *X-axis markers (calculated markers), page 123*.

Always show X-axis on bottom-most visible graph
If more graphs are open than can fit in the display area, a scroll bar is shown at the right-hand margin. If you do not enable this option, the scale (X-axis) remains on the bottom-most graph and is no longer visible when you scroll the view upward. If you enable this option, the scale is always automatically positioned below the lower graph, which is still fully visible in the recorder window.

5.1.3 X-axis – Frequency

If the X-axis is in *Fast Fourier Transformation (Time)* mode, the following settings are available.

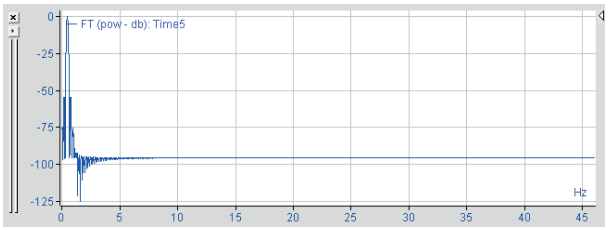
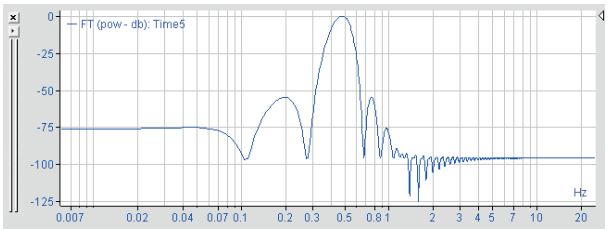


Autoscale in range
Selected by default. Autoscaling automatically adjusts the X-axis to the data file. You can also specify an upper and lower limit for the scaling of the frequency axis (for the FFT display). This is useful because a specific frequency range is usually of interest when using FFT representation.

Manual scale
With manual scaling, you can set fixed start values and end values for the scale. You can also use expressions to define these values dynamically, for example depending on certain process parameters. The <fx> button opens the expression builder. Regardless of the signal duration, only the defined section is displayed. Manual scaling is indicated by a hand symbol at the origin of the scale.



Logarithmic
If you enable this option, the X-axis is scaled logarithmically instead of in linear mode. This is helpful for FFT views across wide frequency ranges. The following images show the difference.

Linear frequency axis	Logarithmic frequency axis
	

Harmonic markers ... below/above
Specify the number of harmonic markers above or below the main frequency to be displayed in the graph (FFT).

See ↗ *Harmonic markers, page 120*.

Show labels
Enable the flags to display the frequency values of the harmonic markers.

Sideband markers
Enable this option if you also want to display the sidebands around the main frequency. You can additionally enter an expression for configuration of the sideband markers, and specify the number of sideband markers to be displayed.

See ↗ *Markers, page 119*.

Markers table
The *Markers* table shows all currently defined X-axis markers. You can define or delete markers here. This table contains the same markers that you can also access by selecting *Markers* in the context menu of a graph. You decide subsequently whether a marker is displayed in the graph in question by dragging it from the signal tree into the graph. See ↗ *X-axis markers (calculated markers), page 123*.

Always show X-axis on bottom-most visible graph
If more graphs are open than can fit in the display area, a scroll bar is shown at the right-hand margin. If you do not enable this option, the scale (X-axis) remains on the bottom-most graph and is no longer visible when you scroll the view upward. If you enable this option, the scale is always automatically positioned below the lower graph, which is still fully visible in the recorder window.

5.1.4 X-axis – 1/Length

The settings correspond to those for to *Frequency*, see ↗ *X-axis – Frequency, page 64*.

5.2 Y-Axis

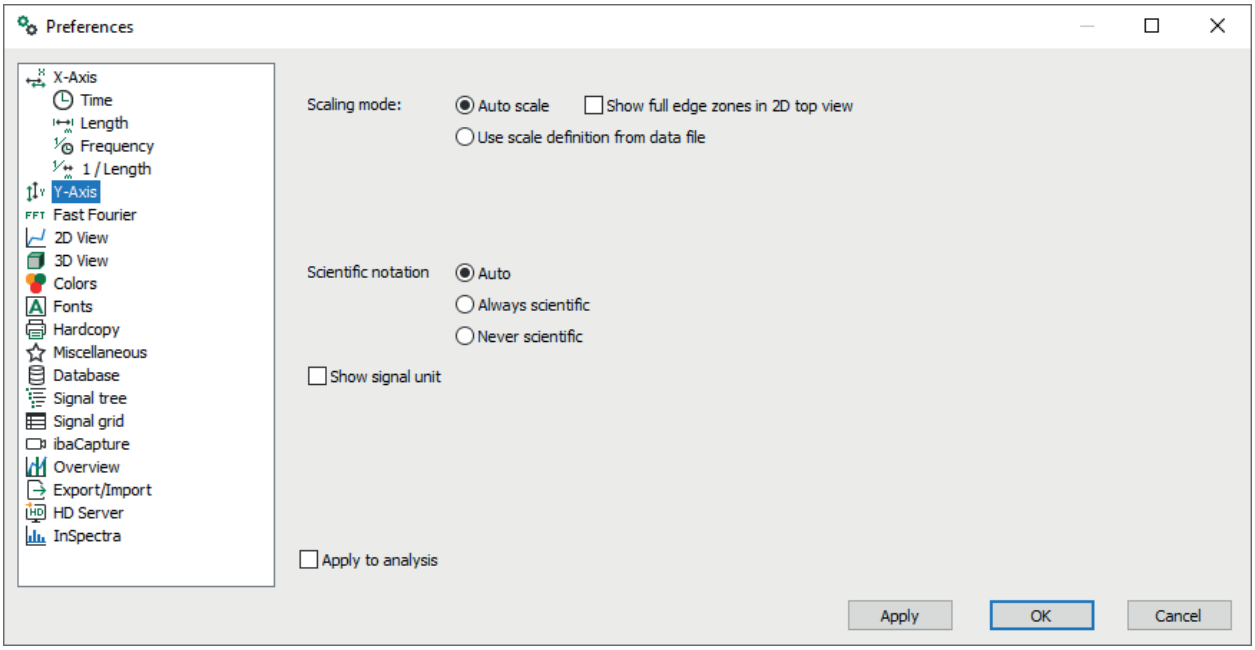
The Y-axis settings differ significantly between *Preferences* and *Graph setup*.

The preferences only offer basic options that are independent of the data file, see ↗ *Y-axis in Preferences, page 66*.

The graph setup provides more configuration options, as *ibaAnalyzer* has access to more information for individual graphs, see ↗ *Y-axis in Graph setup, page 68*.

5.2.1 Y-axis in Preferences

In the *Preferences*, you can only configure general Y-axis settings that are independent of the data file.



Scaling mode

- *Autoscale*
  Selected by default. The Y-axis of the graph is scaled so that it includes the smallest and largest value of the displayed signals.
- *Use scale definition from data file*
  You can define measurement range values for each signal in the module settings during data acquisition with *ibaPDA*. The values are saved in the data file.
  If you select this option, *ibaAnalyzer* interprets the measuring range limits as the start and end values of the scale.

Scientific notation

- *Auto*
  Depending on the order of magnitude of the scale values (number of digits before and behind the decimal point), *ibaAnalyzer* uses scientific notation (decimal powers) for the scales or does not.
- *Always scientific*
  Scale values in decimal powers
- *Never scientific*
  Scale values with digits before and after the decimal point

Show signal unit
If you enable this option, the unit of measurement from the signal table is shown after the scale values.

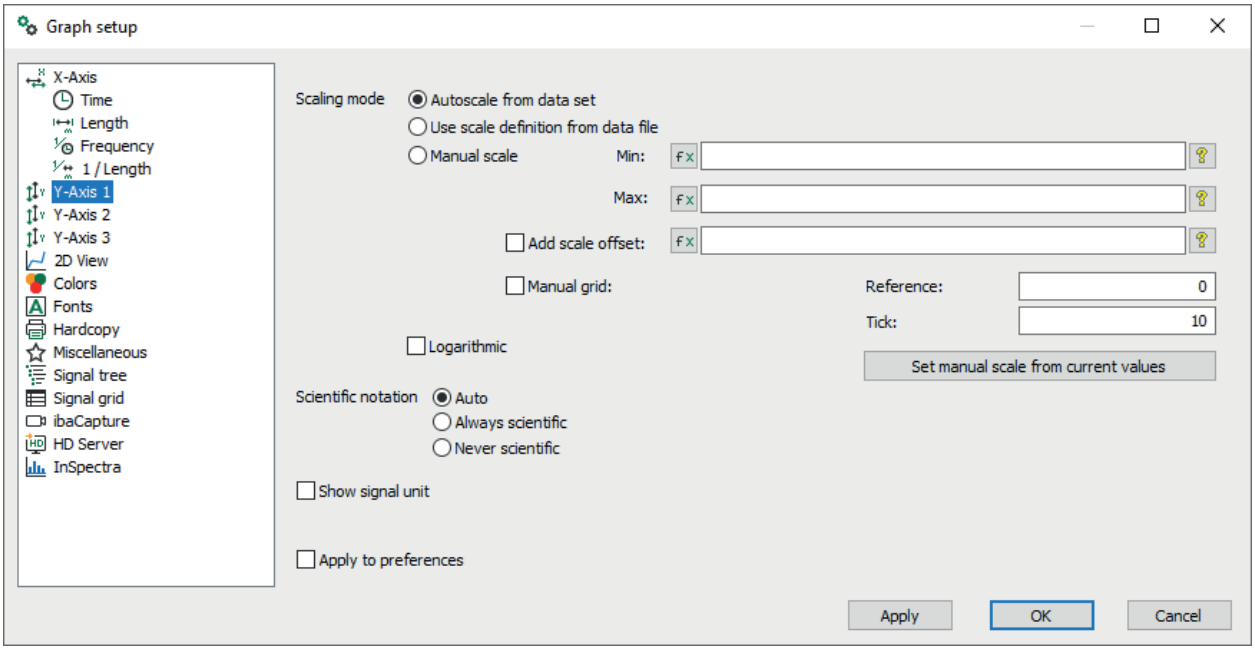
Show full edge zones in 2D top view (only with 2D top view graphs)
If you enable this option, with autoscaling the Y-axis in the 2D top view is scaled to the entire width including the empty marginal areas of the outer zones. Areas remain empty because the first and last values that can be displayed and interpolated are in the middle of the zone.

For more information on zone settings, see ↗ *Settings when using zone widths, page 140*.

5.2.2 Y-axis in Graph setup

The *Y-axis* node in the *Graph setup* provides more information and options than the *Preferences*. If there are multiple Y-axes in a graph, the graph setup offers a separate *Y-axis #* node for each axis. This allows you to make custom settings for each Y-axis.

You can also access the Y-axis settings via the Y-axis context menu.



Scaling mode

- *Autoscale from data set*
  Selected by default. The Y-axis of the graph is scaled so that it includes the smallest and largest value of the displayed signals.
- *Use scale definition from data file*
  You can define measurement range values for each signal in the module settings during data acquisition with *ibaPDA*. The values are saved in the data file.
  If you select this option, *ibaAnalyzer* interprets the measuring range limits as the start and end values of the scale.
- *Manual scale*
  With manual scaling, you can set fixed start values (Min) and end values (Max) for the scale. You can also use expressions to define these values dynamically, for example depending on certain process parameters. The <fx> button opens the expression builder.
- *Add scale offset*
  You can also define a scale offset with manual scaling. To do so, enter a fixed value in the input field. The value shifts the range defined by *Min* and *Max* on the Y-axis. A positive value shifts the scale range upward; a negative value shifts it downward.

  However, a fixed scale offset can be impractical, such as if the level of the measured values fluctuates from file to file. In such cases, make the scale offset variable and calculate it in any way you like, including by using the measurement signals. Open the expression builder via the <fx> button and enter a formula to calculate the scale offset.

- *Manual grid*
  With manual scaling, you can also subdivide the Y-axis, or grid lines, manually. Enter two values for the manual grid in the corresponding fields:
  - *Reference*: The reference value is the basis for the position of the grid. The value does not necessarily have to be identical to the maximum value of the manual scale. It can be either inside or outside that range. The reference value determines where the first grid line appears.
  - *Tick*: This value defines the distance between the grid lines. The grid lines and scale values are displayed at regular intervals based on the reference value.
    If you enter 0.0625 as tick value, for example, the Y-axis and grid will be subdivided in 1/16 increments.

Note



ibaAnalyzer automatically adjusts the scaling to optimize the display based on the value range (min and max values). If the grid becomes too narrow, the grid lines are displayed at integer multiples of the tick mark.

- *Logarithmic*
  If you enable this option, the scale will be subdivided logarithmically on the Y-axis. In an X-Y view, both axes (horizontal and vertical) can have a logarithmic scale. If a graph is switched to X-Y mode, the *Logarithmic* option in the graph setup is also available for the X-axis.
- <Set manual scale from current values>
  If autoscaling is enabled and you click this button, the current display settings of the graph are transferred to the manual scale fields. This makes manual scaling easier if, for example, you have adjusted the Y-axis in the recorder window with the mouse.

Scientific notation

- *Auto*
  Depending on the order of magnitude of the scale values (number of digits before and after the decimal point), *ibaAnalyzer* uses scientific notation (decimal powers) for the scales or does not.
- *Always scientific*
  Scale values in decimal powers
- *Never scientific*
  Scale values with digits before and after the decimal point

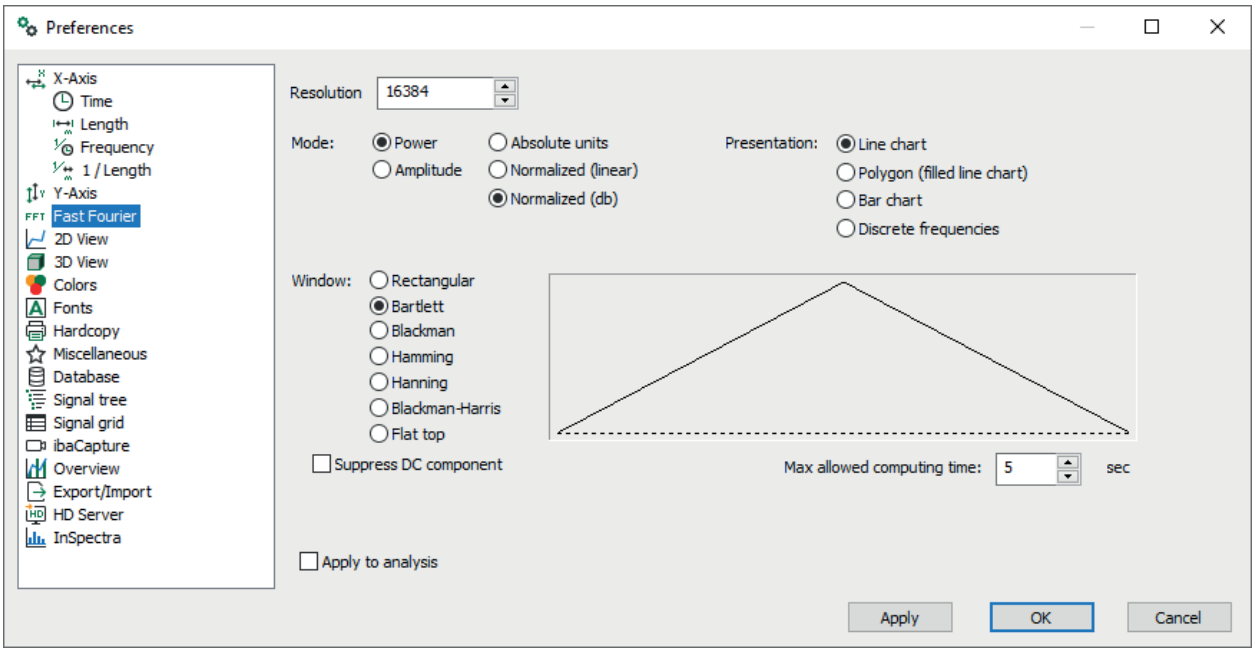
Show signal unit
If you enable this option, the unit of measurement from the signal table is shown after the scale values.

Show full edge zones in 2D top view (only with 2D top view graphs)
If you enable this option, with autoscaling the Y-axis in the 2D top view is scaled to the entire width including the empty marginal areas of the outer zones. Areas remain empty because the first and last values that can be displayed and interpolated are in the middle of the zone.

For more information on zone settings, see ↗ *Settings when using zone widths, page 140*.

5.3 Fast Fourier

In the settings for Fast Fourier Transform (FFT), select the calculation bases and the algorithms according to which *ibaAnalyzer* performs the FFT analysis in FFT display mode for a graph. The calculation mode or analysis window that you select for the FFT depends on the application case.



Resolution
In this input field you set the resolution in increments of powers of two between 128 and 131072. A higher value results in a more granular and denser FFT display because more frequencies in the spectrum are included.

Mode
The mode determines what is calculated.

- *Power*: Calculation based on power: The square of the amplitude of the FFT coefficient is calculated.
- *Amplitude*: Returns the amplitude of the FFT coefficients.

You can combine each of the two options above with one of the following three.

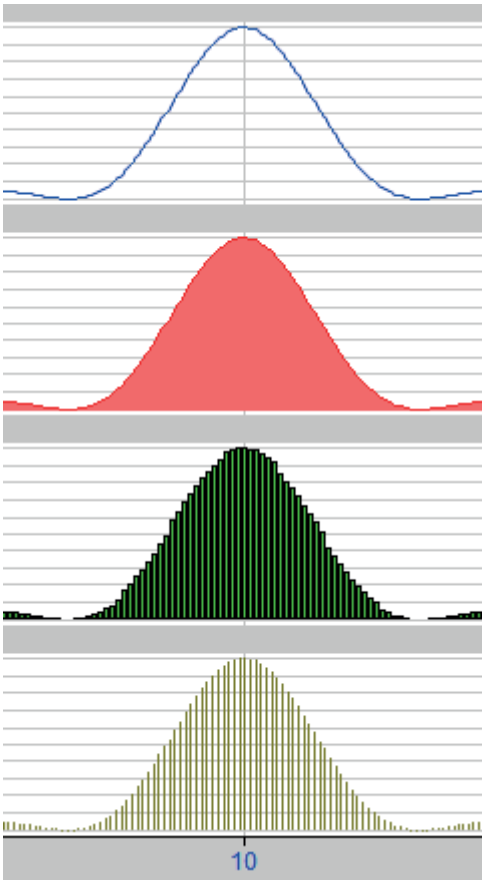
- *Absolute units*: Returns the power or amplitude of the FFT coefficient with no alternating component.
- *Normalized (linear)*: Normalizes the result relative to the estimated amplitude of the input signal with the *Amplitude* option or to the square of the amplitude with the *Power* option.
- *Normalized (db)*: Normalizes the result in dB.

Window
Selection of the analysis window for FFT. The shape of the window indicates which samples of a finite signal are weighted how strongly in FFT.

- *Rectangle*:
  All measuring points (samples) of a signal are weighted equally from start to finish.

- *Bartlett*, *Blackman*, *Hamming*, *Hanning*, *Blackman-Harris*, *Flat top*:
  Samples in the middle portion of the signal are weighted more strongly than the samples at the margin (start, end).

Display mode

<table>
<tr><td rowspan="4">
</td><td>Line chart
Frequency amplitude values as a simple temperature curve</td></tr>
<tr><td>Polygon (filled line chart)
Frequency amplitude values as colored filled curve (like 2D view)</td></tr>
<tr><td>Bar chart
Frequency amplitude values as wide vertical bars at the corresponding frequencies</td></tr>
<tr><td>Discrete frequency
Frequency amplitude values as vertical lines at the corresponding frequencies</td></tr>
</table>

Suppress DC component
If you enable this option, the DC component (frequency = 0) of a signal is excluded from the FFT analysis.

Max. allowed computing time
With very long data files and many values (samples), the calculation can take a long time, especially with a high resolution. This can lead to problems in automated analyses or processes running in parallel. In such cases you can limit the calculation time, possibly at the expense of accuracy.

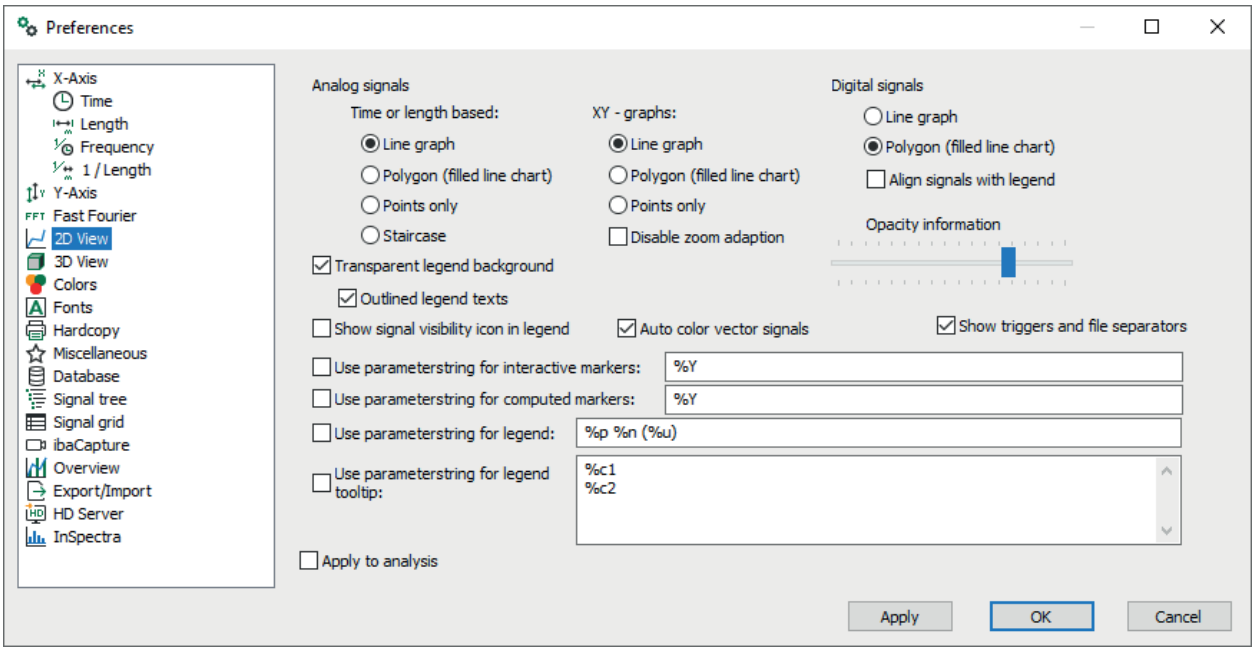
Note

The highest frequency to be observed is a maximum of 0.5 times the sampling rate. With a sampling rate of 1000 Hz, for example, only signal frequencies up to 500 Hz can be displayed.

On this see also ↗ *X-axis mode FFT, page 130*.

5.4 2D View

In this dialog, you specify how the signal trends are displayed in 2 dimensions.



The default settings are *Line chart* for analog values and *Polygon* for digital signals.

In the filled line chart view, the curves may overlap each other if there are multiple signals per graph. The "front" curve belongs to the signal at the bottom in the graph legend.

The *Points only* option represents the signal trend as a series of dots (one dot for each sample) without the connecting lines.

The *Staircase* function is suitable for values that remain constant between two points, such as batch or product numbers. The value of the last sample remains graphically constant until a new value occurs.

You will find examples of the different signal representation types in chapter ↗ *Standard view, page 135*.

More options:

Align signals with legend
This option applies to digital signals only. The digital signals are aligned exactly at the height of the matching signal legend.

Disable zoom adaption (for X-Y graphs only)
If you zoom into a trend graph with time-based signals and then switch to the X-Y view, the X-Y graph adopts the value range of the zoom level by default. You only see the values from the zoomed area in the X-Y view. If you enable this option, the zoom level is ignored on switching to the X-Y view, and the X-Y graph shows all the value ranges.

Transparent legend background
If you enable this option, the background of the legend becomes completely transparent. Although this does improve the visibility of the curve, the readability of the legend may be impaired. If you disable this option, the legend has a background. See also ↗ *Formatting the legend, page 110*.

Outlined legend texts
If you enable this option, the legend remains visible even if it is overlaid by a curve of the same color. The contours of the text are displayed in a contrasting color.

Opacity information
All types of markers, legends, units and cursors in a graph are assigned to a transparent layer, the so-called information layer, which is located above the graph. You can control the opacity or transparency by adjusting the slider.

- Position at far left = no opacity (100% transparent): No information is visible.
- Mid position = approx. 50% opacity: Curves are still visible behind the information level.
- Position at far right = 100% opacity (not transparent): Curves are concealed.

Show signal visibility icon in legend
If you enable this option, the visibility icon appears in front of the signal name in the legend. You can use this icon to switch the visibility of the signal for this graph on and off.



Auto color vector signals
If you drag a vector signal from the signal tree into a normal graph, the signals contained in the vector are shown as individual curves.

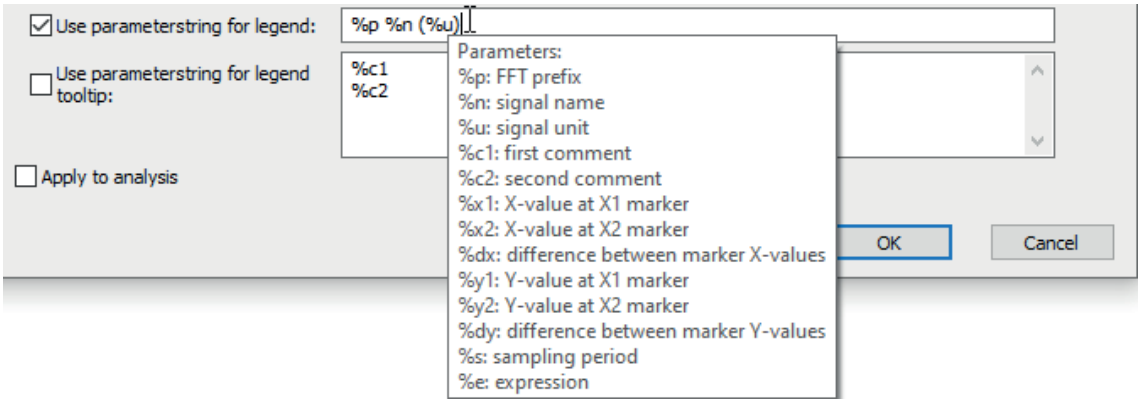
If you enable this option, the individual signal curves are automatically colored differently according to the color scheme (see *Preferences – Colors*). If you disable this option, all curves of the vector signal will have the same color.

Show triggers and file separators
If you enable this option, start and stop triggers as well as file limits (in the case of appended files) are displayed in the graph.

Use parameterstring for interactive/calculated markers
If you enable this option, you can add additional information or comments to the marker text. Enter the parameter strings as placeholders for the various options in the input field. By default, the Y-value is displayed at the marker position ("%y"). You can also enter any text.
Below you will find a description of the parameter strings.

Use parameterstring for legend
If you enable this option, you can add additional information or comments to the legend. Enter the parameter strings for the various options in the input field. By default, an FFT prefix, the signal name and the unit are displayed in brackets ("%p%n (%u)").

You can also replace the signal name by information or comments. Below you will find an explanation of the parameter strings.



Use parameterstring for legend tooltip
If you enable this option, you can add additional information to the tooltip. The tooltip is shown when the cursor is positioned on the legend. Enter the parameter strings for the various options in the input field. By default, the comments on the signal are displayed, if there are any ("%c1%c2"). You can thus decide which information is always displayed in the legend and which is displayed in the tooltip.

Overview of parameter strings

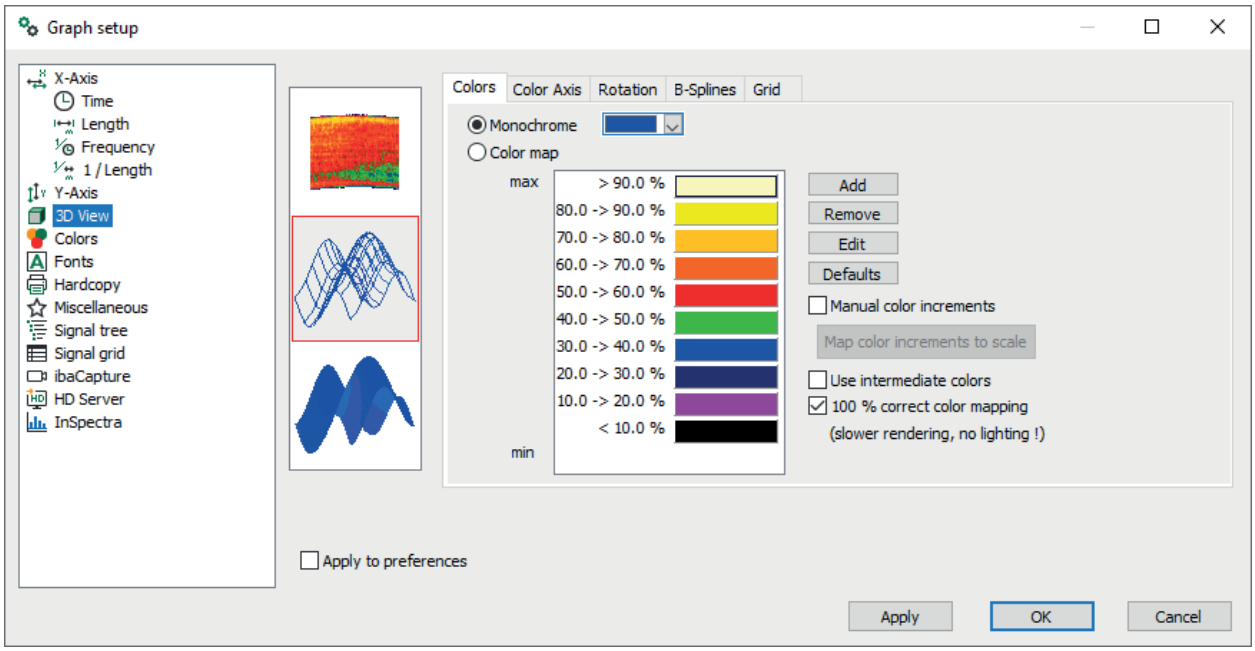
The following parameter strings are available. If these parameters are not assigned in the signal, e.g. if the signal has no unit, the parameter is ignored in the display.

Parameter	Description
%p	Inserts a prefix in the FFT view, e.g. "FT" before signal mode: "FT (amp-norm)"
%n	Inserts the signal name.
%u	Inserts the unit.
%c1, %c2	Inserts the first or second signal comment.
%x	Marker only: Inserts the X-value at the marker position.
%x1, %x2	Legend/tooltip only: Inserts the current position of marker 1 or 2 on the X-axis.
%dx:	Legend/tooltip only: Inserts the difference between the marker positions in X-axis units.
%y	Marker only: Inserts the Y-value at the marker position.
%y1, %y2	Legend/tooltip only: Inserts the current signal value at the position of marker 1 or 2 in the corresponding signal unit. Can be used with text channels.
%dy	Legend/tooltip only: Inserts the difference between the signal values at the positions of markers 1 and 2 in the corresponding signal unit.
%s	Inserts the time or length base with which the signals were written to the data file. The time-based data is given in seconds (sec), the length-based data in meters (m).

Parameter	Description
%e	Inserts the expression on which the signal is based or with which the signal is generated/calculated (as in the *Expression* column in the signal table).
%py	Marker only: For a subchannel, inserts the Y-value of the parent channel at the marker position.
%py1, %py2	Legend/tooltip only: For a subchannel, inserts the values of the parent channel. Can be used with text channels.

5.5 3D View

In this dialog, you configure the 3-dimensional view.



There are three variants of the 3D view:

- 2D top view, multicolored; see ↗ *2D top view, page 138*
- 3D wireframe, monochrome or multicolored; see ↗ *3D wireframe, page 137*
- 3D surface, monochrome or multicolored; see ↗ *3D surface, page 138*

You have various setting options for the different representation modes.

Colors tab

First choose whether you want the view to be monochrome or multicolored. The amplitudes of the measured values have different colors in multicolored views. 2D top views are always multicolored.

For a monochrome view, you can set the color using the drop-down list next to it.

Editing the color map
Corresponding to the value range of the signal measured, the colors are distributed over ten stages each representing 10% of the maximum value.

You can edit the colors and the subdivision of the scale.

Use the <Add> and <Remove> buttons to change the number of color levels. The percentage distribution is recalculated automatically.
Adding: Click on the color field above which you want to add another color level.
Removing: Click on the color field that you want to remove.

Use the <Edit> button to change the color of the color field.

Use the <Defaults> button to reset the changes to their defaults.
You can also adjust the default color settings in the *Preferences* under *3D View*. Here you can reset the colors to the *ibaAnalyzer* default settings using the <Defaults> button.

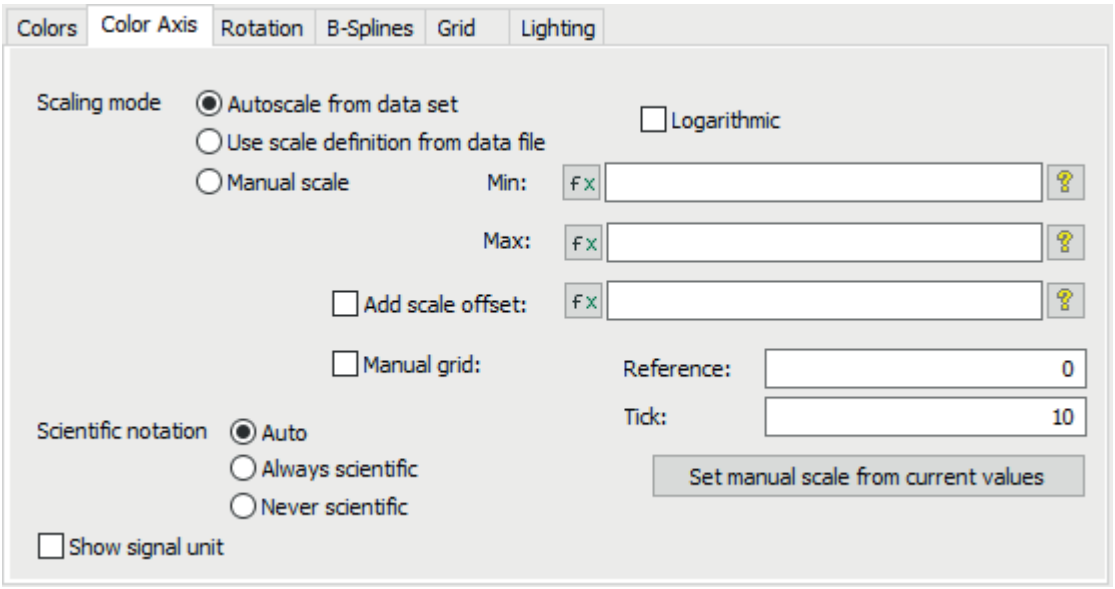
If you enable the *Use intermediate colors* option, the color transitions will be fluid.

If you enable the *100% correct color mapping* option, the color representation will be smoother and more precise. However, with large volumes of data rendering can take longer, and you cannot adjust the lighting.

Color Axis tab

On this tab you set the end values, position and subdivision of the color scale.
To do this, select *Manual scale*, and enter fixed values for the start and end of the scale.

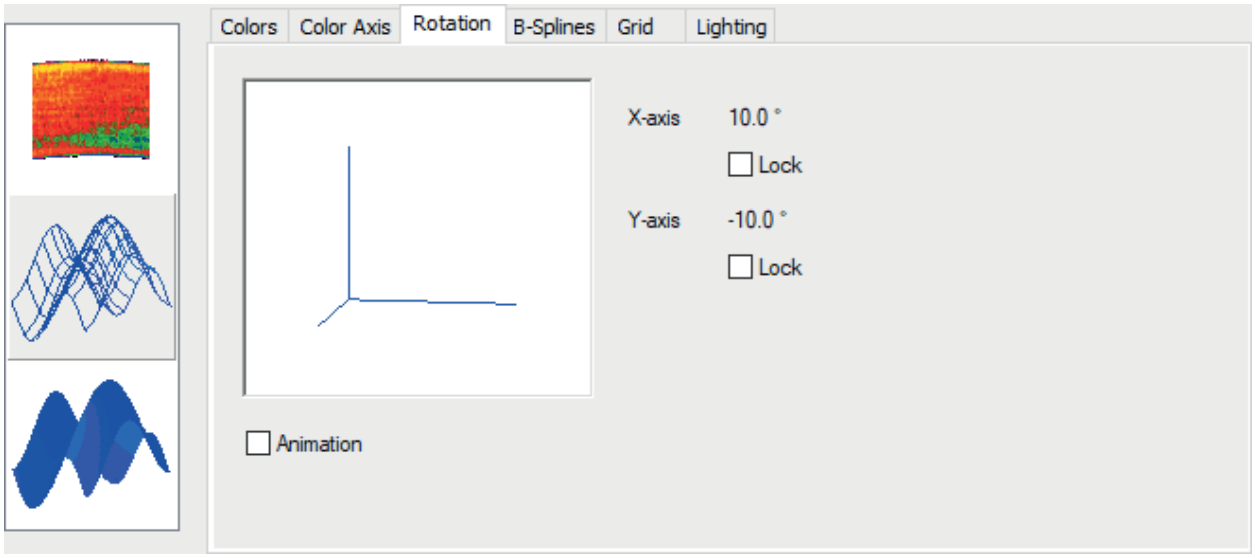
This tab is only available in the graph setup, and not in the preferences.



The settings correspond to those on the *Y-Axis* tab of the standard signal display; see ↗ *Y-axis in Graph setup, page 68*.

Rotation tab

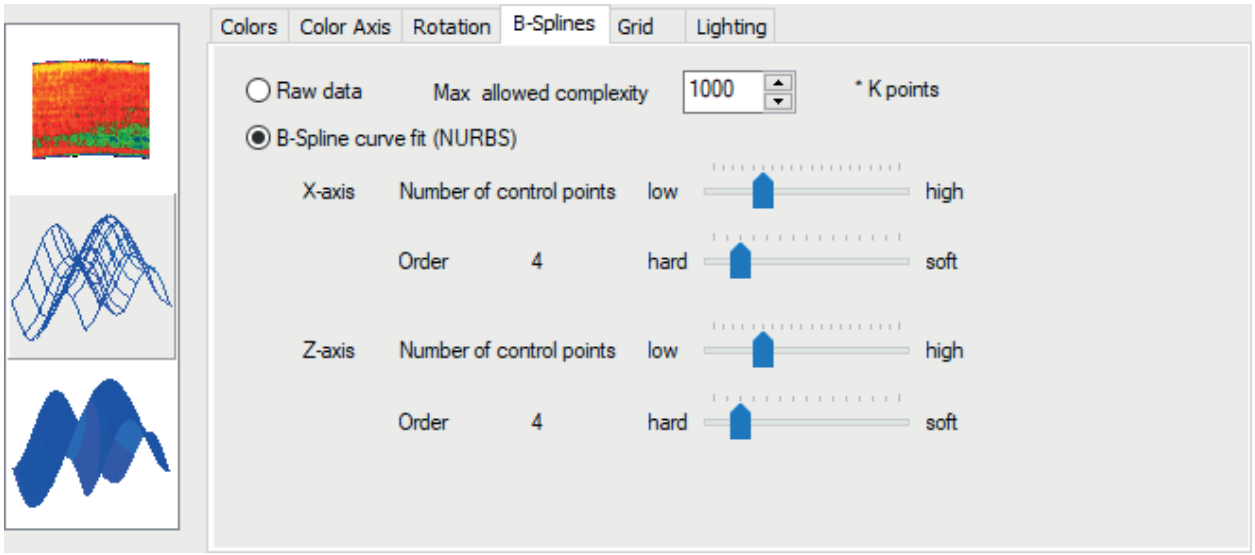
On this tab you can lock or release the X-axis and Y-axis for manual and animated rotation. When an axis is locked, you can no longer rotate the view around that axis.



If you enable the *Animation* option, a rotating animation of the graphic around all enabled axes starts in the graph.

B-Splines tab

You can use the B-spline settings to adjust the density of the grid.



If you select the *Raw data* option, the original measuring points are presented, and interconnected by straight lines in both the X-direction and the Z-direction.

If you select the *B-Spline curve fit (NURBS)* option, the surface is smoothed and rounded by means of B-spline calculations. In this, the connecting lines between the measuring points are converted into curves using additional interpolation points.

In the *Max. allowed complexity* input field, you can specify the total number of points (10,000 to 1,000,000 points = 10*K to 1000*K points). With big volumes of data, a large number of points can impair performance.

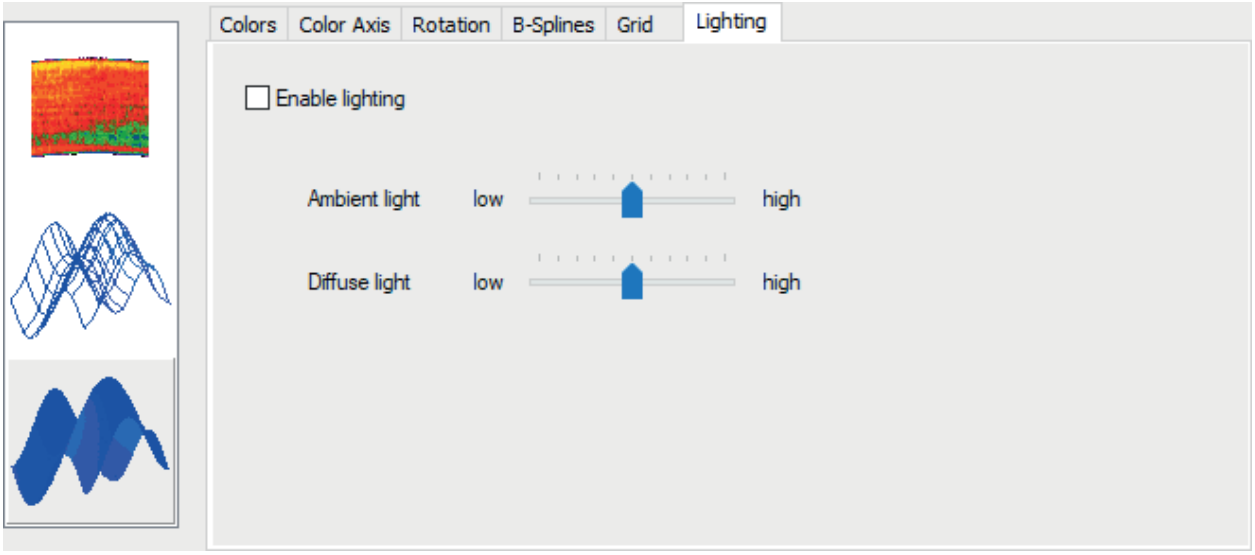
Use the sliders to set the point density and the curve characteristic for the X-axis and Z-axis.

Grid tab

On this tab, you can switch the display of the 3D grid and the scale values on the axes on or off.

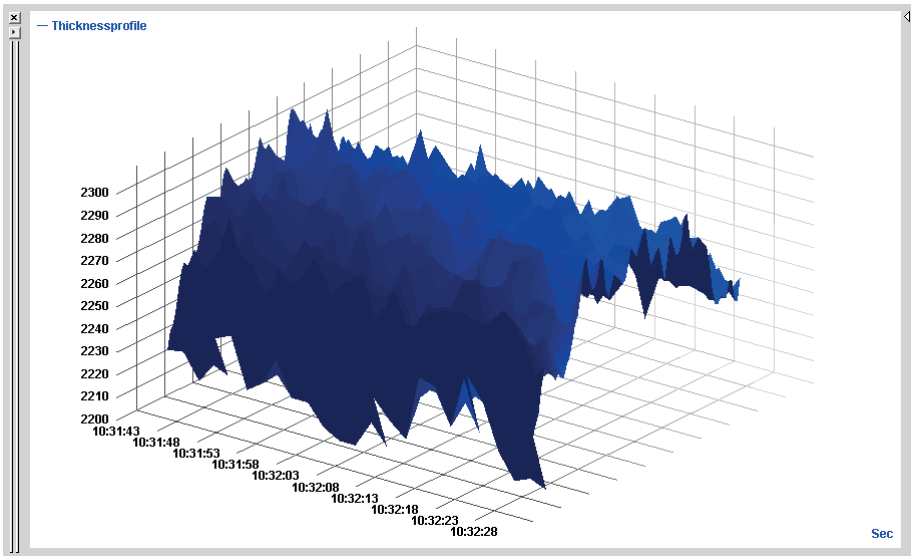
Lighting tab

The *Lighting* tab only appears in the dialog window if the *100% correct color mapping* option is disabled for wireframe or surface view mode on the *Colors* tab.



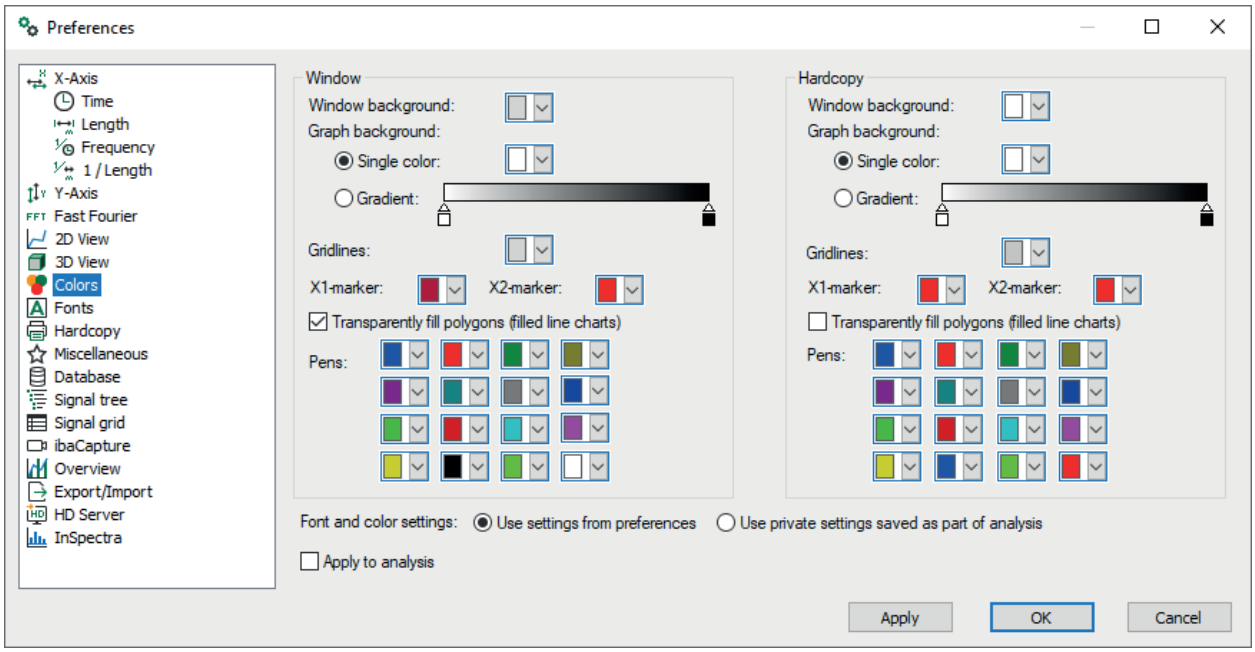
If you enable the *Enable lighting* option, *ibaAnalyzer* simulates side-lighting on the 3D graphic. This is possible for monochrome and multicolored views.

You can use the sliders to adjust the brightness and light intensity to achieve an optimum appearance.



5.6 Colors

You can adjust the color scheme for the user interface and the curve colors in this dialog. The color schemes are presented in separate sections for the screen and for the hardcopy in order to enable optimizing of colors for the two different media. A dark background is suitable for the screen, for example, but printing it uses a lot of ink.



About the background

You can fill the backgrounds of the graphs with a fixed color or a gradient.

If you select the *Gradient* option, you can adjust the colors by double-clicking on the small boxes to the left and right of the color bar.

X1-marker/X2-marker

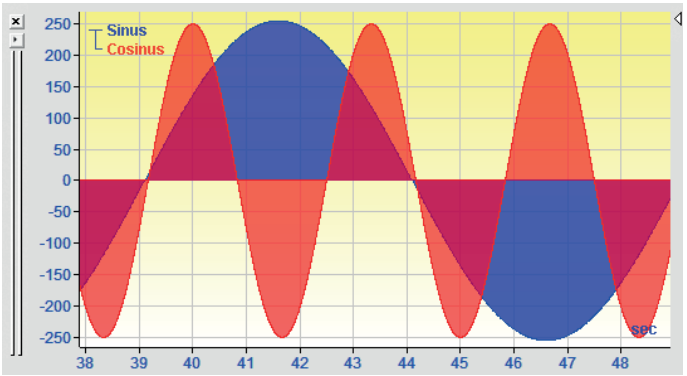
You can customize the marker colors.

Tip

If you are using *ibaAnalyzer* with video recordings from *ibaCapture*, different marker colors will make it easier to identify the video markers.

Transparently fill polygons (filled line charts)

If you enable this option, polygons (filled line charts) will be displayed transparently. This means that grid lines and overlapping signals remain visible, see image.

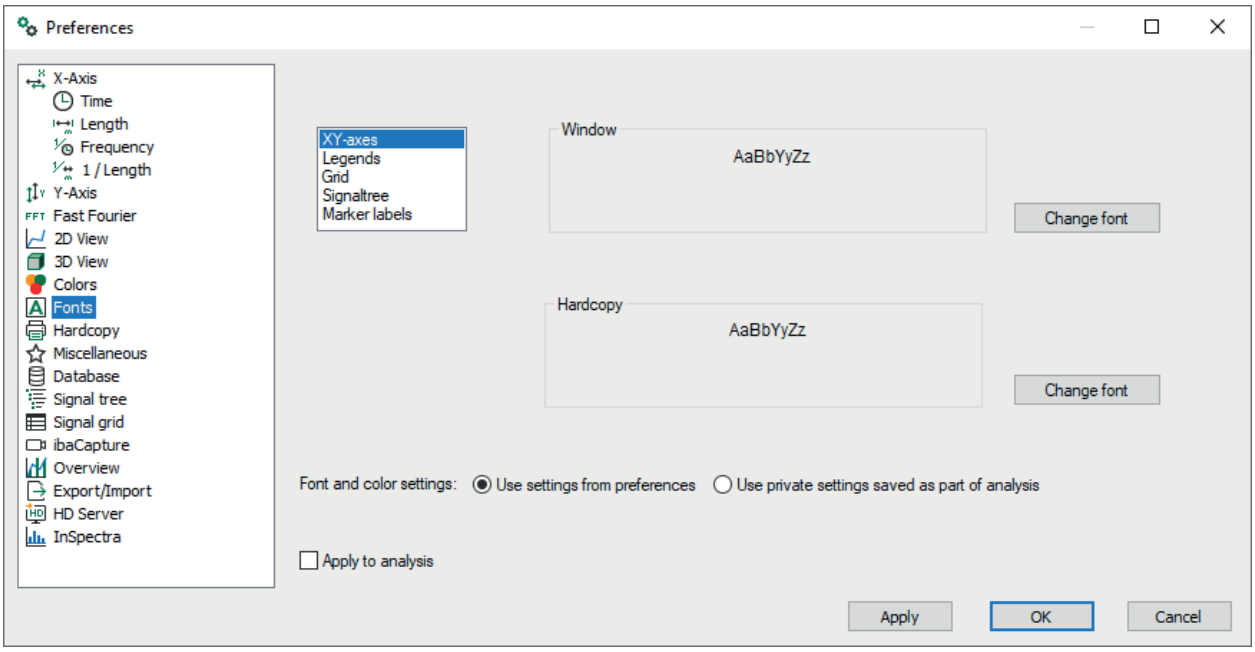


Pens
These colors define the 16 curve colors in the recorder window. *ibaAnalyzer* automatically uses these colors for the curves. This sequence (line by line from top to bottom) also dictates the default sequencing of signal colors in signal definition.

Font and color settings
The selection applies to *Colors* and *Fonts*. You specify whether the settings from the Preferences or those saved in the analysis are used.

5.7 Fonts

In this dialog, you can adjust the fonts for on-screen display (Window) and for printing (Hardcopy). You can select any fonts installed under Windows, and change the font, font size, font color and font style.

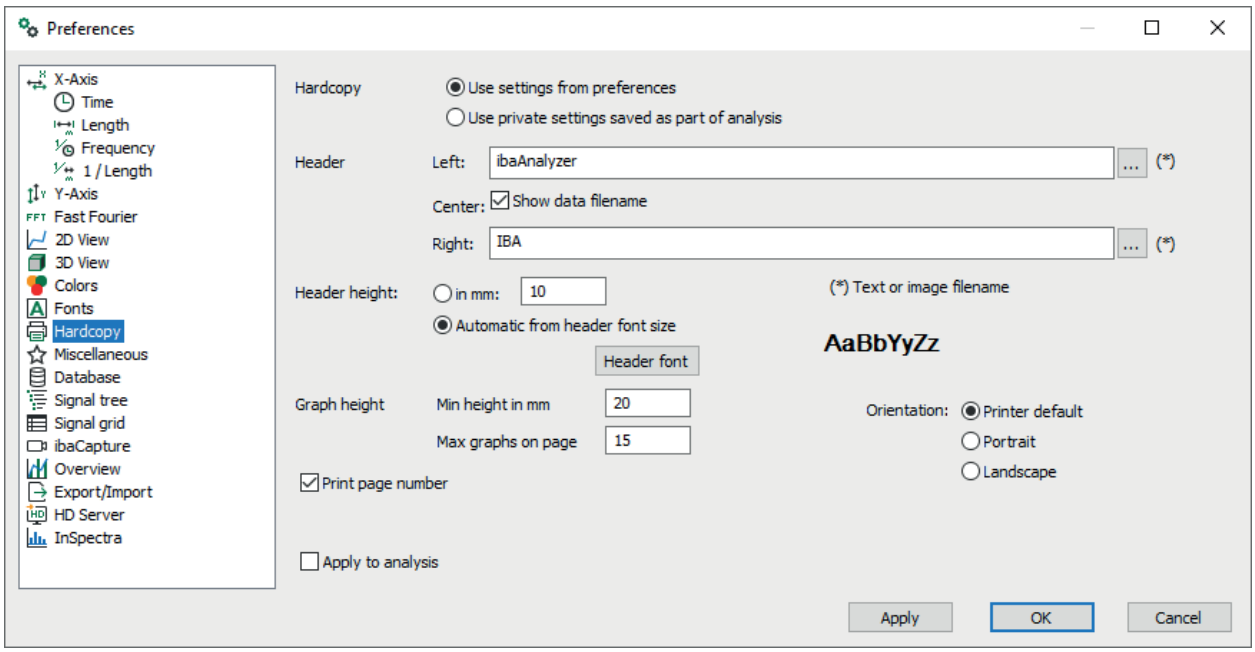


To change a font, select an interface area in the small window on the left. Then click <Change font> and adjust the font.

Font and color settings
The selection applies to *Colors* and *Fonts*. You specify whether the settings from the Preferences or those saved in the analysis are used.

5.8 Hardcopy

In this dialog, you can configure a printout of the current *ibaAnalyzer* view. The visible graphs and the current signal table tab will be printed.



Hardcopy settings
Choose whether you want to use the hardcopy settings from *Preferences* or those saved in the analysis.

Header
You can customize three areas for the title bar or header of the printout.

In the corresponding input boxes, you can enter any text or include images, such as a company logo.

To use a graphic file, enter the full path and the file name. You can also use the browser button <...>. Graphics must be in a standard format such as BMP, JPG, PNG, etc.

Header height
The header height is the distance between the header baseline and the top of the page. If you select the *Automatic from header font size* option, the header height will be adjusted to the selected font height or the graphic you are including. If you select the *in mm* option, enter the height. The header height is then fixed.

When you include an image in the header, *ibaAnalyzer* automatically scales its size to the set header height. If the image becomes too small, you must increase the header height.

<Font selection>
Specify the font for the header.

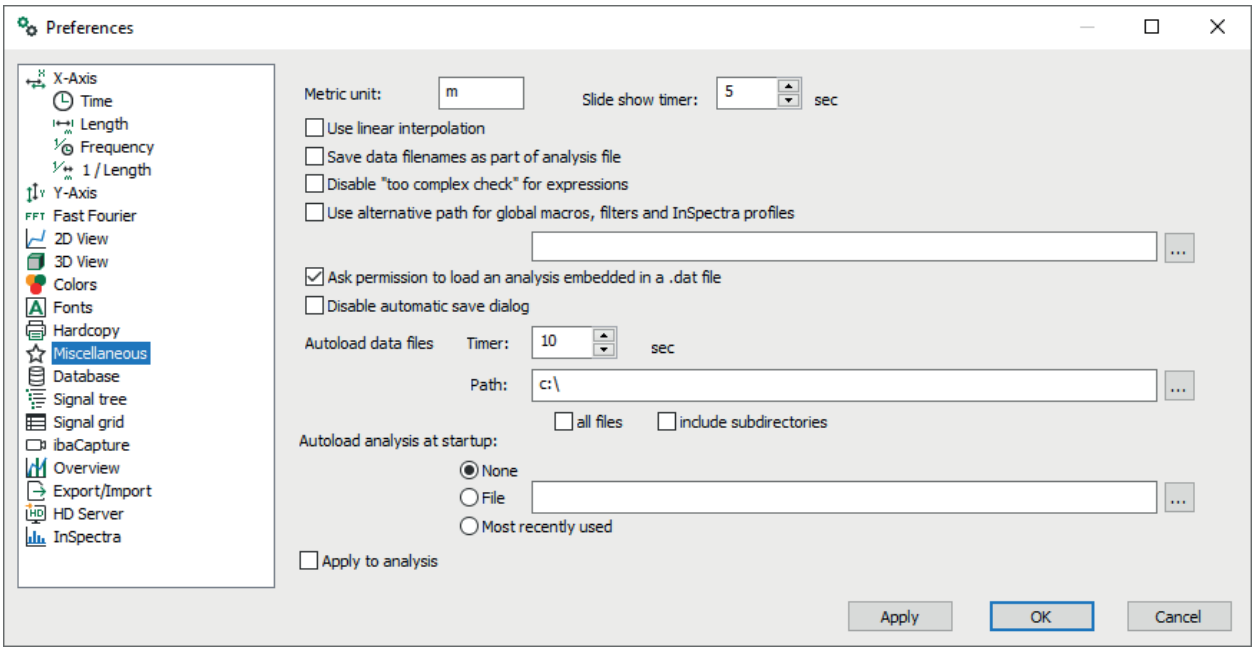
Graph height
The graph height relates to the graphs included in the printout. If a large number of graphs are opened, *ibaAnalyzer* tries to print as many of them as possible on the first page, which can impair readability. So you can specify a minimum height in mm for the graphs and a maximum number of graphs per page.

Orientation
Specify the orientation of the printout (portrait or landscape), or use the printer's default settings.

Print page number
If you enable this option, the page numbers are printed consecutively on each page.

5.9 Miscellaneous

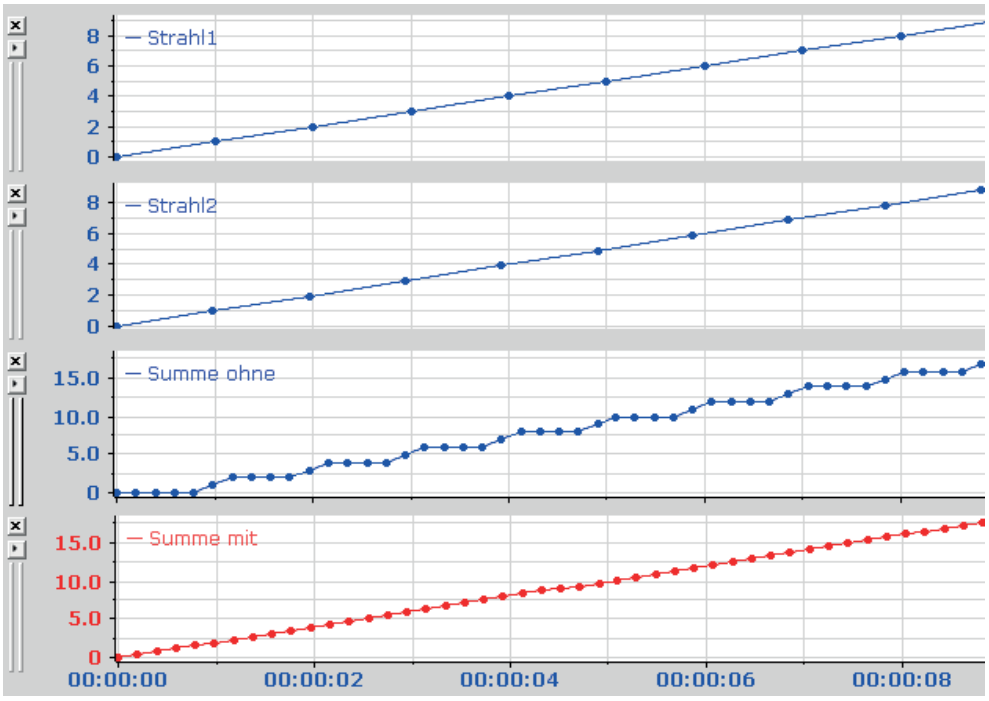
On the *Miscellaneous* tab, you can make key settings that affect the appearance of signals or the handling of data files for example.



Metric unit
Enter the unit of the length axis for length data here, e.g. m, km or inch. This entry only affects the labeling of the X-axis, not the calculations in the analysis. If you deviate from the metric system, and use Imperial or US units, for example, you must apply appropriate conversion factors in the expressions.

Slide show timer
Here you can set the time (in seconds) for switching between the data files in a slide show, see also ↗ *Slide show*, page 31.

Use linear interpolation
If you enable this option, the curves are displayed with linear interpolation. This is helpful if you add curves with different time bases, for example after a database query. Without linear interpolation, the resulting curve might look irregular or erratic (see blue curve below). Linear interpolation is used to connect the points of the curve evenly (see red curve below).



Save data filenames as part of analysis file
If you enable this option, the names of the data files that are open at the time of saving will be saved in the analysis. This is helpful if you have created an analysis specifically for a data file. Note that only the file name is saved, not the file itself. If you want to use the analysis with the measurement data on another computer, you must also copy or transfer the relevant data file.

This option is also available in the *Save analysis as* dialog.

Disable "too complex check" for expressions
By default, there is a limit on the calculation of expressions to prevent performance issues on the computer. This can be the case, for example, if you are using signals with large numbers of samples (> 10 million) in extensive calculations. If *ibaAnalyzer* classifies an expression as too complex, the result graph remains empty, and the diagnostics indicates a problem. (You can access the diagnostics via the <?> button in the signal table.)

If you want to deliberately bypass this limitation, enable the option.

Note

Functions such as "Resample", "Margin" and "Time" can also use up system resources if the parameters are too large. These functions have limitations that you cannot disable.

Use alternative path for global macros, filters and InSpectra profiles
If you open an analysis in a different environment and the paths to global macros or filters are no longer correct, you can specify a new path here to find those items automatically. *ibaAnalyzer* then finds the global macros and filters automatically.

Ask permission to load an analysis embedded in a .dat file
If you enable this option, you will be asked when opening data files whether you also want to load the analysis embedded in the data file, if available. If you disable this option, a data file with an embedded analysis will open immediately and the analysis will load.

Disable automatic save dialog
If you have made changes to an analysis and close *ibaAnalyzer* or open another analysis, you will normally be asked whether you want to save the changes. If you enable this option, the save dialog will be suppressed. Any unsaved changes will then be lost.

Autoload data files: Timer/Path
To automatically reload new data from the data file during a process-synchronous analysis, enter a time interval in seconds. Enter also a path in which *ibaPDA* will generate the data files. For more information, see ↗ *Process-synchronous analysis*, page 33.

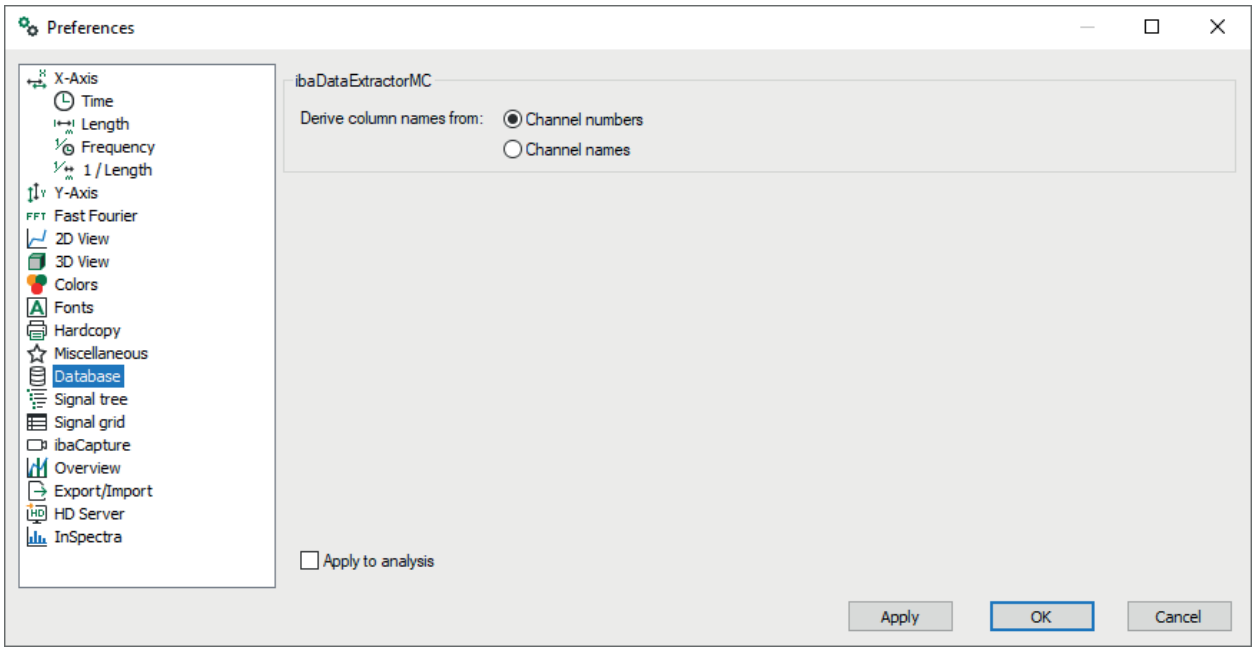
All files
If, in a process-synchronous analysis, you use multiple data files that are written simultaneously, by *ibaPDA* and *ibaLogic* for example, enable this option to reload all files. If you disable this option, only the first data file (at the top of the signal tree) will be automatically reloaded.

Include subdirectories
If, in a process-synchronous analysis, you use multiple data files that are written simultaneously, enable this option to reload all files from the subdirectories.

Autoload analysis at startup
If you want to work with a specific analysis when *ibaAnalyzer* starts, select *File* and enter the path and file name. If you prefer to start with the last analysis used, select *Most recently used*.

5.10 Database

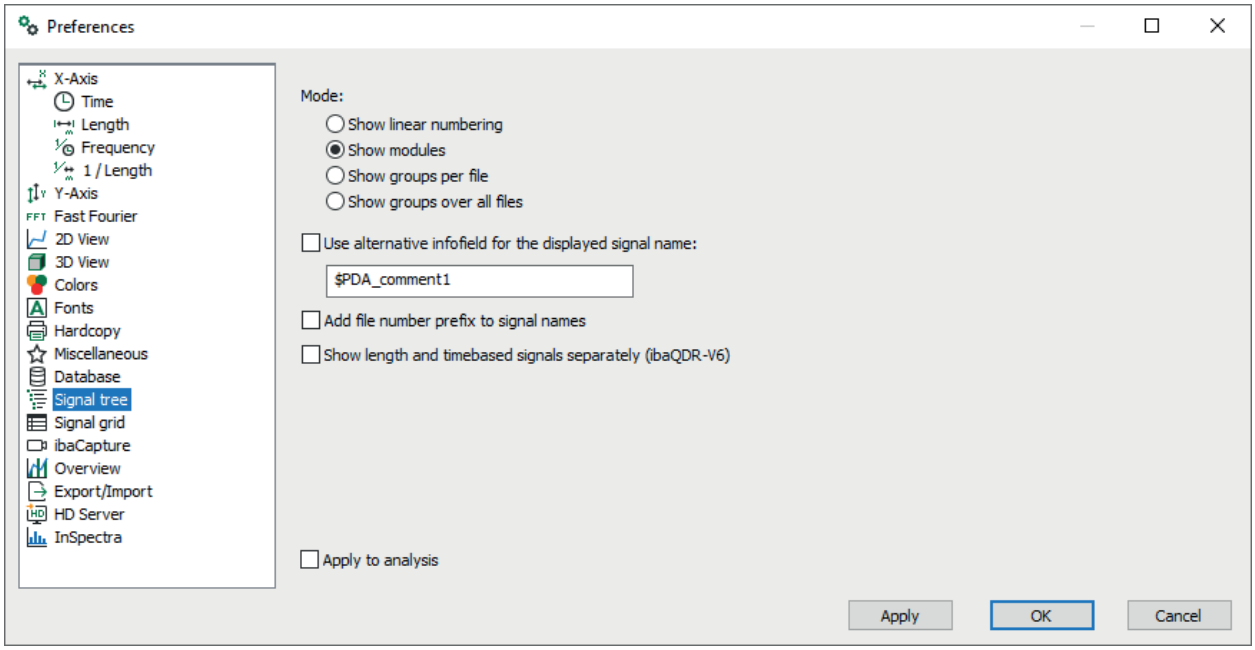
The setting under *Database* applies to database extraction only. If you are using the ibaDataExtractorMC format (multi-column), you can select whether column names in the database tables will be derived from the channel numbers ([module:channel]) or the channel names (signal names).



For more information on database connections, see part 4 of the *ibaAnalyzer* manual.

5.11 Signal tree

In this dialog, you define how *ibaAnalyzer* displays the signals in the signal tree.



Mode

These are the same options as in the context menu in the signal tree window.

- *Show linear numbering*:
  All signals in a data file are listed one after the other, with no a module identifier. All that remains is the change in analog signals and digital signals. Choosing linear numbering makes sense if large numbers of signals from a technology unit of the same kind extend over multiple modules, such as the 72 measuring zone values of a flatness measuring roll. This is helpful when creating arrays (vectors) for profile views.

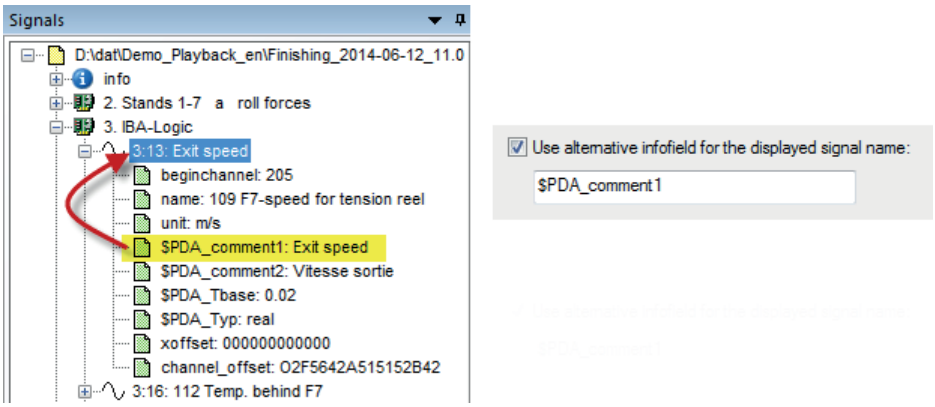
- *Show modules*:
  The signals are shown in the module structure defined in *ibaPDA*, so that the arrangement of the signals reflects the technological structure.
- *Show groups per file/Show groups over all files*:
  You can display signals in groups if a signal-to-group assignment has been saved in the data files.
  With *Show groups per file*, the data files are displayed at the top level of the signal tree window and the respective signal groups are displayed below.
  With *Show groups over all files*, the signal groups are displayed at the top level in the signal tree window.

Use alternative infofield for the displayed signal name
Each measurement signal has info fields that are configured in the *ibaPDA* or *ibaQDR* system. These fields are included in the signal tree of the data file if the corresponding information has been configured.

If you select this option and enter a different info field, the content of this field is then displayed in the signal tree instead of the signal name.

By default, "$PDA_comment1" is used as the alternative display name, and "$PDA_comment2" as the alternative signal reference, see also ↗ *Signal grid, page 87*. But you can also use any other channel info field.

The screenshot shows an example using comment field 1 "$PDA_comment1".



Note

This setting only changes the display name of the signals in the signal tree and in the legend. It does **not** change the signal reference as it is used in expressions, for example.

You can change the signal reference in Preferences in the *Signal grid* node, see ↗ *Signal grid, page 87*.

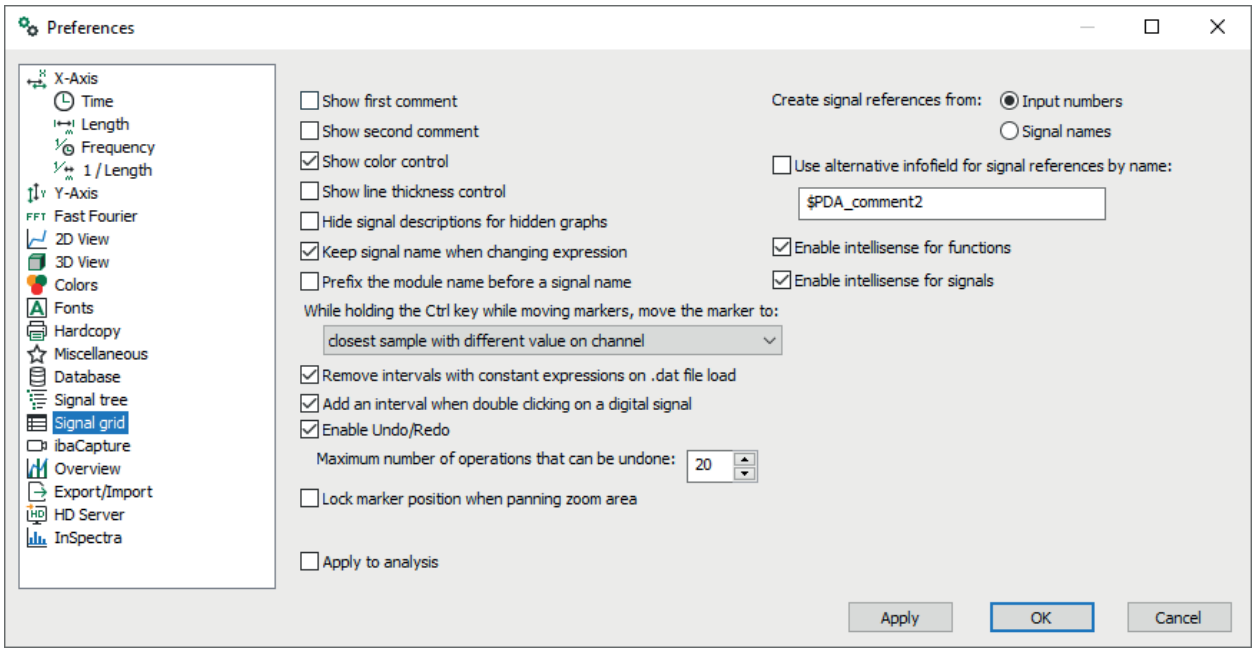
Add file number prefix to signal names
If you enable this option, you will see the file number before the signal number in the signal tree. The numbering starts with 1 at the second file. The signals of the first file are not assigned an additional number.

Show length and timebased signals separately (ibaQDR-V6)
This function is only relevant if you are using data files from *ibaQDR-V6* or older. *ibaQDR-V6* can store measured values on both time and length base in a data file. The signals of these files are only listed in the signal tree in *ibaAnalyzer*. To plot a signal on the length or time scale, select the corresponding type of presentation in the recorder window.

If you enable this option, all signals in the data file with a length and time base will be listed twice in the signal tree – including the measurement location and module.

5.12 Signal grid

This dialog offers multiple settings for configuration of the signal grid.



Show first/second comment
ibaPDA offers the possibility to enter up to two comments for each signal, such as for multilingual applications. The comments are also included in the data files. If you enable these options, the *ibaAnalyzer* signal table includes additional columns for the comments.

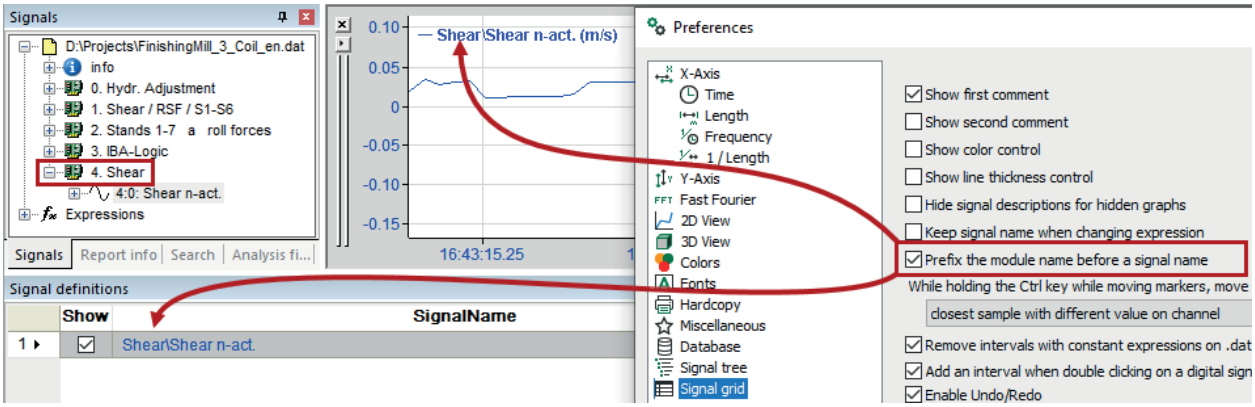
Show color control/line thickness control
If you enable these options, the *ibaAnalyzer* signal table includes additional columns for the color or line thickness.

Hide signal descriptions for hidden graphs
Enable this option if you want to hide signal table rows of signals when you hide the graphs in the recorder window. This will save space and make the table clearer.

Keep signal name when changing expression
If you enable this option, the new expression is not automatically adopted as the signal name when you make changes in the *Expression* column.

If you disable this option, the content from the *Expression* column is always used as the signal name.

Prefix the module name before a signal name
If you enable this option, the module name will be displayed in the graph and in the signal table in addition to the actual signal. This applies to signals being opened from the signal or analysis tree as well as from the search dialog.



Create signal references from
Choose how the signal is designated in the *Expression* column: either by the unique module/signal designation or the signal name (plain text). Plain text is easier to understand and clearer, but may not be unambiguous and can be misinterpreted.

Use alternative infofield for signal reference by name
Each measurement signal has info fields that are configured in the *ibaPDA* or *ibaQDR* system. These fields are included in the signal tree of the data file if the corresponding information has been configured.

If you enable this option, the content from the info field is used as the signal reference.

By default, "$PDA_comment1" is used as the alternative display name, and "$PDA_comment2" as the alternative signal reference, see also ↗ *Signal tree, page 85*. But you can also use any other channel info field.

Enable intellisense for functions/signals
Here, you can enable or disable the Intellisense feature. The Intellisense feature helps you enter expressions and signals, such as in signal definitions or markers. A pop-up appears with appropriate suggestions as you type. When you enter the first letter of a function, for example, a list of all the functions beginning with that letter is displayed. Enter "[" to show a list of available signals.

Select the function or signal using the arrow buttons or the mouse, and confirm with <Enter>.

While holding the Ctrl key while moving markers, move the marker to
Dragging the marker with the mouse moves the marker continuously. When you press the <Ctrl> key, the marker will jump from sample to sample. Specify the samples to which the marker locks on here.

Remove intervals with constant expressions on .dat file load
If you enable this option, only the intervals with a start or stop expression based on a dynamic calculation will be retained when loading a data file.

Add an interval when double clicking on a digital signal
Enable this option to add the interval with the first double-click on a digital signal and remove it with another double-click on the signal.

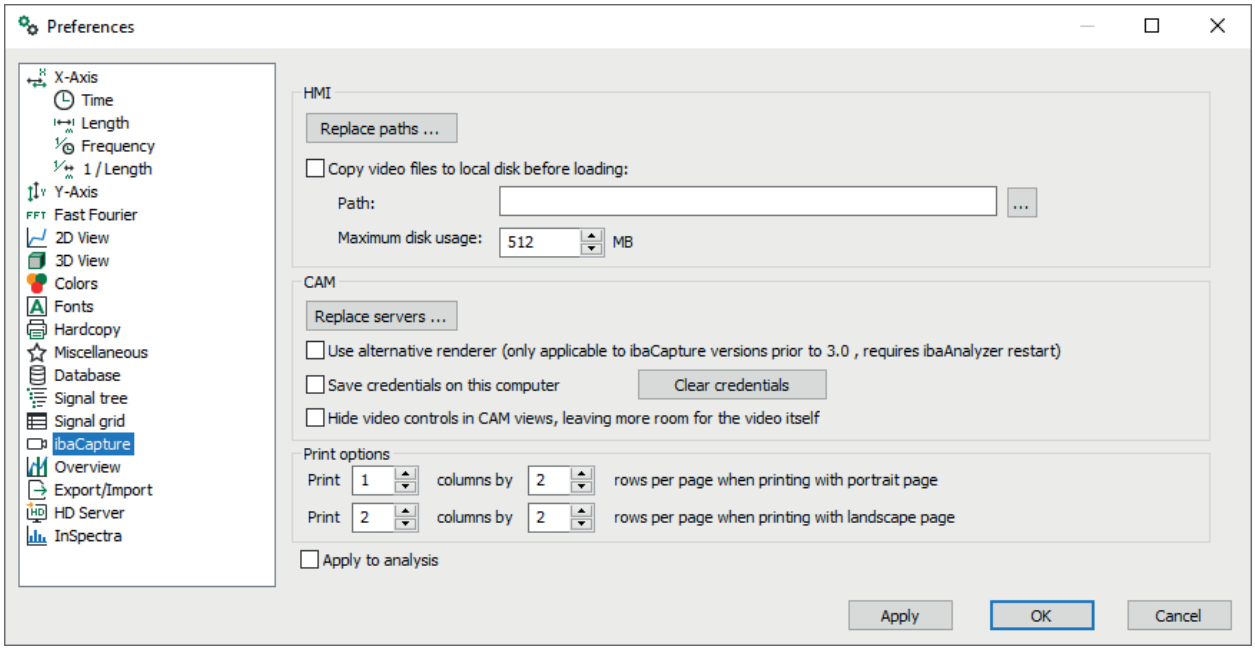
Enable Undo/Redo
The *Edit* menu includes the *Undo* and *Redo* functions, allowing you to undo or redo your last editing action. Set the number of actions you want to undo or redo in the field below.

Limiting the number of saved actions, or disabling the Undo function, is useful if you perform a lot of actions over a lengthy period of time without closing *ibaAnalyzer*, for example. Every saved action takes up memory, which can affect performance. To still benefit from the function, you are recommended to enable the function and limit the number of actions to between 10 and 20.

Lock marker position when panning zoom area
If you enable this option, the position of the markers relative to the zoom area will be fixed. Thus, the markers remain at the same position within the zoom area even when you are panning along the X-axis, e.g. by scrolling the X-axis with the middle mouse button or shifting the frame in the navigator pane.

5.13 ibaCapture

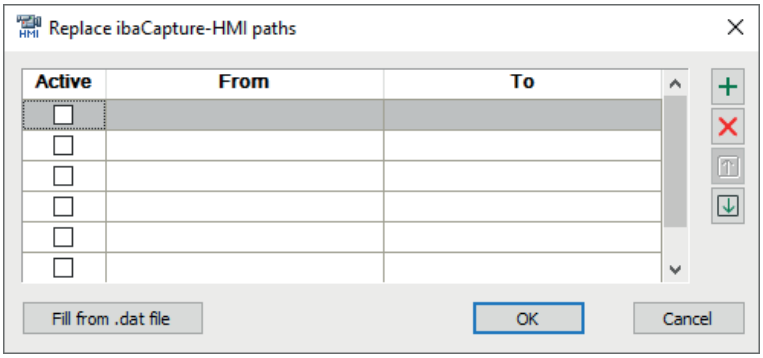
On the *ibaCapture* tab, you can adjust the settings for data files that also contain *ibaCapture* modules. For more information on using *ibaCapture* videos in *ibaAnalyzer*, see ↗ *Analysis with ibaCapture videos, page 256*.



HMI

These settings relate only to *ibaCapture-HMI* (V1 and V2).

<Replace paths>
In the dialog, you can adjust the path for storing the video files if the video files are stored in a different location than where *ibaCapture* saves them (if you previously copied the video files to a different drive, for example).



In the *From* column, enter the original path from the data file, and in the *To* column enter the new path where the files are actually located. If you have loaded a data file, and the cursor is in the *From* column, you can read out the original path with the <Fill from .dat file> button and enter it in the table.

Enable the paths you want to use in the *Active* column.

You can add, delete or change the order of rows using the buttons on the side.

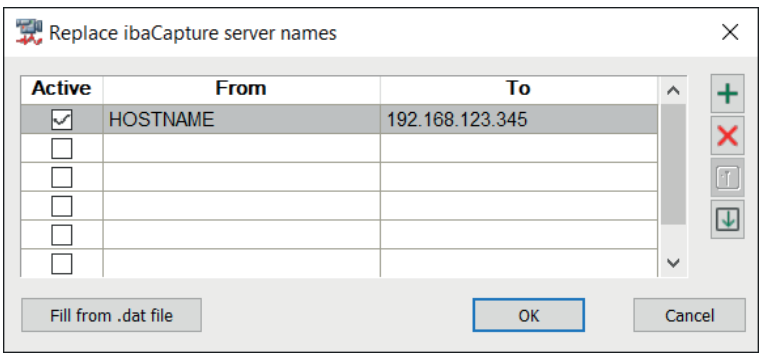
Copy video files to local drive before loading
If you enable this option, *ibaAnalyzer* copies the video files for loading to the local drive and plays them from there. This makes the videos run more smoothly, and reduces the network load.

- *Path*: Enter a path where you want to store the copies.
- *Maximum disk usage*: Limit the storage space for the video data in the specified directory so that the drive does not become too full. When the specified limit is reached, the oldest video files are deleted and overwritten.

CAM

These settings relate to *ibaCapture*.

<Replace server>
If *ibaCapture* and *ibaPDA* server are connected to multiple networks, you may have to replace the server. Even if the server has been renamed, or the videos have been moved to another server, you can specify the new server on which the *ibaCapture* server service must also run. The server saved in the data file is used by default.



In the *From* column, enter the original path from the data file, and in the *To* column enter the new path where the files are actually located. If you have loaded a data file, and the cursor is in the *From* column, you can read out the original path with the <Fill from .dat file> button and enter it in the table.

Enable the paths you want to use in the *Active* column.

You can add, delete or change the order of rows using the buttons on the side.

Note

The *ibaCapture* Server, where the videos are stored, must be accessible for the computer with *ibaAnalyzer* via the network.

Use alternative renderer (only applicable to ibaCapture versions prior to 3.0…)
If you are still using the Windows 7 operating system with *ibaCapture-CAM* <v3.0, you are recommended to enable this option in order to use the Windows graphics library for image generation. If you want to use the pre-installed iba renderer, disable this option.

If this option is enabled, *ibaAnalyzer* must be restarted.

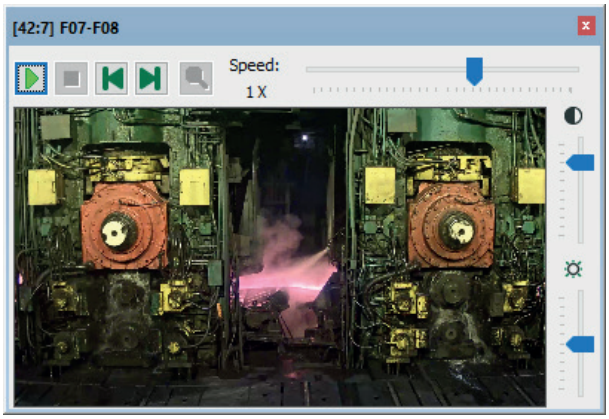
Save credentials on this computer
If the playback rights for a video have been restricted in the user administration of *ibaCapture*, you must log in with your user name and password when opening a camera channel in *ibaAnalyzer*. No videos are displayed without the appropriate authorization. This login is required once per *ibaAnalyzer* session.

If you enable this option, your credentials are saved locally, so that you do not need to log in again after restarting *ibaAnalyzer*.

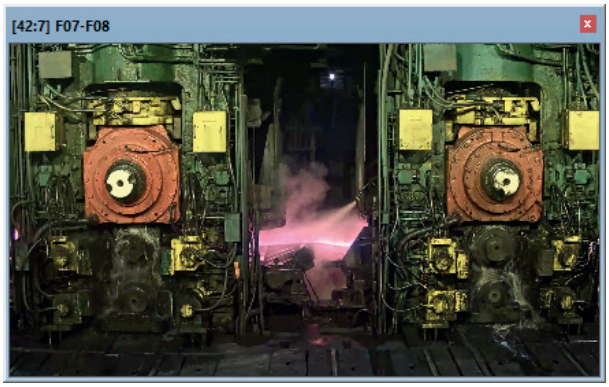
The <Clear credentials> button removes the saved login credentials from the computer.

Hide video controls in CAM views, leaving more room for the video itself
If you enable this option, the controls in the playback window are hidden. If the video controls are not visible in camera views, you will find them at the top of the *ibaAnalyzer* toolbar with the same functions.

With control elements



Without control elements



Print options

Specify how video images are printed using the print function in the *File* menu. You can set printout in portrait and landscape format separately.

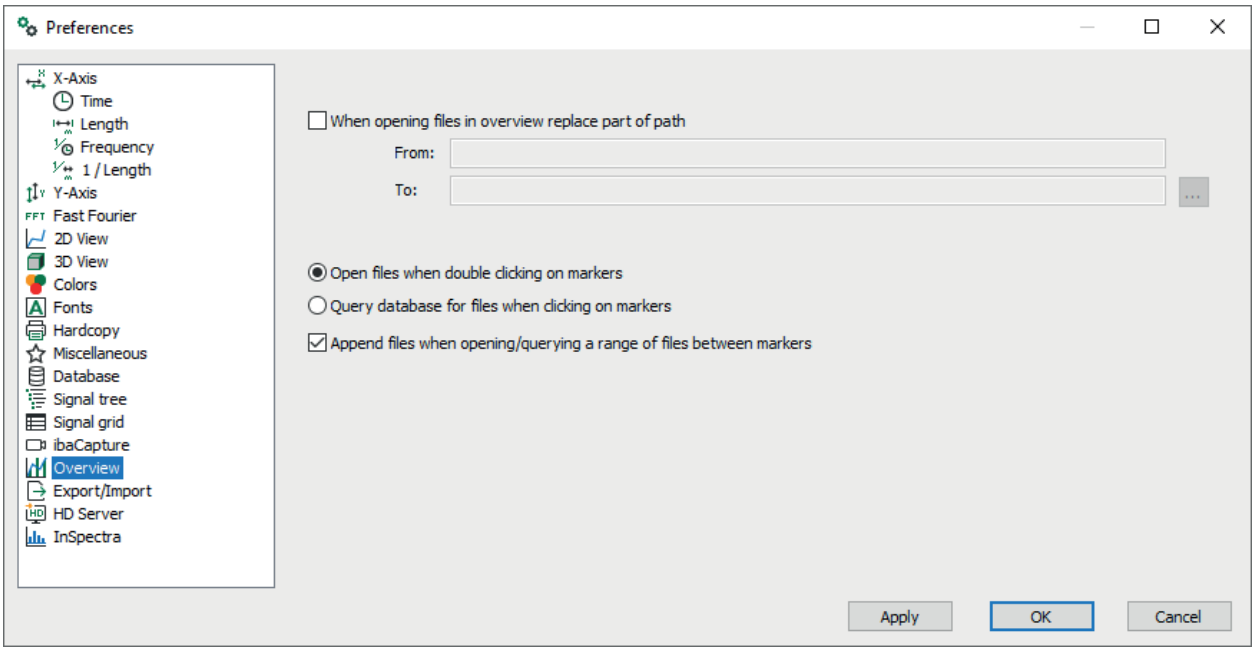
The number of columns determines how many images are to be printed next to each other on a page. The images are scaled accordingly.

The number of rows determines how many images are to be printed one below the other on a page.

You can check your settings in a new analysis by choosing *File – Print preview*.

5.14 Overview

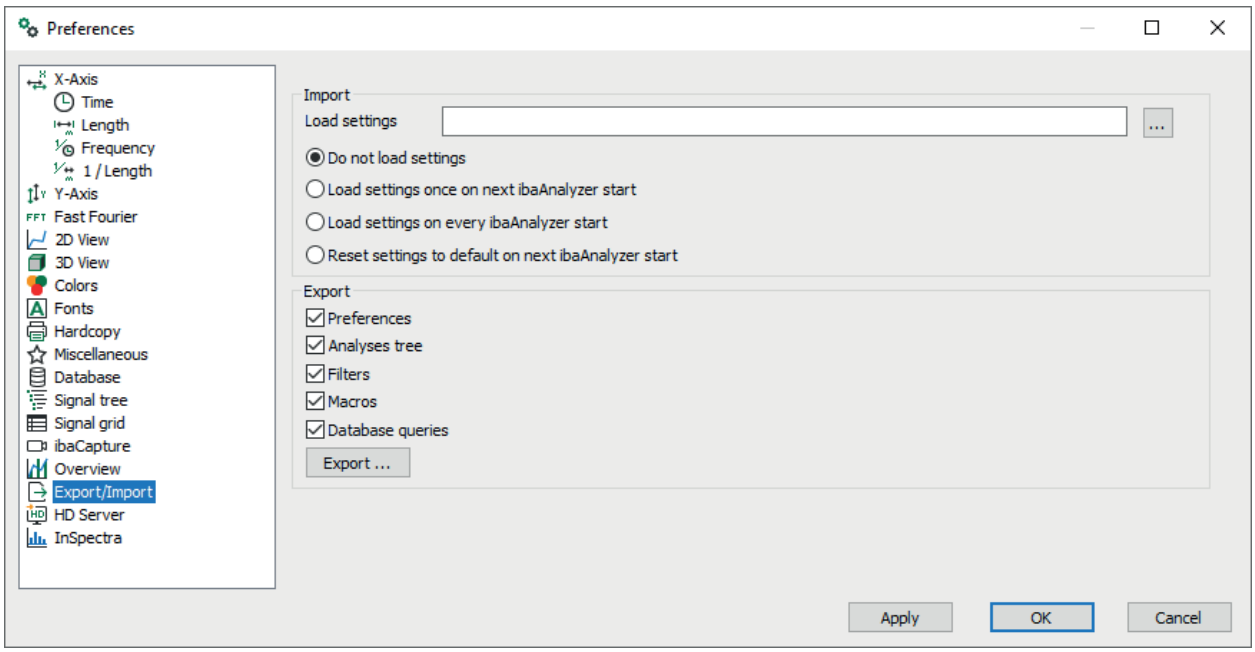
These settings refer to a specific form of database trend query. The result of this query is presented in the *Overview* tab in the area of the signal table.



For more information, see part 4 of the *ibaAnalyzer* manual, chapter *Overview options*.

5.15 Export/import settings

You can use this dialog to import or export existing settings. When exporting, the default settings are saved in a ZIP file, which you can re-import later.



Import

You have various options for importing settings. Enter the path to the desired file. Then select one of the options:

- *Do not load settings*:
  ibaAnalyzer does not load any settings at startup.
- *Load settings once on next ibaAnalyzer start*:
  ibaAnalyzer loads the default settings once from the ZIP file the next time it is started.
- *Load settings on every ibaAnalyzer start*:
  ibaAnalyzer loads the default settings from the ZIP file every time it is started.
- *Reset settings to default on next ibaAnalyzer start*:
  The next time *ibaAnalyzer* is started, it loads the settings as they were set during the initial installation.

Regardless of the selection, the setting is only applied after restarting *ibaAnalyzer*.

Export

For the export, first select which settings you want to export.

- *Preferences*:
  Exports all settings that are not listed separately.
- *Analyses tree*:
  Exports the settings from the *Analysis* tab in the signal tree window. You can also export or import these settings via the context menu in the signal tree window.

- *Filters*:
  Exports all global filters, see ↗ *Filter editor dialog window, page 204*.
- *Macros*:
  Exports all global macros, see ↗ *Export and import of macros, page 199*.
- *Database queries*:
  Exports the settings from the query dialogs. You can also export or import these settings in the relevant dialogs.

Finally, click on the <Export> button and enter a path and file name for the export file.

Note

In addition to export/import, all global filters or macros are copied from/to the *ibaAnalyzer* master directory, for example:

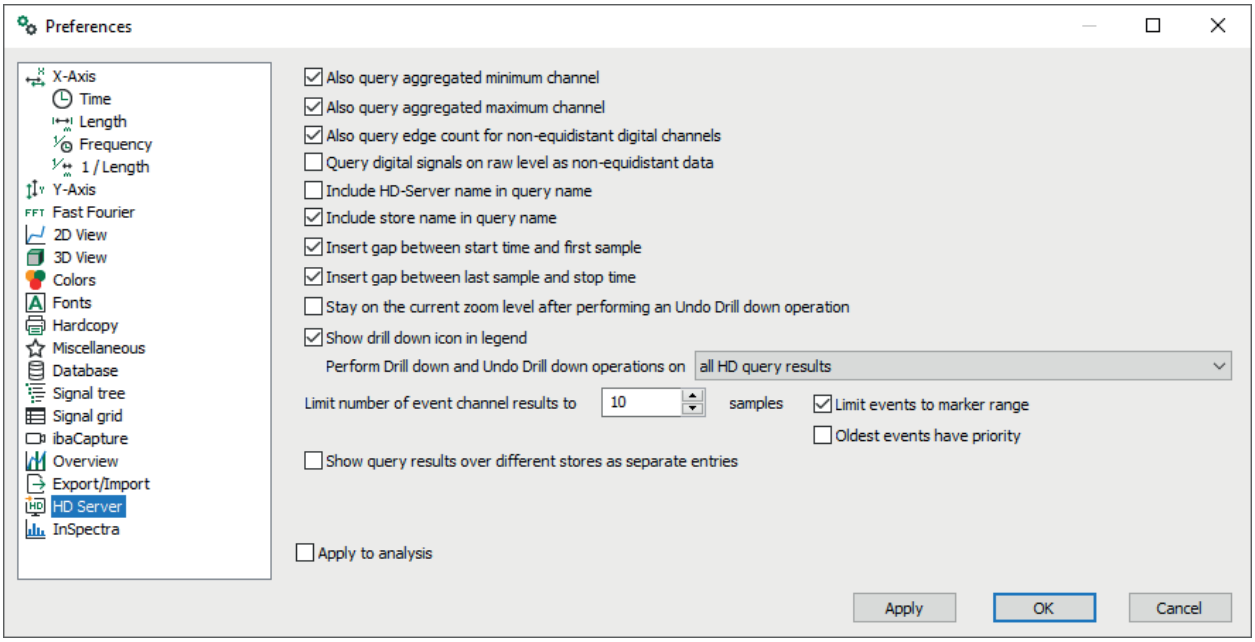
`C:\Documents and Settings\user name\Application Data\iba\ibaAnalyzer`

Macro files have the extension .mcr and filter files have thee extension .fil.

5.16 HD server

ibaAnalyzer enables access to data stored with *ibaHD-Server*. For more information on querying HD data, see ↗ *Access to HD data with ibaAnalyzer, page 219*.

You can use a number of settings to determine which signals are contained in the HDQ file after an HD query and how they are loaded and displayed. Changes to the default settings affect new analyses. If you make changes in the graph setup of a current analysis, where this item is also available, the HD data is automatically reloaded.



Also query aggregated minimum/maximum channel
By default, when loading a signal, its average value, maximum and minimum are loaded, if available. If you deselect these options, the corresponding value will not be loaded in *ibaAnalyzer* and you can save memory space.

Also query edge count for non-equidistant digital channels
If you enable this option, an additional subchannel ('edgeCount') is visible and evaluated. This subchannel returns the number of edges of non-equidistant digital channels within the aggregation period for the aggregation level.

Query digital signals on raw level as non-equidistant data
If you enable this option, digital signals are always queried as raw data, regardless of the aggregation level for analog values. This is particularly useful for long-term queries because the time stamp and duration of signal changes can be recorded correctly without errors due to data aggregation. *ibaAnalyzer* only recognizes signal changes and queries one data point for each change. The result is a non-equidistant digital signal.

Note



Before activating, check whether all functions and expressions in your current analysis support non-equidistant signals.

This option can impair performance if there are many signal changes. In this case, deselect this option.

Include HD server name in query name/Include store name in query name
The name of the HD query result basically consists of the name of the HD server, the store name and the start time and stop time of the queried area. If you enable these options, the HD server name or store name is part of the name. To shorten the name, deselect these options and the HD server name or store name will not be used.

Insert gap between start time and first sample/Insert gap between last sample and stop time
It is possible that there are no values at the beginning or end of the specified query range. If you check these options, a gap is shown in the graph between the start time and the first measured value or between the last measured value and the stop time. The X-axis corresponds exactly to the specified time range.
If you deselect these options, no gaps are displayed and the X-axis starts with the first measured value or ends with the last measured value.

Stay on the current zoom level after performing an Undo Drill down operation
If you enable this option, the display remains at the zoom level from which the drill-down was triggered after undoing the drill-down. If you deselect this option, the display shows the complete query area again after undoing the drill-down.

Show drill down icon in legend
If you enable this option, an arrow appears in the signal legend as soon as a drill-down is possible. If you deselect this option, you switch off the display of the arrow.

Perform Drill down and Undo drill down operations on …

- all HD query results (default)
- all HD query results for which a related channel or expression is present in the currently selected graph
- the HD query result the currently selected channel or expression is related to

Select which query results the drill-down function should affect.

If, for example, you have included several HD stores in the query and received a corresponding number of query results, then all HD stores would be queried again during a drill-down. However, if you are only interested in the drill-down for a specific store, you can limit the drill-down here.

Limit number of event channel results to … samples
The number of events in a given HD query period can vary greatly. If a large number of events occur, the display in the trend graph can quickly become confusing. You can use this setting, for example, to reduce the number of events that are displayed in the graph after an HD query.

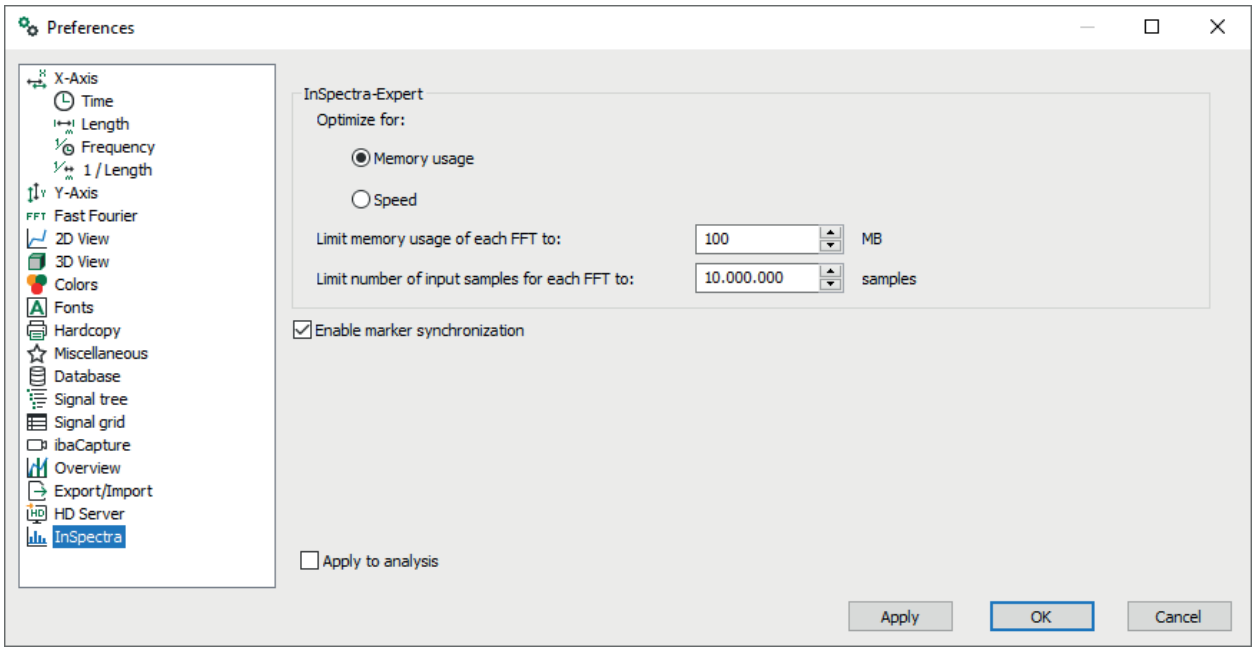
Limit events to marker range
If you enable this option, only events within the time range of the markers in the recorder window are displayed in the trend graph and in the event list.

Oldest events have priority
If you enable this option, the oldest events are displayed first in the event list. By default, the most recent events are displayed first.

Show query results from different stores as separate entries
If you include several HD stores in an HD query and use a signal condition, you will receive a separate query result from each HD store each time the condition is met. All query results appear in the file group window. Depending on the number of stores and results, the list can become very long. If you deselect the option, only one line is used per query result in the file group window, which summarizes all HD stores. This makes the list shorter and clearer. When you open a query result, you will see the detailed breakdown by HD stores in the signal tree.

5.17 InSpectra

These settings are only relevant for the use of *ibaAnalyzer-InSpectra* in an FFT view.



Optimize for
Choose whether *ibaAnalyzer* is optimized for memory usage or speed. If you select *Speed*, *ibaAnalyzer* buffers the signals before they are transferred to the FFT component. This requires more memory, but is much faster.

Limit memory usage for each FFT to … MB
Set a storage volume limit for each FFT view. For example, if you drag more signals into the FFT view than the memory capacity allows, you will receive a warning message.

Limit number of input samples for each FFT to … samples
Limit the number of samples used to calculate the FFT to prevent performance issues.

Enable marker synchronization
If you enable this option, all markers move synchronously in the different views (playback area, InSpectra view, Orbit view). This means that moving a marker in a view automatically moves all other markers to the same position. If the option is disabled, the markers move independently in the different views.

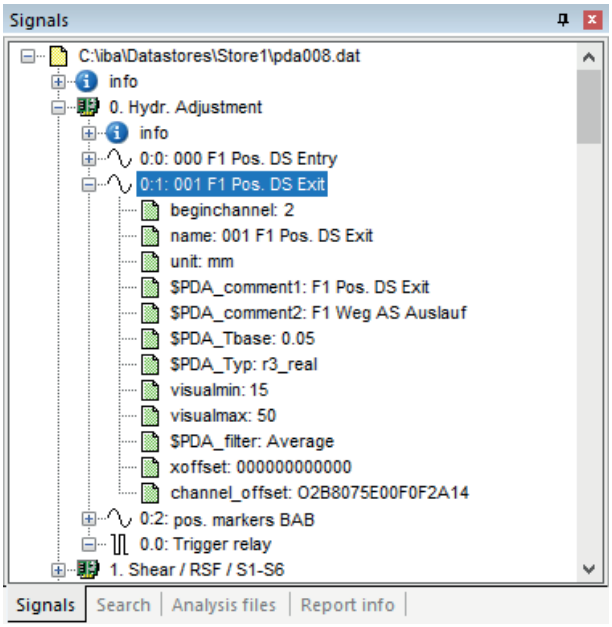
6 Displaying signals

You can display signals in different views in the recorder window. The default view in *ibaAnalyzer* is graphs in the recorder window. The following chapters describe this view, as well as other more specialized views, such as the 3D and FFT views.

For basic information on the recorder window, see *ibaAnalyzer* manual part 1, chapter *The recorder window*.

6.1 Signal information in the signal tree

The signal tree includes another branch with information below the signal.



Singal identification

Symbol	Singal type	Signal identification
	Analog signal	Module:Channel
	Digital signal	Module.Channel

Important info fields for signals

- name: Channel name acc. to PDA module setting
- unit: physical unit of the signal
- visualmin, visualmax: scale limit values acc. to PDA module setting (analog values only)
- $PDA_comment1/2: signal comments acc. to PDA module settings
- $PDA_Tbase: time base acc. to recording profile
- $PDA_Type: Data type
- $PDA_filter: filtering (average, maximum or minimum) acc. to recording profile

Depending on the application, *ibaPDA* configuration and type of data file, more information can be included, e.g. if the file is generated by *ibaQDR*.

6.2 .Selecting and displaying signals

Note



This chapter and the following chapters describe the signal display in the default recorder window of *ibaAnalyzer*. This recorder window has the following key features:

- fixed position
- graphs arrangeable only one below the other
- one X-axis for all graphs with the same X-axis mode
- zoom area and positions of the interactive markers are the same for all graphs of the same X-axis type

For a more flexible signal display, there is a separate graph similar to the one in *ibaPDA*. For more information see ↗ *PDA trend view, page 153*.

When a data file is open, you can select any signals and display them in the recorder window, such as to carry out an analysis.

Display signals in the recorder window

You have various options for displaying signals:

- Open the context menu in the signal tree on the desired signal and select *Show signal*.
- Double-click on the desired signal in the signal tree. This opens the signal in a new graph.
- Drag and drop the desired signal into the recorder window. You can open the signal in a new graph or add it to another open signal, see ↗ *Moving signals, page 100*.
  You can also drag and drop modules or groups from the signal tree into the recorder window. The signals are then displayed individually.

If you do not want to open each signal in a separate graph, so the signals are readable and clear, you can also include multiple signals in one graph, see also ↗ *Moving signals, page 100*. The signals in a graph can have a common Y-axis or different Y-axes. There is only one X-axis for all time-based graphs, and another X-axis for all length-based graphs if these are any.

Tip



To display signals in the selected graph with **separate Y-axes**, double-click on the signal name in the signal tree while holding down the <Ctrl> key.

To display signals in the selected graph with **a common Y-axis**, double-click on the signal name in the signal tree while holding down the <Shift> key. Each additional signal is assigned to the Y-axis of the lowest signal.

Display multiple signals simultaneously in the recorder window

You can select multiple signals in the signal tree by the standard Windows method of a mouse click + <Shift> or <Ctrl>, and drag them into the graph with the mouse.

When dragged into an existing graph, the selected signals are treated as a group, and are assigned a common Y-axis.

When you drag the signals onto the Y-axis of an existing graph, all signals have the same Y-axis – including the existing signal. When you drag the signals into the graph, only the new signals have a common Y-axis. The existing signal retains its own Y-axis.

If no graph is open yet, or the group is dragged onto the X-axis, the behavior is as follows:

- Drag the group by holding down the <Ctrl> key or <Shift> key:
  The signals are displayed together in a new graph.
- Drag the group with the mouse only:
  Each signal is displayed in its own graph.

6.3 Moving signals

You can move signals back and forth between graphs. This means that you can drag a signal from one graph into another one that has an existing signal.

The signals in a graph can have a common Y-axis or different Y-axes. There is only one X-axis for all time-based graphs, and another X-axis for all length-based graphs if these are any.

Procedure

1. In the graph, move the cursor to the name of the signal you want to move.

→ A wavy line at the cursor position indicates that the cursor has picked up the signal.



2. Drag the signal to the other graph and drop it in a vacant area.



→ The two signals have separate Y-axes.



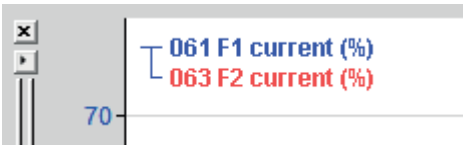
3. If, instead of dragging the signal into a vacant area, you drag it onto the Y-axis or the existing signal until a small gray arrow appears, the moved signal is assigned to the same Y-axis.



→ The two signals have a common Y-axis.



4. The color of the signals remains the same when they are moved. If you want to distinguish the signals by color, you can use the automatic color assignment via the button . Alternatively, you can also select the color in the signal table.



If you want to separate the signals and the Y-axes, drag a signal into a vacant area in a graph or on the X-axis of the recorder window.

6.4 Searching for signals

When working with data files containing a large number of signals, it is difficult to find a specific signal by opening the modules. It is tricky even with linear numbering. This is why the signal tree window offers a search function in the *Search* tab, which you can use to search specifically for signal names, expressions, logical expressions or markers.

The search results remain in the list until they are overwritten by new results or *ibaAnalyzer* is closed. The search results are not stored in the analysis.

using the context menu, you can hide and show columns.

Search options

If you enable the *Search in previous search results* option after a previous search and enter a different search string, only the results from the first search will be searched for the new search string. The new search results overwrite the previous search results.

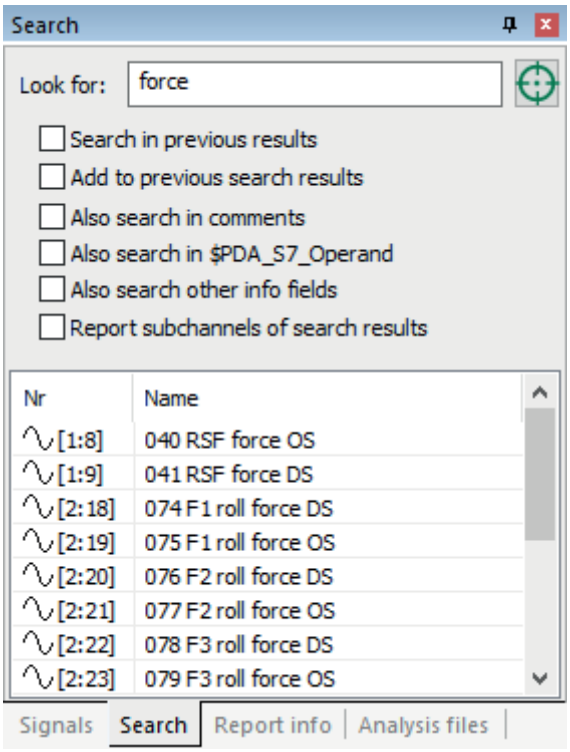
If you enable the *Add to previous search results* option after a previous search and enter a different search string, the old search results will not be overwritten.

If you enable the *Also search in comments* option, the search will also include all signal-linked comments. (Two comments can be assigned to each signal in *ibaPDA*.) If you move the mouse over the signals in the results list, the comments appear as a tooltip.

If you enable the *Also search in $PDA_S7_Operand* option, the S7 operands will also be searched and listed in the results. To do this, you need signals recorded by an S7 controller via *ibaPDA* which are then available as additional info fields.

If you enable the *Also search other info fields* option, the channel info fields belonging to each signal will also be searched.

If you enable the *Report subchannels of search results* option, the subchannels of all result channels will be presented as additional entries.



Procedure

1. Open the *Search* tab in the signal tree window.
2. Enter a search string in the *Search* field above.
   The search string may be an entire signal name or just a part of it. This is a full-text search, so all signals with the specified character string are displayed as results.
3. Choose the desired search options.
4. Press the <Enter> key to start your search.

→ The signals found are listed in the table.

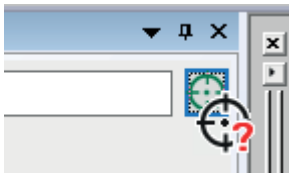
5. You can transfer the signals, expressions or markers to the recorder window for viewing either by double-clicking or using drag & drop.
6. You can refine the search using the *Search in previous search results* and *Add to previous search results* options, and search for a new search string.

Find signals via trend graphs

If you want to identify a specific signal from a set of graphs and locate it in the signal tree, the targeted search function is available.

To use targeted searching, follow these steps:

1. In the signal tree window *Search* tab, click on the focus symbol at the top right.
   The cursor changes shape.



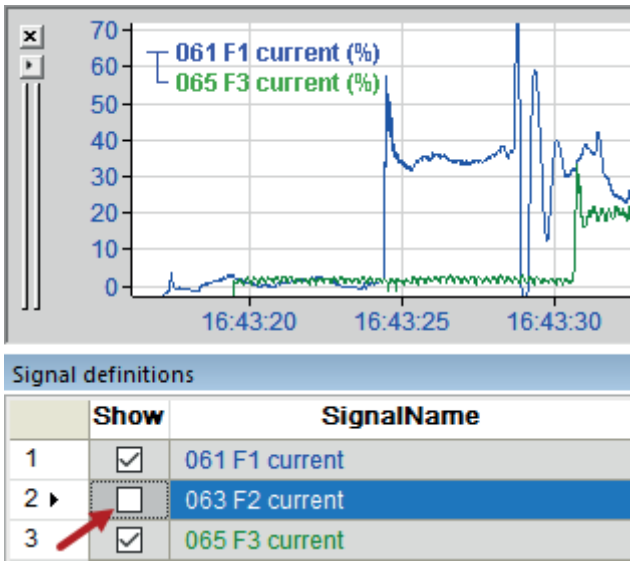
2. Then click with the cursor on the trend graph of the desired signal.

→ The signal appears in the search results list.

6.5 Hiding signals

You can hide signals for display in the graph without removing them from the analysis. This keeps the graphs clearer if you only need signals for calculations (in expressions), e.g. for intermediate results of large calculations.

In the signal table in the *Signal definitions* tab, you can switch the display of signals in the *Show* column on and off. If you deselect the checkbox for a signal in this column, the signal is no longer displayed as a curve.



6.6 Removing signals

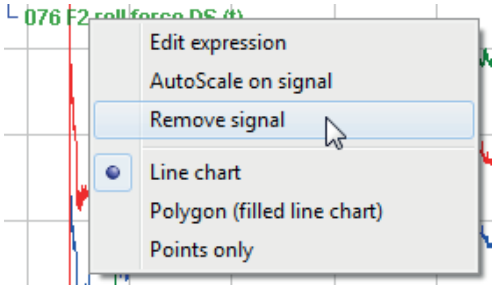
Note

Note that calculated signals (expressions) that you have defined in the signal table are then permanently deleted. You can only use the *Undo* function to restore a deleted calculated signal.

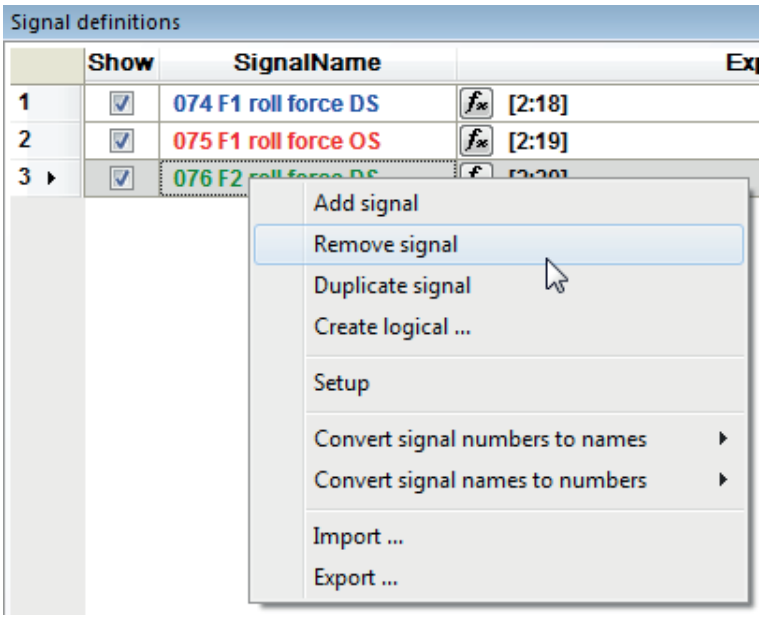
Calculated signals from the data file or the *Logical Expressions* dialog are not deleted when they are removed from the graph.

You have multiple options for deleting a signal:

- Drag the signal outside the recorder window and drop it there.
- Open the context menu on a signal name in the recorder window and select *Remove signal*.



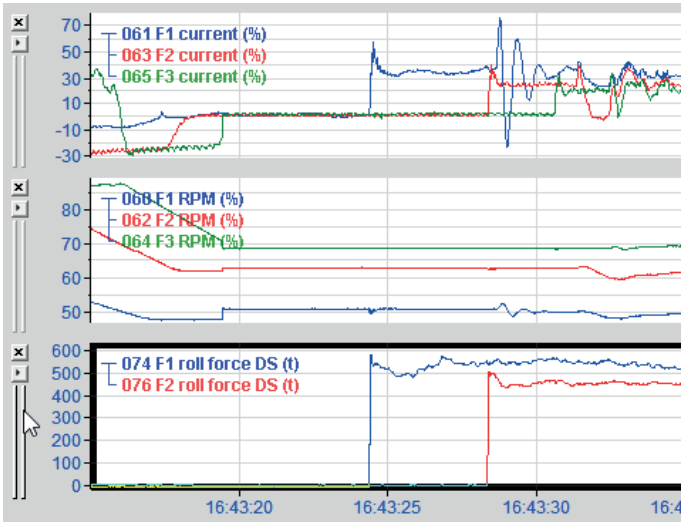
- Open the context menu on a Y-axis and select *Remove axis*. Note that all signals belonging to that Y-axis are deleted.
- Open the context menu on a signal in the signal and select *Remove signal*.



6.7 Moving graphs

You can change the order of the graphs in the recorder window. Note that the order of the signals in the signal table change accordingly.

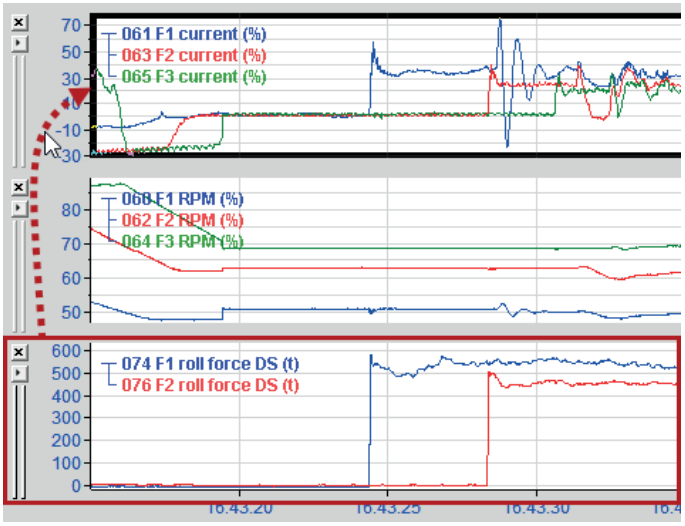
1. Hold down the mouse button on the header bar of the desired graph to the left of the Y-axis so that a thick black frame appears.



2. Drag the graph to the desired position.

   Initially, only the black frame moves and not the content. This indicates above which graph the moved graph will be inserted.

   For example, to place the graph at the topmost position, the current topmost graph must be framed.



3. Release the mouse button at the desired position.

→ The graph is inserted at this position. All other graphs move to the corresponding free positions.

6.8 Hiding graphs

To improve clarity, you can hide graphs without removing them and their signals from the analysis. To do this, click on the small arrow at the top right of the graph. The arrow remains visible in the recorder window and points downwards to indicate hidden graphs.

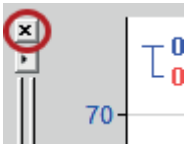


When a graph is hidden, the check marks in the *Show* column are removed for all its signals. This means, you can hide a graph by deactivating all its signals.

6.9 Removing graphs

You have multiple options to remove a graph:

- Click the small cross in the upper left corner at the header bar.



- Open the context menu in a free area of the graph and select *Remove graph*.

6.10 Scale signals

You can change the scaling of signals in the Y-direction in the recorder window by moving the scale ends using the mouse, see ↗ *Shift scales, page 108*. You also have more option in the menu *Graph setup* or the context menus, see ↗ *Y-Axis*, page 66.

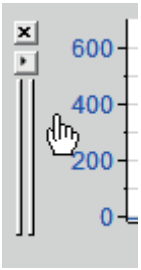
6.11 Y-axis

The signals in a graph can have a common Y-axis or different Y-axes. Common Y-axes are created when you link one signal to another, see also ↗ *Moving signals, page 100*.

If you want to separate the signals and the Y-axes, drag a signal into a vacant area in a graph or on the X-axis of the recorder window.

6.12 Shift scales

To adapt the view of a curve and make details more visible, you can adjust the scales.



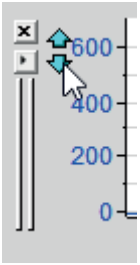
To shift the Y-axis, move the cursor over it until the hand symbol appears. While holding the mouse button down, shift the scale up or down.

The X-axis can be shifted in the same way in zoom mode.

6.13 Compressing and stretching scales

To adapt the view of a curve and make details more visible, you can both compress and stretch the scales.

1. Position the cursor in the upper area of the Y-axis until two blue arrows appear.

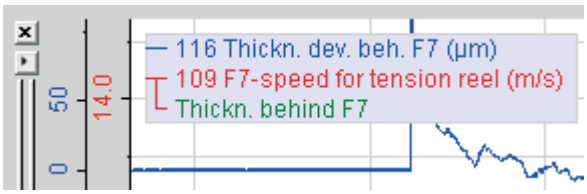
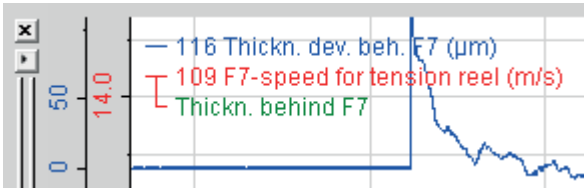
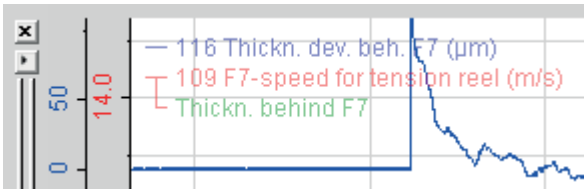


2. Click on one of the arrows and hold down the mouse button to compress or stretch the Y-scale until the desired scaling is achieved.

Alternatively, you can use the mouse wheel: Place the cursor on an axis and turn the mouse wheel to change the scaling.

6.14 Formatting the legend

The legend of the signals is displayed by default in the top left-hand corner of the graph. You can adjust the view of the legend as required. For better legibility, you can select an opaque background. However, if it is more important to see the signals in the legend area, you can make the legend transparent.

	Legend without transparency
	Legend with transparent background and 100% opacity (Slide control in the settings on the far right)
	Legend with transparent background and 50% opacity (Slide control in the settings in the middle position)

You can find the corresponding settings in the preferences or the graph setup under *2D view*.

In addition to the appearance, you can also customize the content of the legend. By default, only the signal name is displayed, but you can add additional information such as module name, comments or marker values.

For more information, see ↗ *2D View*, page 72.

6.15 Zoom in and out

When you zoom within a graph, all other graphs that use the same X-axis base are also zoomed. Graphs with a different X-axis base remain unchanged. An exception to this is the FFT display: It follows the zoom factor and shows the FFT for the zoomed area while the frequency axis remains unchanged.

Zoom in

You can zoom in anywhere in a graph.

Hold down the left mouse button and drag a rectangle over the desired area. When you release the mouse button, the selected area is enlarged.

Zooming affects both the X- and the Y-directions.

When zoomed in, you can adjust the scaling in the Y-direction at any time without affecting the zoomed section of the X-axis. Autoscaling in the Y-direction applies only to the values in the zoomed area.

If you hold down the <Shift> key while zooming, the zoom frame remains limited to the height of the graph.

Zoom out

You can zoom out of a zoomed view using two different buttons.

Icon	Function	Description
	Zoom out	Zooms out one stage (incrementally) from a zoomed-in view. The command affects the currently selected graph and all other graphs that have the same X-axis base (time, length, FFT).
	Zoom out all	Resets all zoom factors in all graphs, regardless of which graph is selected and whether there are different X-axes.

Alternatively, the context menu in the graph also offers the *Zoom out* function.

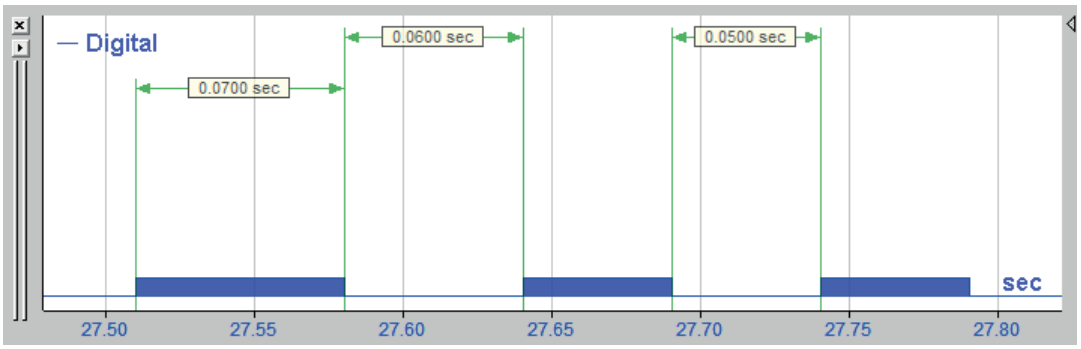
Zoom with mouse

Alternatively, you can use the mouse wheel: Place the cursor on an axis and turn the mouse wheel to change the scaling.

6.16 Intervals in graphs

You can use intervals to measure sections on the X-axis and display the values in the X-axis unit in the graph.

An interval is similar to a dimension line and consists of two vertical lines between which there is a horizontal line with arrow ends. On this line there is a label indicating the distance between the vertical lines in X-axis units. If there is not enough space between the vertical lines, the label is displayed next to it.



The intervals are directly linked to the signal. If you move the signal to another graph, the intervals are also transferred.

Simple application options and the more complex configuration dialog are described below.

6.16.1 Showing digital signal intervals

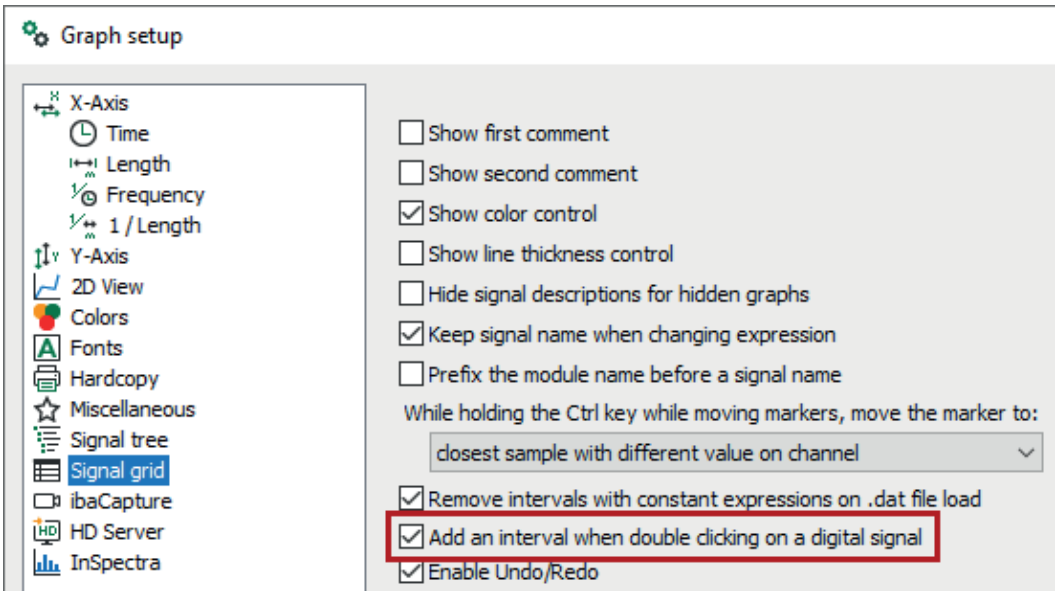
You can use intervals to easily display the on or off times of a digital signal. To do this, double-click on the area where the signal is True or False.

Depending on where you click, the interval between the rising and falling edge, or vice-versa, is displayed.

If you press the <Ctrl> key while double-clicking, all intervals are displayed with the same value (TRUE or FALSE, depending on the position).

Tip

The function to add intervals by double-clicking on a digital signal is enabled by default. You can disable the function in *Graph Setup* or *Preferences* under the *Signal grid* node:



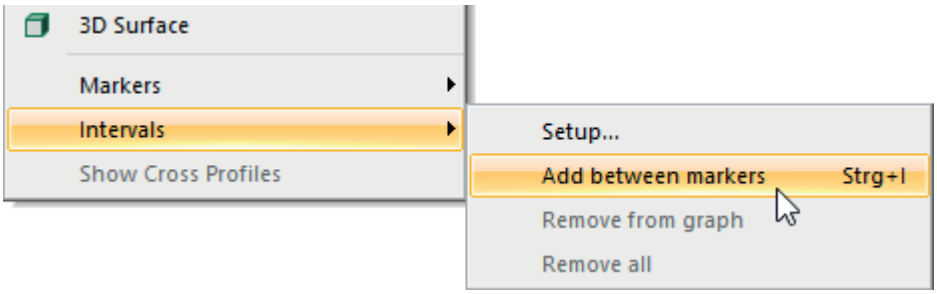
6.16.2 Showing intervals between markers

You can also use the interval function together with the markers. By doing so, you can measure any sections on the X-axis by positioning markers and then displaying the interval between the markers. This allows you to easily measure analog signals and distances between different signals.

To do this, use the interactive markers X1 and X2, see ↗ *Interactive markers, page 120*.

When in the marker view, you can add an interval between the markers X1 and X2 in the selected graph as follows:

- Menu *Graph Mode – Intervals – Add between markers*
- Context menu in graph: *Intervals – Add between markers*



6.16.3 Configuration of the intervals

You can also calculate and additionally configure intervals in a dialog. The following options are available to access the Channel intervals dialog:

- Menu *Graph Mode – Intervals – Setup*
- Context menu in graph: *Intervals – Setup*

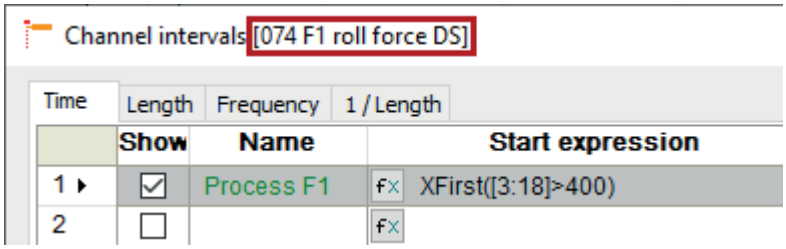
In this dialog, you can configure multiple intervals per domain (time, length, frequency, 1/length) for each signal. You can define the start and stop positions of the intervals either dynamically, depending on any signals, or constantly.

If you view multiple data files of the same type in succession, such as via the file group slide show, calculated intervals are always automatically positioned correctly.

Note

Note that each signal in the graph has its own interval dialog! The entries you make in this dialog always relate to the signal that you last clicked on.

The header shows the corresponding signal.



You have the following setting options in the dialog.

Show
If you want the interval to appear in the graph, enable the interval here.

Name
Optionally, enter a name to be displayed in the interval label before the value. If you leave the field empty, no name will appear.

Start expression
Enter an expression to be used to define the beginning of the interval on the X-axis. You can use any signals and conditions. The expression can also be a constant.

Stop expression
Enter an expression to be used to define the end of the interval on the X-axis. You can use any signals and conditions. The expression can also be a constant. If the expression results in a smaller value than the start expression, the distance is shown negatively.

Color
Optionally, assign a color to the intervals. By default, the third color from the color settings is used. The other color selection also corresponds to the colors in the color settings (*Preferences*).

Start value, stop value, and difference
These columns show the currently calculated values for the start and end positions of the interval as well as the distance, which is then also displayed in the label. For new intervals, the values are only displayed when you click the <Apply> button.

<Show all> and <Hide all>
Enable or disable all intervals in the *Show* column.

<Remove all>
Remove all intervals in the table.

<Remove selection>
Remove only the previously marked intervals. Mark an interval in the table with a mouse click on the cell in the first column (Number). Multiple selection is possible with the <Ctrl> or <Shift> key. When you click in another column, you remove the values from this cell.

<Add from markers>
The positions of markers X1 (start) and X2 (stop) are adopted, and entered in the marked row.

<Apply>
Apply all changes without closing the dialog.

Tip



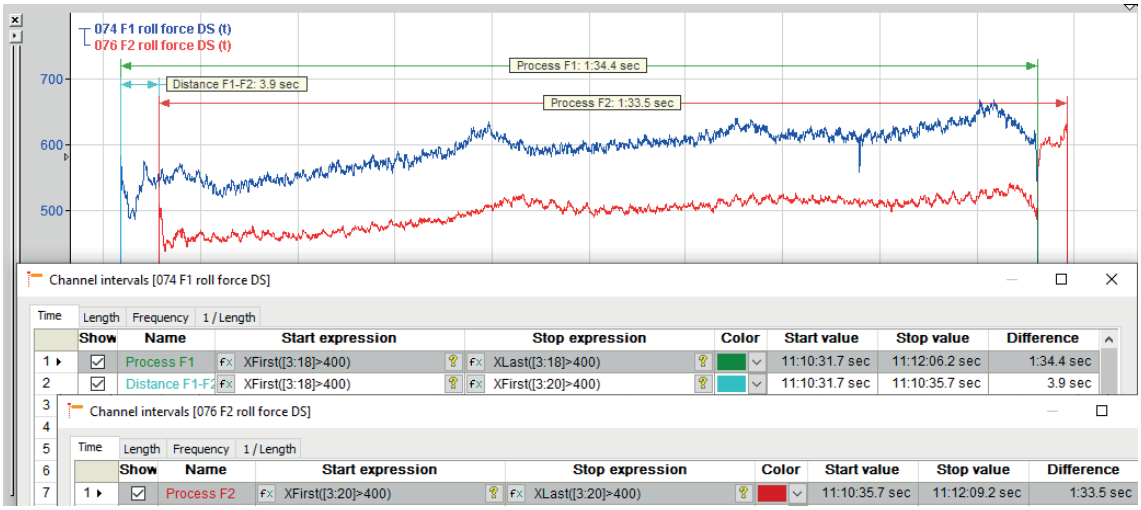
You can change the font and font size for the label in *Preferences* or *Graph setup* under *Fonts – Marker labels*.

Example: Dynamically calculated intervals

The interval indicates the duration of the rolling process dependent on certain states of the rolling force signal.

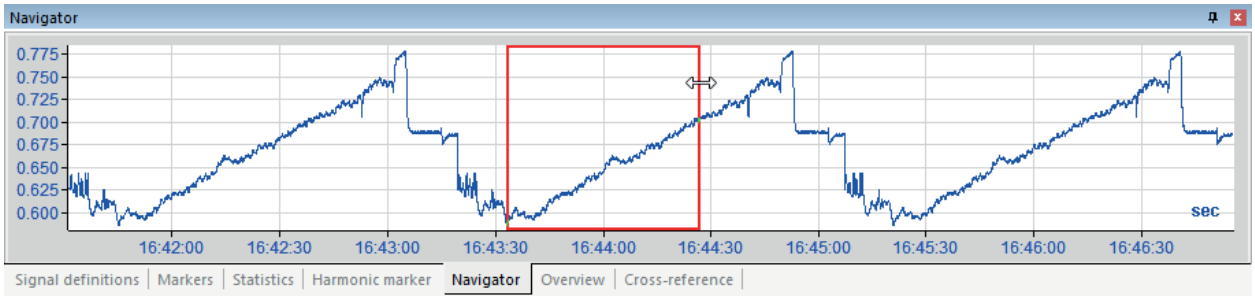


A separate interval has been defined for a second signal (red). The image below shows that only the intervals of a signal are included in the interval dialog. Finally, another interval has been defined for the first signal (blue) displaying the distance between the starts of both rolling processes F1 and F2. You can use any signals and conditions to determine the start and stop positions.



6.17 Using the navigator

The *Navigator* tab always shows the full content of the data file with a time or length axis for the graph that is at the topmost position in the recorder window.



The red frame in the window represents the section that is visible in the recorder window. In turn, you can adjust the red frame to zoom in to the graph.

The navigator window always shows the time- or length-based signal, but never the FFT view. When a signal is displayed in the FFT view, the FFT is calculated from the visible samples on the time or length axis. When moving or zooming via the frame, the zoomed section of the FFT view adapts accordingly.

For FFT views, it is helpful to set the frame width in the X-direction. To do this, the *Setup fixed range* function is provided in the context menu of the navigator, see ↗ *Navigator X-range, page 117*.

Select zoomed section

1. Position the cursor on the red frame line.

→ The cursor is now a double arrow.

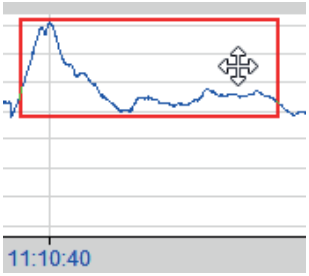
2. Drag the frame to the desired size.

→ When shifted in the X-direction, all graphs are adjusted with the same X-axis mode. When shifted in the Y-direction, only the topmost graph is adjusted.

Move zoomed section

You can move the minimized red frame in various ways to adjust the zoomed image section.

- Position the cursor within the red frame line and move the frame to the desired position.



- Use the keyboard arrow keys to move the frame to the left and right.
- Click on the desired signal position in the navigator, and the red frame will jump there. The center of the frame is then at the mouse position.

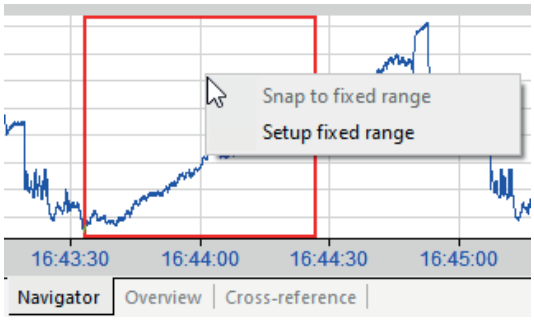
→ When shifted in the X-direction, all graphs are adjusted with the same X-axis mode. When shifted in the Y-direction, only the topmost graph is adjusted.

6.17.1 Navigator X-range

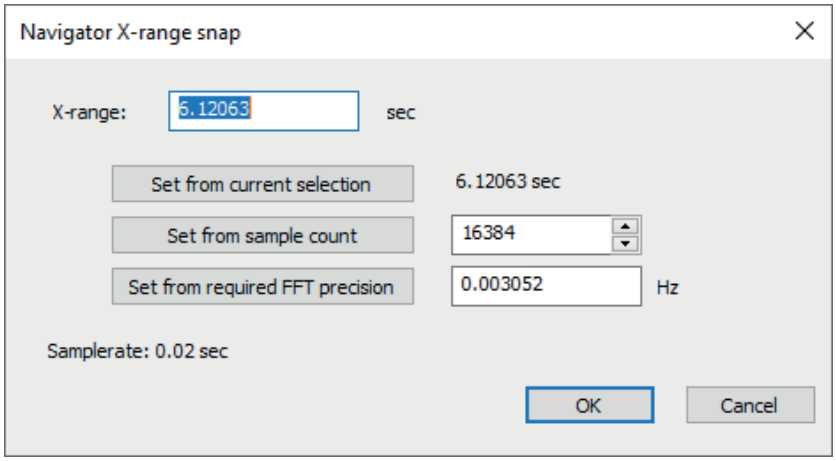
A fixed range width in the X-direction is helpful for FFT operations. When a signal is displayed in the FFT view, the FFT is calculated from the visible samples on the time or length axis (rounded to the nearest power of 2). When moving or zooming via the frame, the zoomed section of the FFT view adapts accordingly.

You can therefore set the frame width so that the frame always contains the desired number of measuring points.

To do this, select the *Setup fixed range* function from the context menu of the navigator. You can then set the frame accordingly in the dialog so that the width remains the same in the X-direction.



Setting options in the dialog



X-range
Enter the desired X-range here if you know it.

Alternatively, you can derive the range setting from different parameters, see the following options.

<Set from current selection>
The X-range corresponding to the current setting of the navigator frame is adopted.

<Set from sample count>
Enter the desired number of samples (measuring points) in the field. Click on this button to set the corresponding frame width.

<Set from required FFT precision>
Enter the desired FFT precision in the field as a frequency. The frame width is then set so that it contains a sufficient number of measuring points to obtain FFT data for each multiple of the entered frequency between the minimum and maximum frequency. (The minimum and maximum frequencies are specified in the frequency axis settings.)

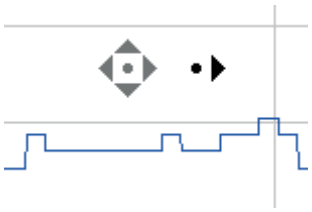
Note

ibaAnalyzer can adjust all values that you enter so that they correspond to a power of 2 or guarantee a minimum number of 128 measuring points.

Only the applied value in the *X-range* field is saved. All other parameters are reset to their default values when the dialog is closed.

6.18 Autoscrolling

When zooming in on a signal trend, you can use the context menu *Start autoscrolling* function. The autoscroll function is particularly useful with a high zoom factor to capture measuring points precisely.



When active, a scroll symbol appears in the graph which serves as a reference point.

When you move the cursor to the left, right, above or below the icon, the graph scrolls in that direction. This allows you to move through the signal trend.

Note that scrolling in the Y-direction no longer works if you used autoscaling while zoomed.

Alternatively, you can switch on autoscrolling in the graph by clicking the mouse wheel.

To switch off autoscrolling, click in the graph.

6.19 Markers

ibaAnalyzer provides three classes of markers to support you in analyzing graphs:

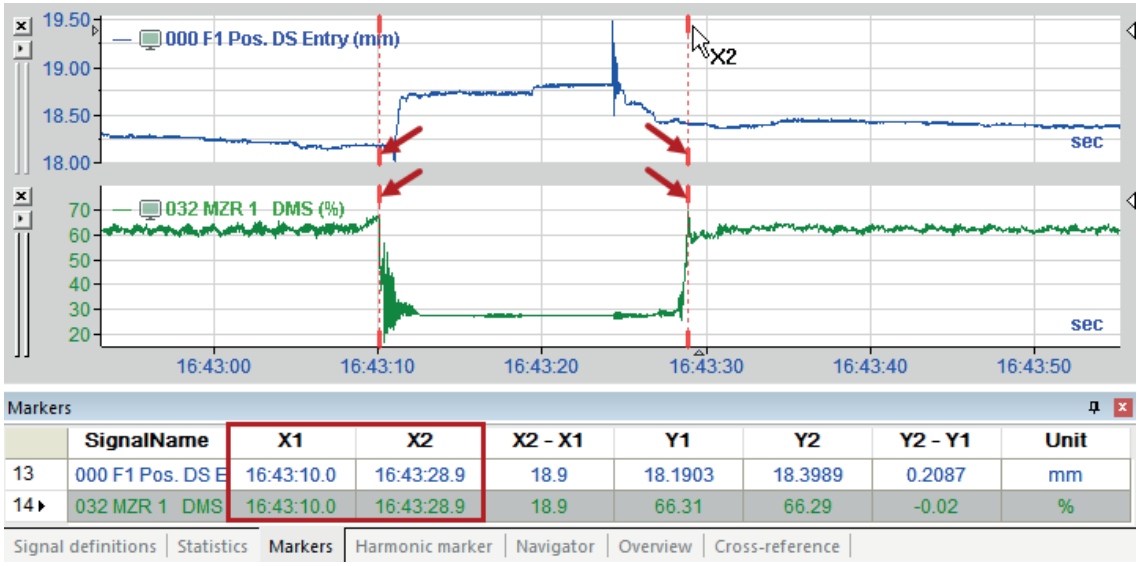
- ↗ *Interactive markers, page 120*
- ↗ *Harmonic markers, page 120*
- ↗ *X-axis markers (calculated markers), page 123*

For more information, see *ibaAnalyzer* manual part 1, chapter *Markers tab*.

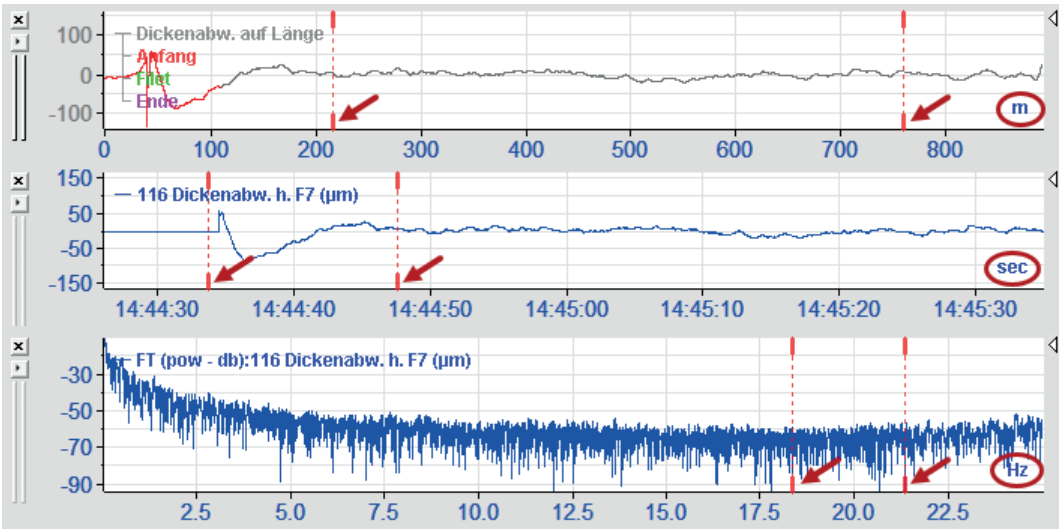
Markers and X-axes

The markers depend on the specific X-axis and its mode. Marker functions are always executed in relation to the common X-axis.

If multiple graphs with the same X-axis mode are open in the recorder window (e.g. length-based or time-based), the marker functions are executed in the same way in all graphs.



If multiple graphs with different X-axis modes (time, length, frequency or 1/length) are open in the recorder window, a separate pair of markers is assigned to each graph. The marker functions then apply to the respective graph.



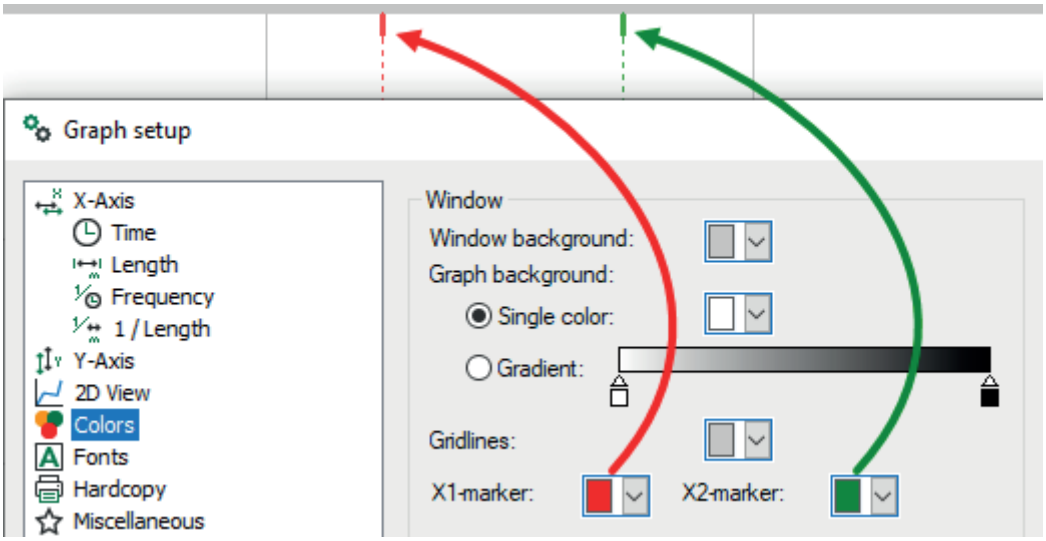
6.19.1 Interactive markers

The interactive markers appear as soon as you select the *Markers* or *Statistics* tab in the signal table window, or open them as separate windows.

For more information, see *ibaAnalyzer* manual part 1, chapter *Markers tab*.

Color control

You can assign custom colors to markers X1 and X2 in the Preferences or Graph setup under the *Color* node. This helps avoid confusion when reading.



6.19.2 Harmonic markers

The *Harmonic markers* tab in the signal table window is available to provide advanced FFT support. When you open the tab, markers similar to the markers in the *Markers* and *Statistics* tabs appear in each graph featuring FFT display mode.

Harmonic markers table

The table in the *Harmonic markers* tab shows Y-values for each signal, for the harmonic markers (integer multiples or parts), and for the sideband markers. If markers are not visible, "--" is displayed in the table.

Harmonic markers in the graph

There are different harmonic markers for the various X-axis modes, which you can configure and move for each X-axis:

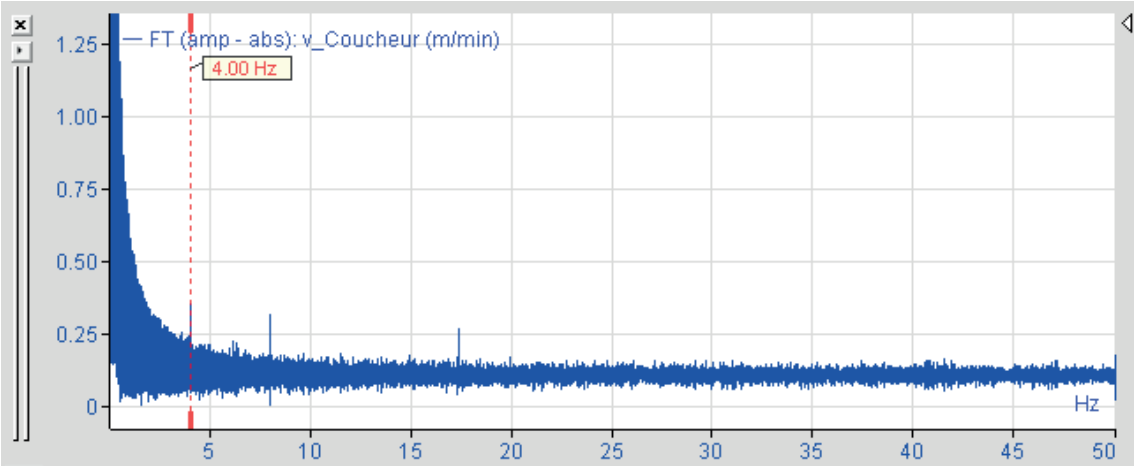
- Markers for frequencies (Hz)
- Markers for inverse lengths (1/m)

There are different types of harmonic marker representation in the graph:

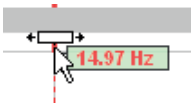
- Main marker (marker of the main frequency)
- Harmonic markers of the main marker
- Sideband markers of the main marker

Main marker

The main marker indicates the main frequency, and forms the basis for displaying the harmonic markers and the sideband markers. The main marker has thick line ends compared to other harmonic markers.

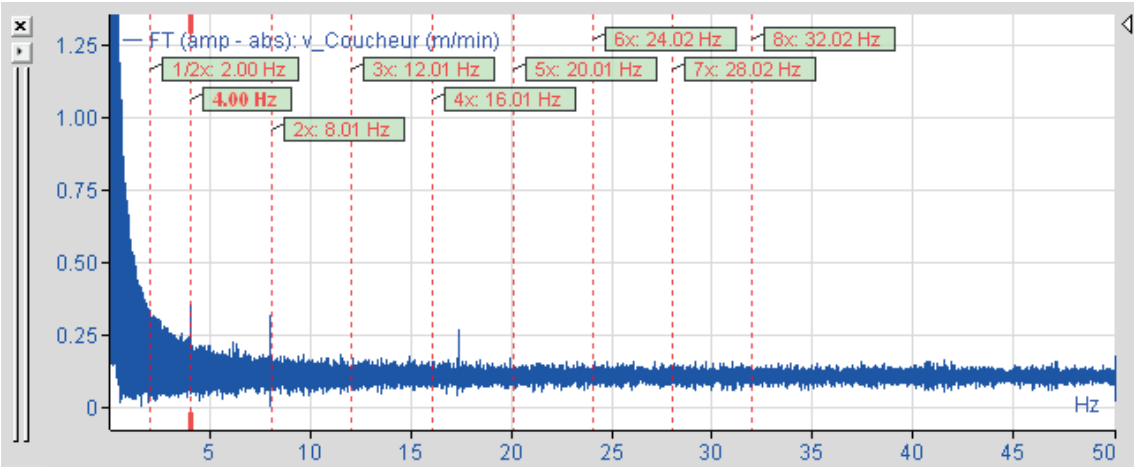


You can move the main marker along the X-axis at the top and bottom. A move mode icon is displayed:



Harmonic markers of the main marker

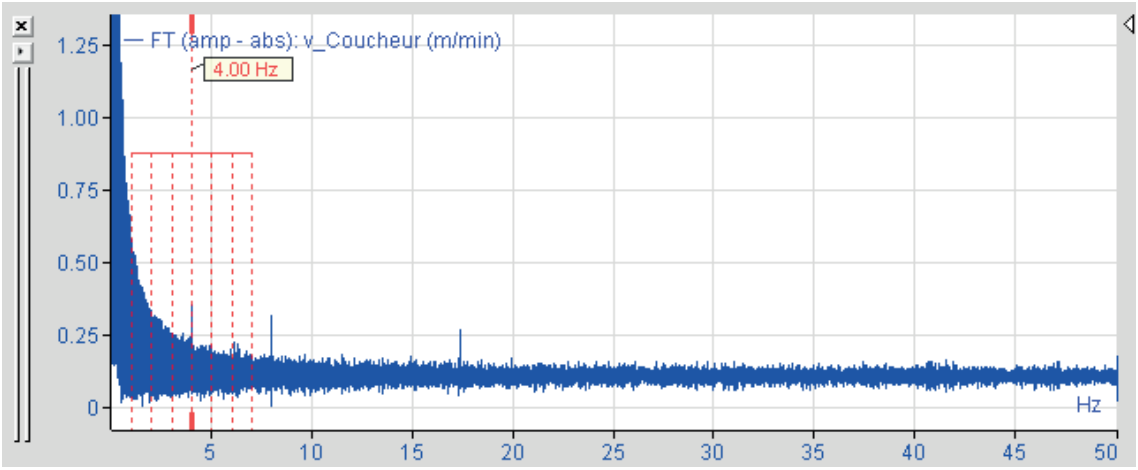
The harmonic markers are located on the harmonics of the main frequency, and are a multiple or a portion of the main frequency. The harmonic markers are also represented by dashed lines, but do not have thickened ends.



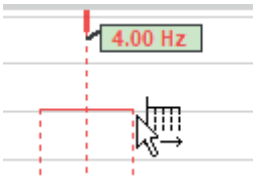
You cannot move these markers, as their X-position depends on the position of the main marker (e.g. 2×, 3×, ½×). If you move the main marker, the harmonic markers follow proportionally at the corresponding distance.

Sideband markers of the main marker

You can use sideband markers to mark sidebands of the main frequency. They are applied symmetrically to the right and left of the main marker in customizable numbers. They are slightly shorter than the other markers and are interconnected at the top by a horizontal line. Sideband markers do not have labels.



You can change the distance between the sideband markers by moving the outermost sideband marker.



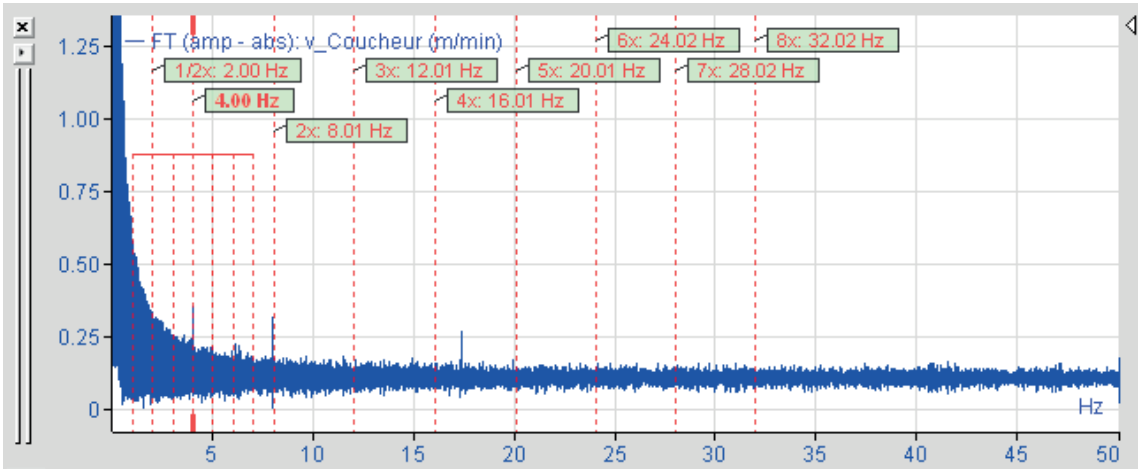
Switch marker types and labels on and off

The main marker is always visible. If you click on the label of the main marker, you can switch the other markers on and off. If no other markers appear, you can adjust the settings, see ↗ *Configuration of the harmonic markers, page 123*.

The following table describes what happens when you click on the label of the main marker. The starting point is that only the main marker is visible.

Click	Function
1	Displays harmonic markers with labels
2	Hides labels of the harmonic markers
3	Displays harmonic markers with labels and sideband markers
4	Hides labels of the harmonic markers
5	Hides harmonic markers
6	Hides sideband markers

A mouse click on the label of a harmonic marker hides the marker label.



Configuration of the harmonic markers

You have various setting options for the markers in the Graph setup or Preferences in the *X-axis – frequency* or *1/length* nodes.

- number of harmonic markers to be displayed below and above the main frequency
- enable and disable sideband markers
- distance between sideband markers (in Hz or in 1/m or 1/inch)
- number of sideband markers to be displayed (symmetrical to main frequency)

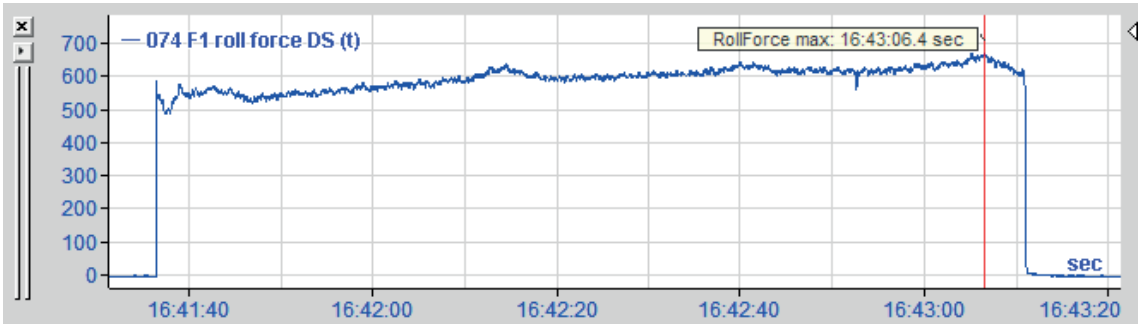
If you change the number of harmonic markers, the number of harmonics in the signal table is adjusted accordingly.

For more information on the settings, see ↗ *X-axis – Frequency*, page 64.

6.19.3 X-axis markers (calculated markers)

You can define additional markers for each graph and each X-axis type of a view (based on time, length, frequency or 1/length). X-axis markers indicate certain X-positions in a graph at which certain conditions are fulfilled or events take place. Such events include, for example, a signal minimum or maximum, the first violation of a limit value, or the rotational frequency of a specific roller.

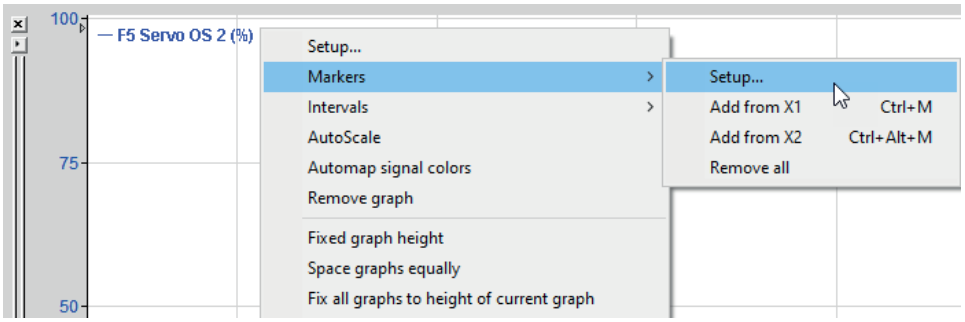
The markers appear as continuous vertical lines in the graph. The markers are red by default, but you can customize their colors. The marker label shows the respective X-value.



The markers are saved in the analysis(PDO file).

Defining markers

1. Open the context menu of a graph and select *Markers – Setup*.

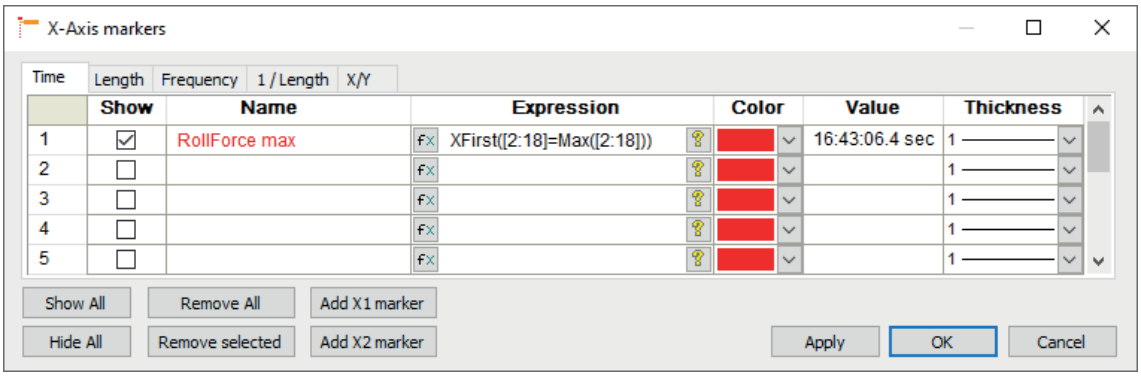


→ The *X-axis markers* dialog opens up.

2. Enter the name and calculation here.

→ The marker is automatically enabled for display.

You can adjust the color and line thickness as required.



3. Confirm the settings with <OK>.

If you only click <Apply>, the X-value appears in the column and the dialog stays open.

→ The marker appears in the graph.

User interface of the dialog

The dialog opens showing the tab that corresponds to the X-axis mode of the graph you clicked on (time, length, frequency, 1/Length or X-Y).

Show column

Enable the marker if you want to display it.

Name column

Enter a name for the marker.

Expression column
Here you can enter an expression for calculation of the marker position. The <fx> button opens the expression builder. The <?> button helps you to find errors in the expression.

The expression must return a position on the X-axis in order to display a marker. The result of the expression must correspond to the unit of the X-axis in the graph.

For graphs in time-based or length-based FFT mode, the dynamic marker and signal depend on the selection in the navigator window. The marker is displayed at the position on the X-axis that corresponds to the average of the expression in the navigator frame. The navigator has no influence on constant expressions such as "XFirst" or "Max".

Color/Thickness column
Select the color and thickness of the marker here. The default color is red.

Value column
If the marker can be calculated, this column shows the value when you click the <Apply> button and the check mark is set in the "Show" column.

Using markers in graphs

Move
You cannot move X-axis markers along the X-axis. The marker only changes position if you change the associated expression. You can, however, move markers from one graph to another one with the same X-axis. Drag the marker by its label into the other graph.

Markers from signal tree
All defined markers are set out in a tree structure below the signal tree. You can drag the markers from there into a graph with the same X-axis base.

If you want to hide the marker, simply drag it out of a graph. The marker is not removed from the signal tree.

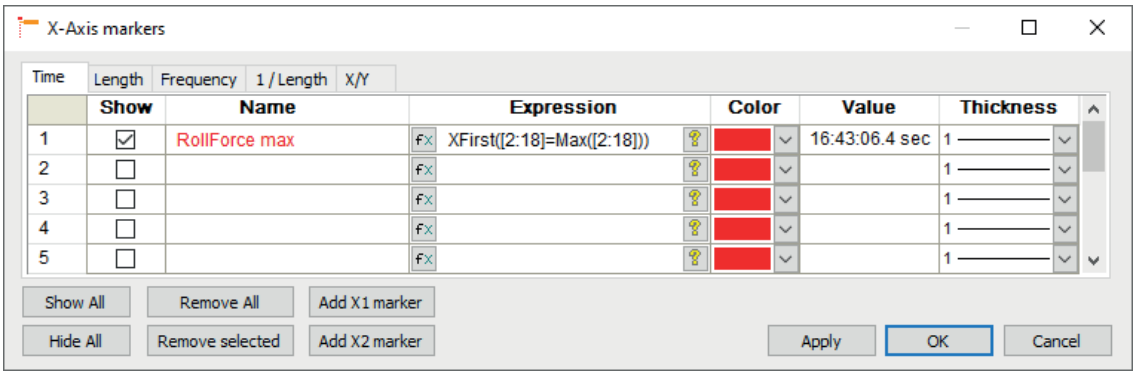
Marker with frequency-based or 1/length-based X-axis
Markers in graphs that are either frequency-based or 1/length-based display the harmonic markers when you click on their legend. The number of markers is the same that you selected for the harmonic markers, see ↗ *Harmonic markers, page 120*.

Calculation examples

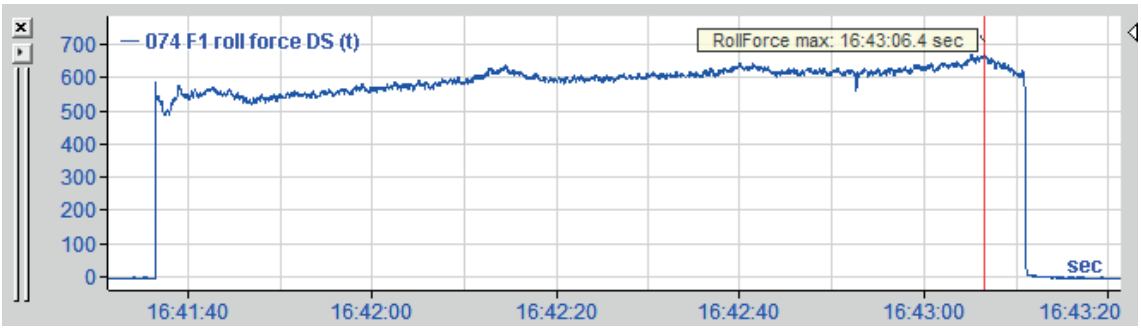
Example: Defining markers for a time-based signal
You want to identify when a signal has reached its maximum. To do this, enter the following expression in the *X-axis markers* dialog:

`XFirst([signal number] = Max([signal number]))`

With "XFirst" you create a reference point on the X-axis, as it is possible that there may be multiple maximums with the same Y-value.



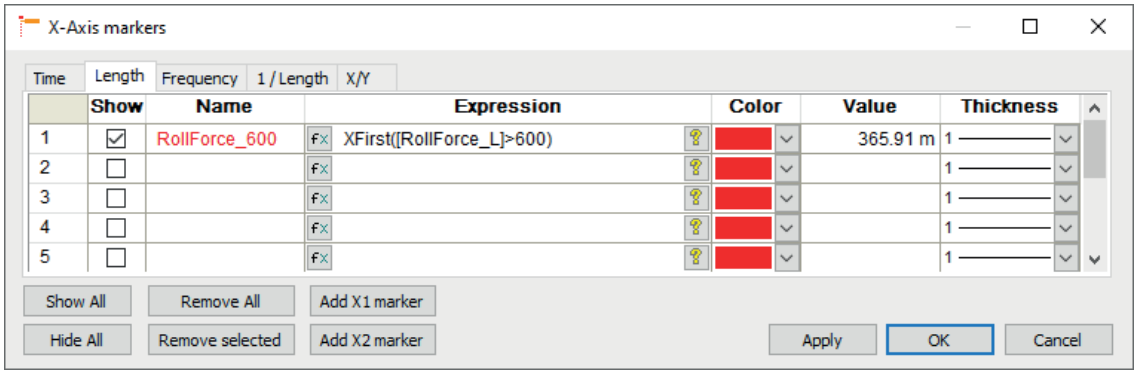
When you click on <OK>, you will see the result as a marker in the graph. The maximum is reached for the first time at 16:43:06.4.



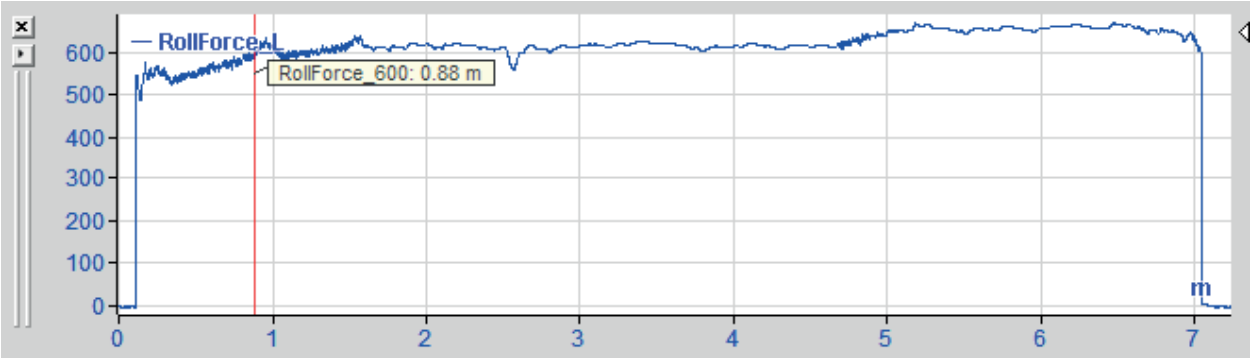
Example: Defining markers for a length-based signal

For a signal you want to identify the length at which the rolling force exceeded the limit value of 600 t for the first time. To do this, enter the following expression in the *X-axis markers* dialog:

```
XFirst([signal name] > 600)
```



When you click on <OK>, you will see the result as a marker in the graph. The rolling force exceeds 600 t at a length of 0.88 m.



6.19.4 Dynamic marker labels

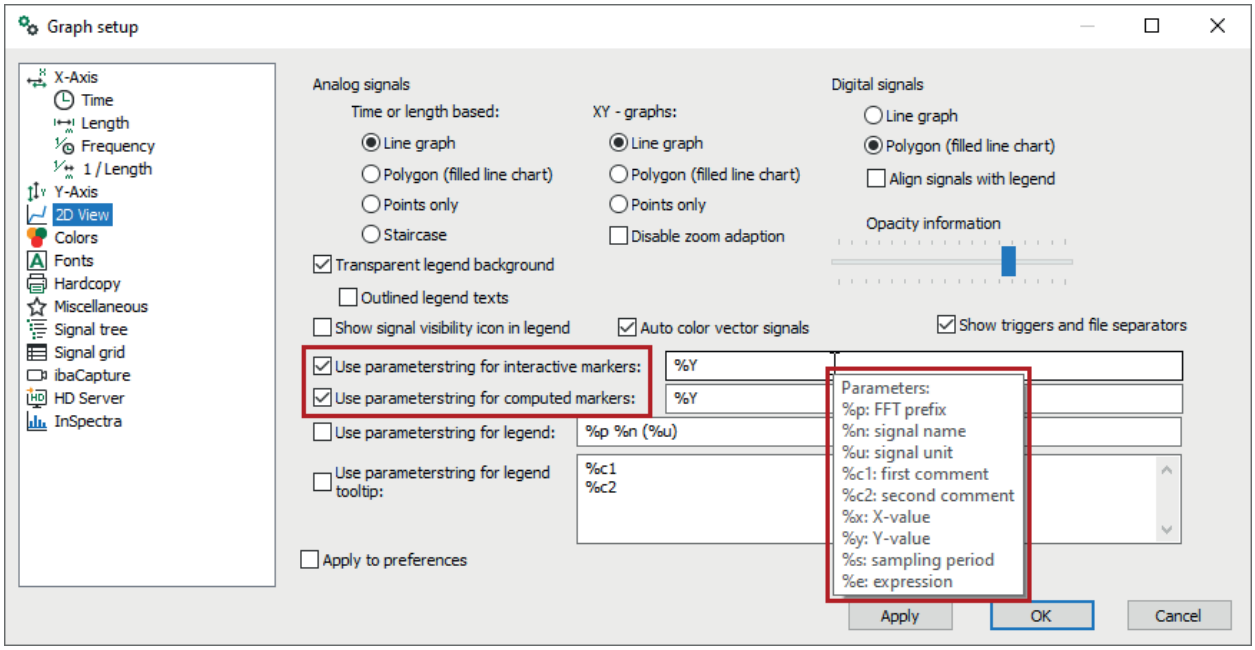
Similarly to the signal legends and tooltips, you can also compose the texts in the labels of the markers dynamically by using placeholders. You can enable or disable this function in the *Preferences* or *Graph Setup* under *2D view*.

A distinction is made between interactive markers (X1, X2) and calculated markers (X-axis markers).

Use parameterstring for interactive/calculated markers
If you enable this option, you can add additional information or comments to the marker text. Enter the parameter strings as placeholders for the various options in the input field. By default, the Y-value is displayed at the marker position ("%y"). You can also enter any text.

The selection of parameter strings appears when you position the cursor in the relevant input field.

For more information on parameter strings, see ↗ *2D View*, page 72.



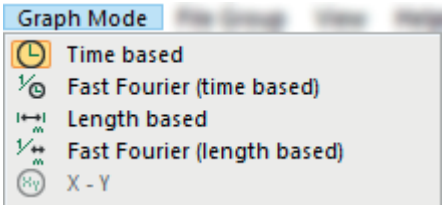
6.20 X-axis modes (reference axes)

You can change the mode of the X-axis, if you want to analyze length-based signals for example. Depending on the type and number of signals, you can choose from the following X-axis modes:

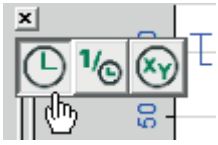
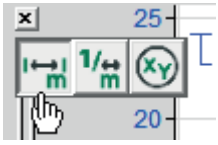
- Time axis (s)
- Length axis (m)
- Frequency axis (1/time, 1/length)
- Signal values (X-Y)

Changing the X-axis mode

The selection in the *Graph Mode* menu always refers to the active graph.



You can also change the X-axis mode of the graph using the triangle button on the graph header.

Selection for multiple time-based signals	Selection for multiple length-based signals
	

Appearance and behavior in the recorder window

All time-based signals share a time axis, all length-based signals share a length axis, and all FFT views share a frequency axis. In the X-Y plot, each graph has its own X-axis. This mode of representation is only possible if there are at least two signals in a graph.

If the X-axis mode does not match the signal reference, no trend curves are displayed.

The scaling of the respective axes is based on the longest signal.

Where graphs have different X-axes, their X-axes are zoomed and shifted independently of each other. When you zoom in a time-based graph, the length-based view remains the same.

The *Marker* tab in the signal table shows the correct cursor positions for each X-axis.

6.20.1 X-axis modes Time based and Length based

The time mode is used for time-based signals, the length mode for length-based signals. If the X-axis mode does not match the signal reference, no trend curves are displayed.

X-axis mode Time

By default, *ibaAnalyzer* uses the time-based X-axis mode because most signals are recorded on a time basis.



X-axis mode Length

There are no length-based signals in the standard *ibaPDA* data format (DAT file). You can only record length-based signals with *ibaQDR* and *ibaHD-Server*.

In *ibaAnalyzer* you can, however, calculate length-based signals from time-based signals using various functions, such as with "TimeToLength", see *ibaAnalyzer* manual part 3, chapter *Conversion from time to length reference (TimeToLength)*.



You can also load length-based signals into *ibaAnalyzer* via a database query, see *ibaAnalyzer* manual part 4, chapter *Analysis of data from databases*. (To read or query databases you need the *ibaAnalyzer-DB-Read* license.)

6.20.2 X-axis mode FFT

Fast Fourier Transformation (FFT) is a mathematical method of Fourier transformation, and a faster variant of Discrete Fourier Transformation (DFT). This method transforms signals into the frequency range. FFT is used to break down periodic signals into individual sine oscillations which are then broken down further into the corresponding spectral frequencies.

FFT mode in ibaAnalyzer

FFT mode in *ibaAnalyzer* generates an FFT analysis for one or more signals within a graph. The result shows the distribution of the oscillations in the signal.

The Y-axis becomes the (frequency) amplitude axis, whilst the X-axis becomes the frequency axis (Hz = 1/s).

When calculating, a Power Spectrum FFT is performed using the squared average amplitude algorithm (default setting). You can adjust the calculation bases and algorithms for FFT in the Preferences and Graph setup, see ↗ *Fast Fourier*, page 70.

This display mode does not offer the comprehensive analysis functions that are available with an *ibaAnalyzer-InSpectra* FFT view, see ↗ *FFT view (ibaAnalyzer-InSpectra), page 141*.

Time-based and length-based FFT mode

In addition to the time-based FFT mode, you can also convert length-based signals as FFT. In this, the X-axis becomes a length/frequency axis (1/m), i.e. an inverse length axis.



Time-based FFT mode



Length-based FFT mode

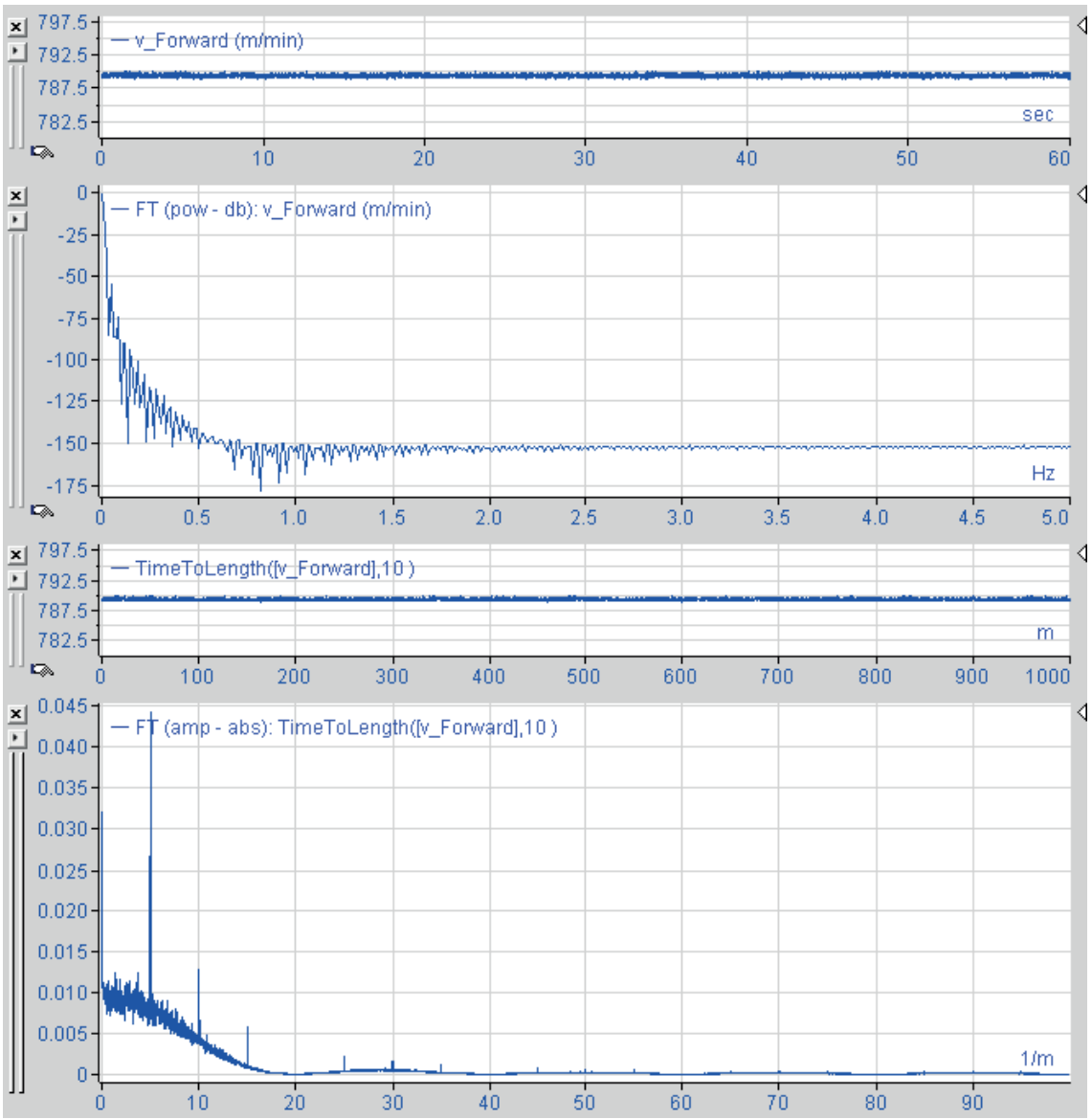
This representation mode is useful for analyzing certain phenomena that occur periodically over the length of the measurement sample, such as thickness variations in rolled sheet metal.

Example: Comparison of time-based FFT and length-based FFT

The example in the image below shows a speed measurement signal (segment over 60 s) with superimposed noise or other frequencies.

The time-based FFT in the second graph provides hardly any useful results.

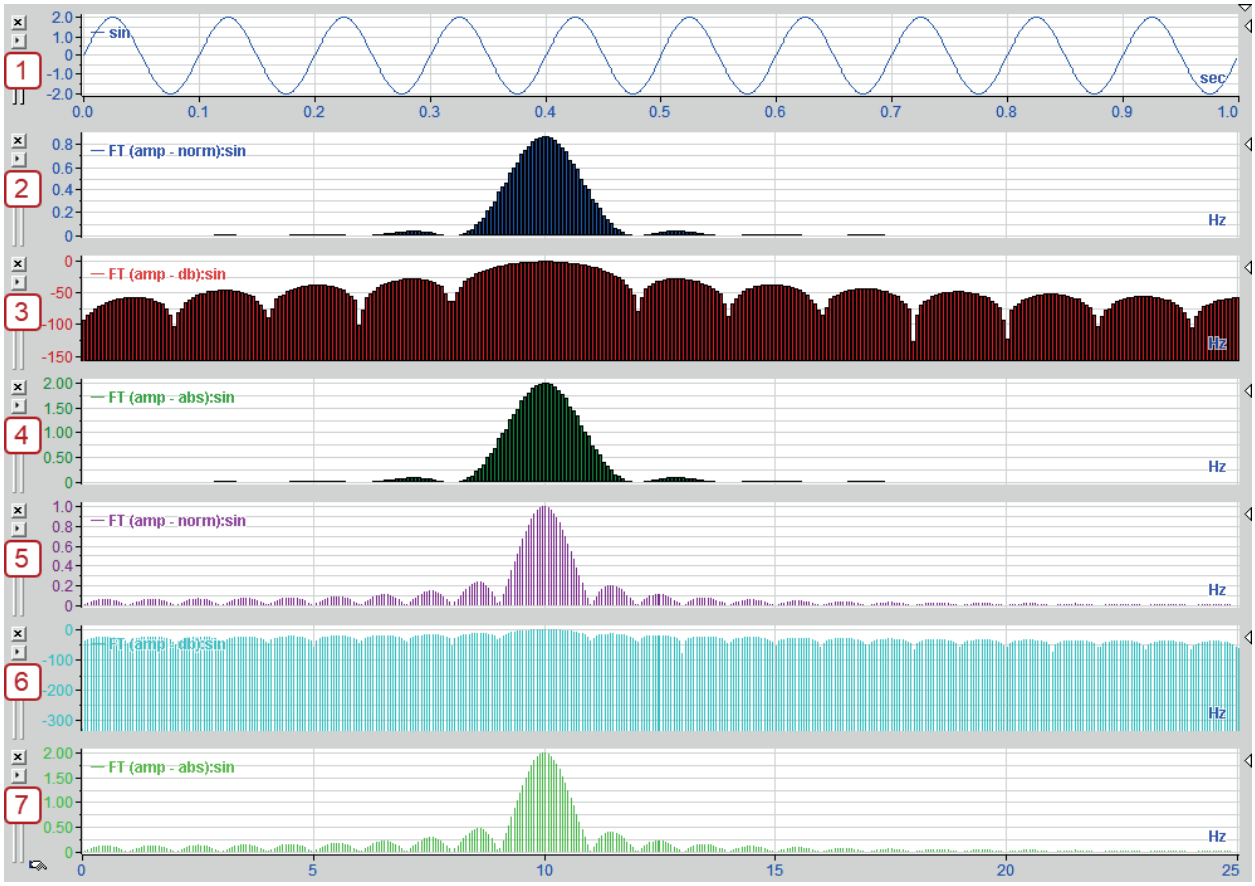
After conversion into a length-based signal, the FFT delivers a result that shows a clear peak.



Examples of the settings

The various calculation bases and display options in the settings produce different FFT results.

The image below shows a sine wave (f = 10 Hz) and the FFT of the signal with different settings.



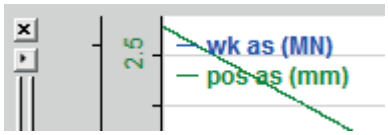
1	Original curve
2	Amplitude, normalized (linear), Bartlett, bar
3	Amplitude, normalized (db), Bartlett, bar
4	Amplitude, absolute, Bartlett, bar
5	Amplitude, normalized (linear), rectangle, discr. frequency
6	Amplitude, normalized (db), rectangle, discr. frequency
7	Amplitude, absolute, rectangle, discr. frequency

6.20.3 X-axis mode X - Y

You can use the X-Y plot to visualize the dependency of multiple time-based or length-based signals. The time or length dimension is eliminated during this process. There must be at least two signals in a graph for you to select X-Y mode. You cannot mix time-based and length-based signals.



The lower signal in the legend is plotted on the X-axis, the other signals on the Y-axis. You can use multiple Y-axes.

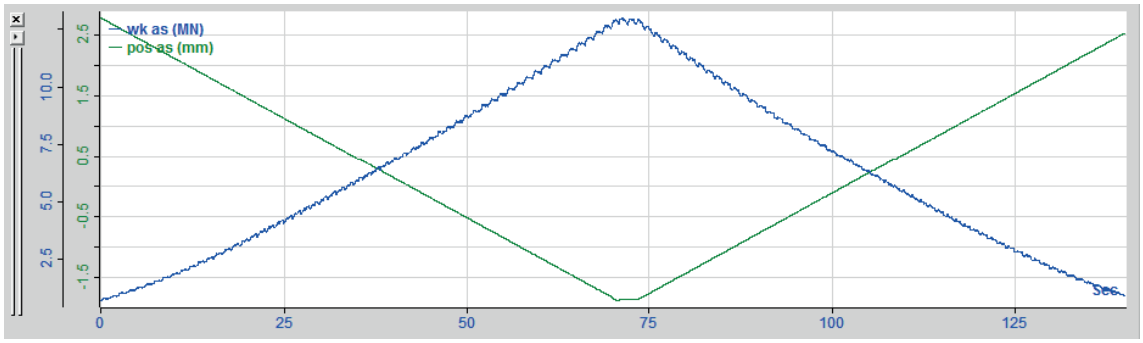


You can use the mouse to change the order of the signals in order to plot a different signal on the X-axis, see ↗ *Moving signals, page 100*.

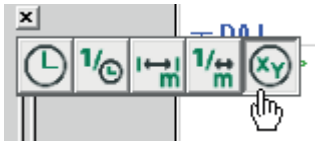
Example of the procedure: Stand characteristic (roll force vs. position)

1. Drag at least two signals into a graph.

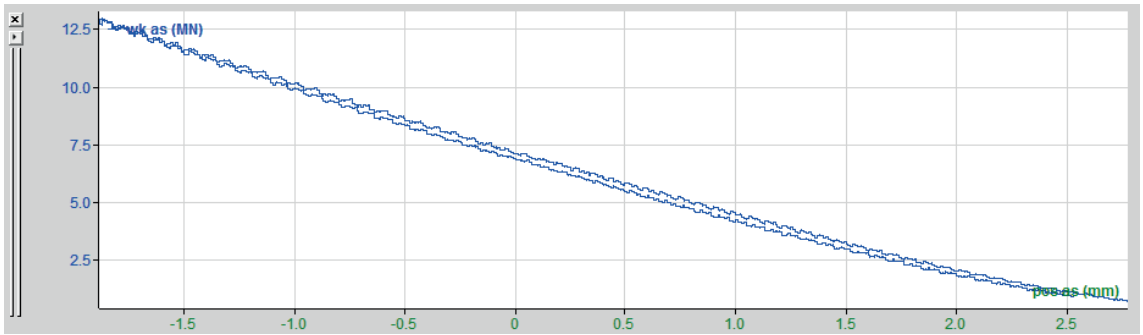
   In the example: Rolling force ("wk as") and position ("pos as")
   As both signals are time-based, the X-axis is divided into seconds.



2. Select graph mode X-Y, for example, via the button in the graph header.

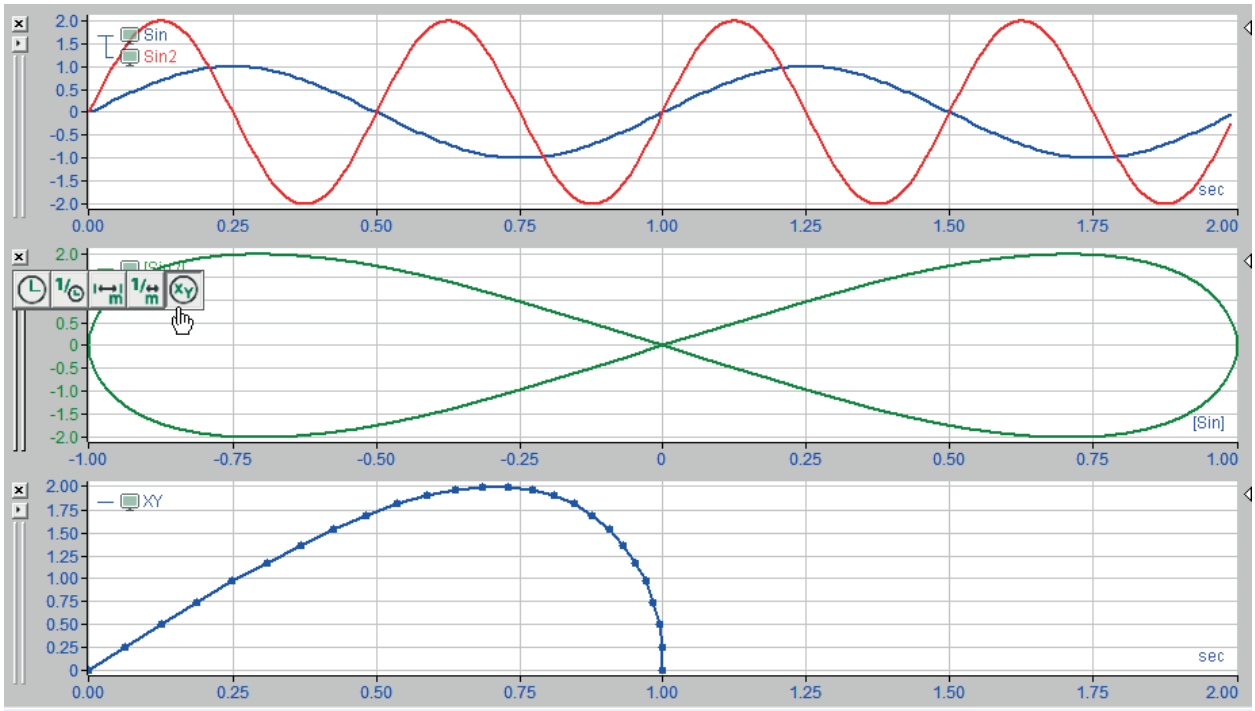


→ The X-axis is divided into the position unit (mm). The Y-axis of the other signal (rolling force) remains unchanged. Now, however, the rolling force values are no longer entered according to time, but according to the corresponding position values.



Comparison with XY function

You can also implement the X-Y plot with a calculation. The graphic shows the results of the X-axis mode and the *XY* function in comparison.



Signal definitions

	Show	SignalName	Expression	Unit	Color
2	☑	Sin	Sin(2*PI()*[T])		
3	☑	Sin2	2*Sin(2*PI()*2*[T])		
4	☑	[Sin2]	[Sin2]		
5	☑	XY	XY([Sin2],[Sin],0.1)		

For more information, see *ibaAnalyzer* manual part 3, chapter *XY*.

6.21 Views

You can change the display of the signals in the recorder window using the toolbar. Depending on how the measurement data was recorded and processed, different display types are available.



For the recorder window, you can select the normal curve view or 3D views for vectors. There are also other displays, e.g. the FFT view for data from an InSpectra module or the map view for measurement data with GPS information.

6.21.1 Standard view

The standard view is a simple 2-dimensional plot. 2D plots are used to display values that only change depending on a variable such as time or length.

Display modes

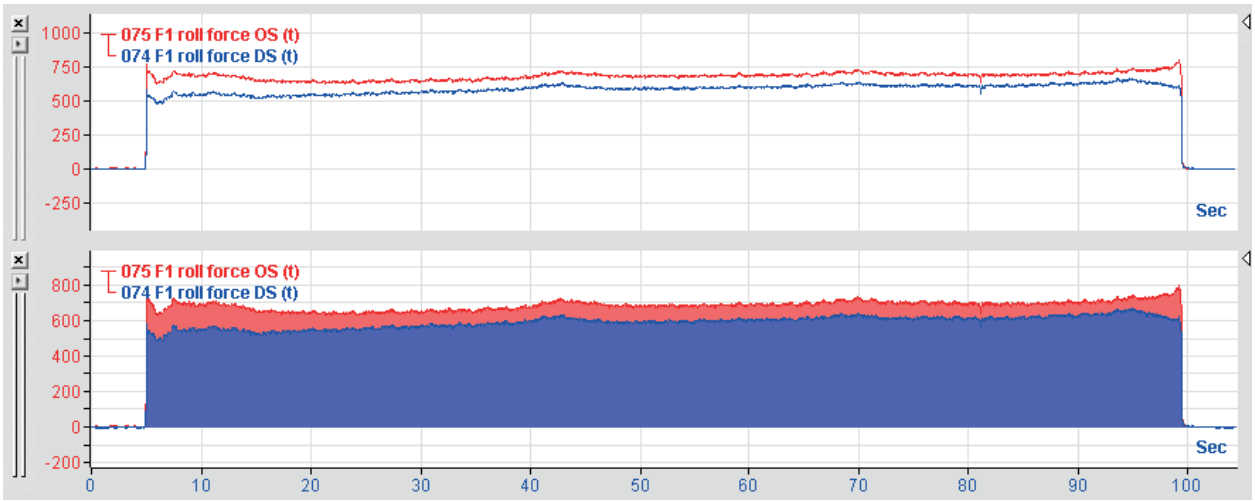
You can change the display mode in the Preferences, the Graph setup, or via the context menu in the relevant graph.

The Time and Length X-axis modes can be displayed as a line chart, polygon, points, or staircase. In FFT mode, bars and discrete frequency lines are also available.

Line chart and polygon

In line chart and polygon plots, straight lines connect the individual measuring points by interpolation along the shortest path. As a result, the measured value between two points is shown to change proportionally, although this does not always correspond to reality.

The graphic shows the line chart plot on the top and the polygon plot on the bottom.



Plotting constant values: Staircase instead of line chart

For plotting values that are constant between two points, such as batch or product numbers, a line chart is not optimal. As the interpolated connecting line suggests a continuous change in value, markers between the measuring points also show incorrect intermediate values.

In such cases, a staircase plot is better. In a staircase plot, the value of the last sample remains graphically constant until the next sample, and is shown as a horizontal line.

The graphic below shows the differences between a line chart (top) and a staircase plot (bottom).



6.21.2 3D view

A 3D view visualizes signals that are dependent on two variables.

In the steel industry, for example, temperature, flatness or thickness measurement profiles are suitable applications for 3D visualization. In addition to the time or strip length, the strip width is also included in the measurement. This additional coordinate is specified, for example, by the position of a traversing measuring device or by the various measuring zones of a flatness measuring roll.

To plot the third dimension, *ibaAnalyzer* requires a vector signal. This signal is a special input variable of the ARRAY type, where the number of field cells corresponds to the Z-axis, see ↗ *Creating vector signals (array), page 177*.

You configure the 3D views in the Graph setup menu of the graph with the 3D view or 2D top view, see ↗ *3D View*, page 75.

Mouse operation for 3D plots

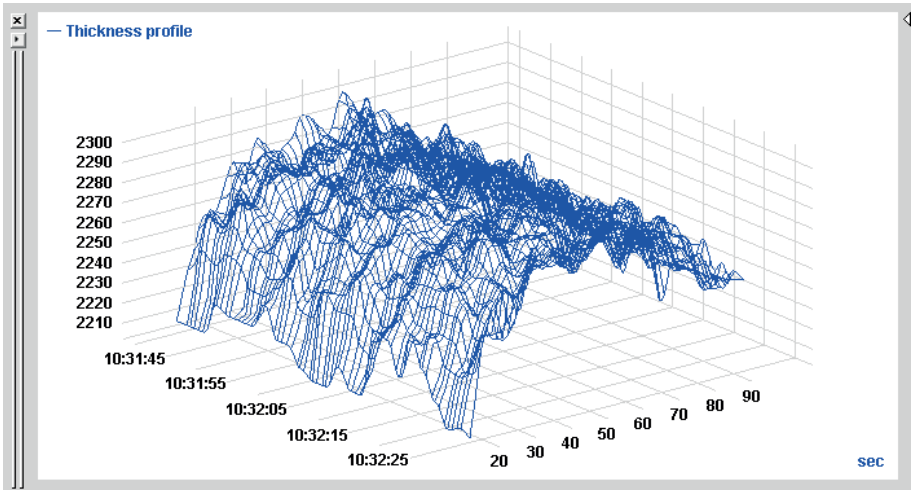
The operations to control a 2D top view are the same as those to control a 2D plot view.

The 3D wireframe and 3D surface plot types have a different mouse control to enable 3-dimensional navigation.

- When the cursor is displayed in the form of a hand, you can move the graphic within the graph by holding down the left mouse button.
- By holding down the <Ctrl> key and left mouse button, you can rotate the graphic on the axes enabled in the settings. The cursor is a rotation symbol.
- By holding down the <Shift> key and left mouse button, you can stretch and compress, or enlarge and reduce, the graphic.
- Use the mouse wheel to zoom in or out of the graphic.

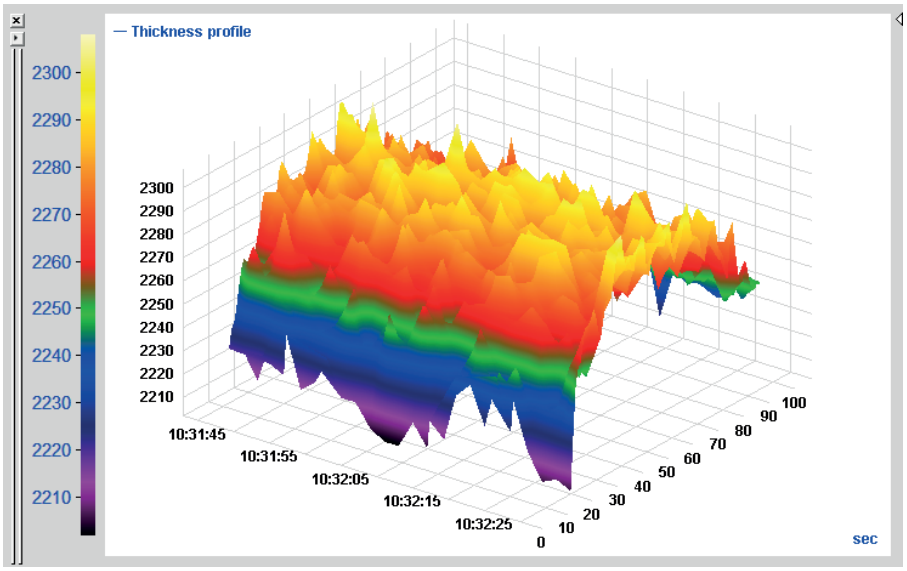
6.21.2.1 3D wireframe

This representation mode is a 3-dimensional view of the measured values as a wireframe structure.



6.21.2.2 3D surface

The 3D surface and 3D wireframe view modes are very similar. In 3D surface mode, the basic structure of the wireframe is overlaid by a solid surface.



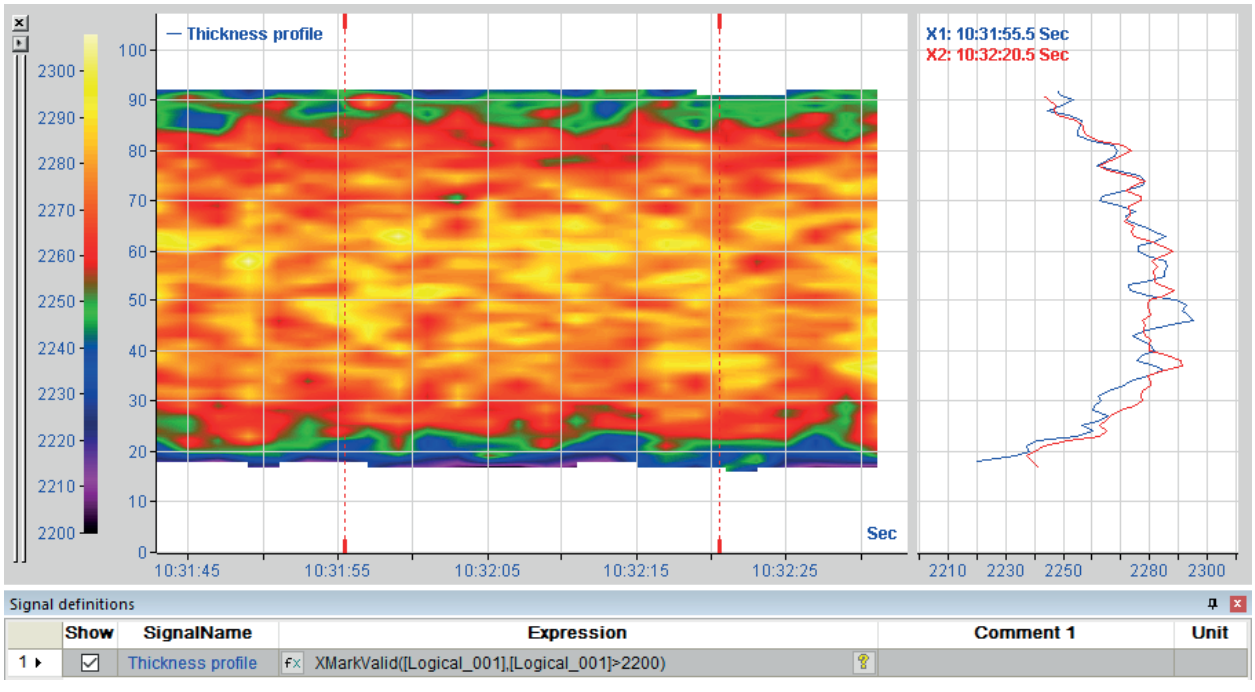
6.21.2.3 2D top view

The 2D top view is a special form of 3D representation showing a top view of a 3D view.

The amplitude of the measured values is implemented as a false color representation. Low values are dark and high values are light colored. You can change the respective colors in the Preferences or Graph setup under *3D View*, see ↗ *3D View*, page 75.

The X-axis is the time or length axis.

The image shows the thickness profile of a steel strip. In the image, the height of the colored band (in the Y-direction) corresponds to the width of the strip.



The right-hand section shows the cross profile at the marker positions X1 and X2 in the colored band. You can switch the cross profile on and off either from the *Graph Mode – Show Cross Profiles* menu or via the context menu of the graph.

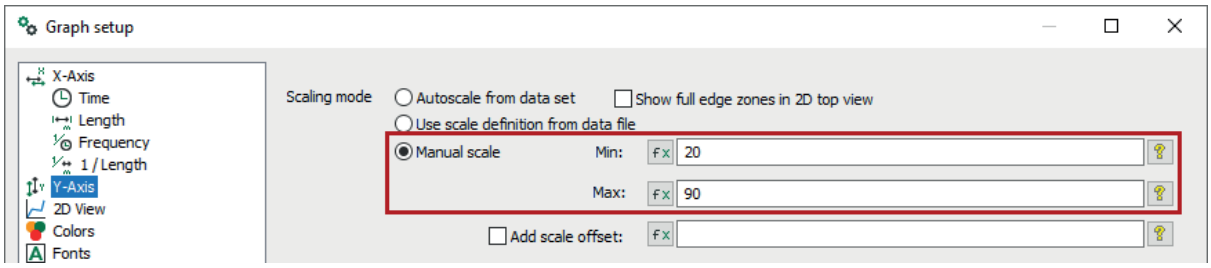
The zoom function works in the same way as for the 2D view.

You can change the settings for the 2D top view in the Graph setup under *3D View*; see ↗ *3D View*, page 75.

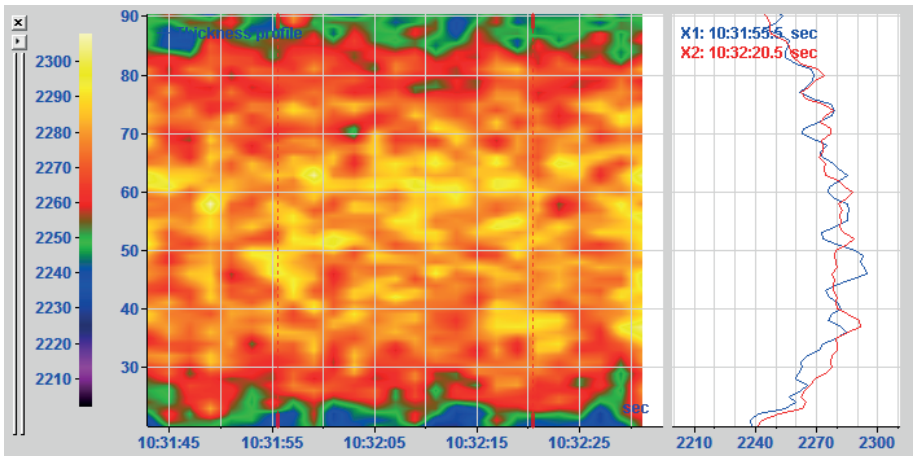
Remove invalid ranges

The margins in the 2D top view can contain invalid ranges, such as ranges with no data as in the chart above. To avoid this, you can crop the margins.

1. Open the Graph Setup for the graph with the 2D top view and go to the Y-axis settings.
2. Select the *Manual scale* option.
3. Adjust the values for *Min* and *Max* so that they correspond to the real values, and the invalid ranges in the 2D top view are excluded.



→ The Y-axis of the 2D top view is adjusted, and only the valid value range is visible.



6.21.2.3.1 Settings when using zone widths

You can assign a width and a physical unit to the individual tracks or zones. This allows you to distribute the values unevenly across the width (Y-axis) and weight them differently depending on the characteristics of the measuring instrument.

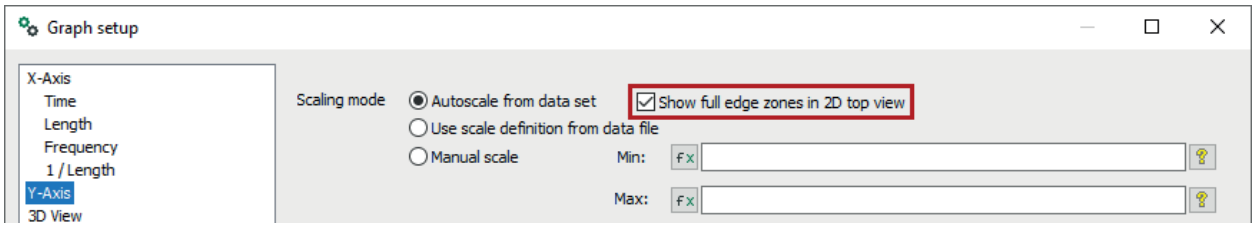
You set the zone width when defining the vector signal in the *Logical Expressions*, see ↗ *Zone control for vector signals, page 181*.

Special setting for the 2D top view

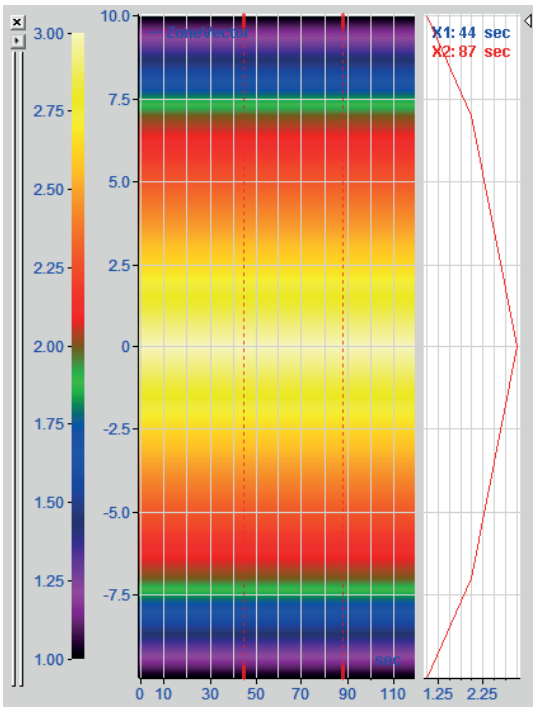
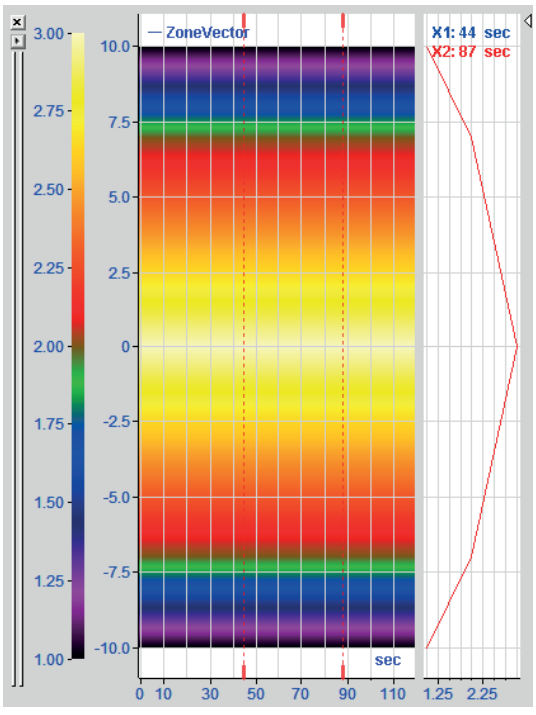
Since the measured values are in the middle of a zone, and interpolation to the value of the adjacent zones is carried out for coloring purposes, the two outermost zones each have empty areas from the middle to the outer margins of the zone. Interpolation can no longer be carried out for those marginal areas as there is no adjacent zone.

With autoscaling, the Y-axis is scaled by default to the smallest and largest valid value, so the marginal areas are not visible.

In the settings under *Y-Axis*, you can display the Y-axis completely if you have set zone widths in the vector signal and are using a 2D top view.



If you enable the *Show full edge zones in 2D top view* option, the Y-axis will be scaled to the full width of all zones, as can be seen in the graphic below on the right.

Zone marginal area cut off at -10 or +10 (option disabled, default)	Zone marginal area displayed (option enabled)
	

6.22 FFT view (ibaAnalyzer-InSpectra)

The view is mainly used to carry out frequency band analyses with data from an *ibaInSpectra* module (*ibaPDA*). The structure and operation of the FFT view are similar to the view in *ibaPDA*.

You can create calculation profiles for the frequency band analysis in *ibaPDA* and use them with *ibaAnalyzer* as well. Conversely, you can also determine calculation profiles offline with *ibaAnalyzer* and then import them in *ibaPDA* in order to execute the process-synchronous analysis in the InSpectra module.

Other documentation



For detailed information about the InSpectra FFT view, see the manual for the product *ibaAnalyzer-InSpectra*.

6.23 Orbit view (ibaAnalyzer-InSpectra)

The view is mainly used to analyze on the shaft position for plain bearings with data from an *ibaInSpectra* module (*ibaPDA*). The structure and operation of the Orbit view are similar to the view in *ibaPDA*.

You can create calculation profiles for the Orbit analysis in *ibaPDA* and use them with *ibaAnalyzer* as well. Conversely, you can also determine calculation profiles offline with *ibaAnalyzer* and then import them in *ibaPDA* in order to execute the process-synchronous analysis in the InSpectra module.

Other documentation



For detailed information about the Orbit view, see the manual for the product *ibaAnalyzer-InSpectra*.

6.24 Computation module

The *Computation Module* view in *ibaAnalyzer* is the counterpart to the computation module in the *Analytics* area of *ibaPDA*. It is based on the use of profiles by which the calculations are defined.

Similarly to the integration of the *ibaInSpectra* and *ibaInCycle* modules, you can also load, change and save profiles for the computation module. Within the view, it is possible to change the profile settings and check the result of the calculation. You can import profiles created in and exported from *ibaPDA* into *ibaAnalyzer* and edit them there.

Conversely, you can transfer newly created or modified profiles in *ibaAnalyzer* to *ibaPDA* to apply them there in a computation module. This is the key use case, as it is much easier to formulate and test a calculation offline than in a running *ibaPDA* system. With a data file as a source containing all signals, you can develop, test and optimize the desired calculation at your leisure, and then save it as a profile.

The computation module in *ibaAnalyzer* takes the input signals for the calculation from the data file open in *ibaAnalyzer*, and so is able to calculate with realistic data.

Ultimately, the calculation results are made available in the *ibaAnalyzer* signal tree for display or further processing.

You can create multiple computation modules that use the same profile.

Computation modules are saved in the analysis (PDO file).

The computation module also offers two special features.

Parallel processing of multiple calculations
In a computation module, you can perform the calculation defined in the profile for multiple input signals in parallel. For example, a data file may contain several similar signals or groups of signals, such as voltage, current, temperature, and the rotational speed of three motors. All motors should use the same calculation. In this case, configure the appropriate number of inputs (in this example, 3) in the calculation module. You will then get the calculation results for each input as signals in the signal tree.

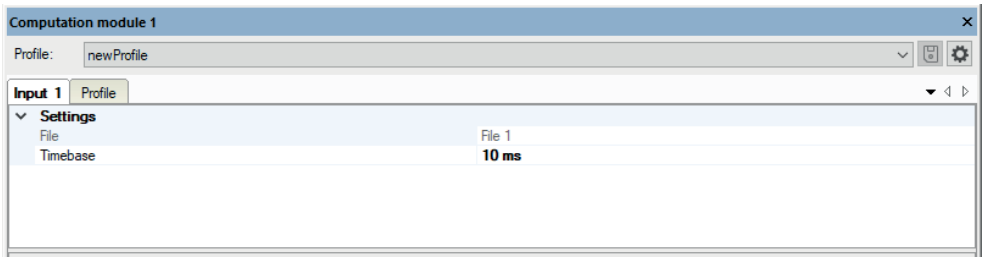
Calculation with input signals from various data files
If you have opened multiple data files of the same type in *ibaAnalyzer*, you can perform the calculation defined in the profile with signals from different data files. In this case, configure an input for each data file, and assign a different data file and the same signals to each input.

6.24.1 Creating computation module

You can open a computation module from the toolbar or via the *View* menu.



In the *Computation Module* window, you carry out the configuration.



Areas of the computation module

Name
ibaAnalyzer automatically assigns default names "Computation module 1...n", which are displayed in the window header. They are numbered in the order in which they were created. You can change the name of a computation module subsequently. To change the name, open the context menu with a right-click on the window header and select *Rename*.

Profile
The selected profile for the calculations in this computation module is displayed on this line. For a new computation module, "New profile" is displayed here. If profiles already exist, you can select the appropriate one from the drop-down list. If there are no profiles yet, you can create a new profile yourself.

For more information on creating a profile see ↗ *Creating and configuring a profile, page 144*.

Input tab
In this tab, you select the data file from which signals are to be used for the calculation. You can set the time base for the calculation. Here you also assign the signals to the placeholders in the calculation depending on the profile.

For more information see ↗ *Linking inputs to the computation module, page 147*.

Profile tab
In this tab, you configure the profile, i.e. the actual calculation. Here you define the placeholders for the calculation, and enter the calculation formulas for analog signals and digital signals.

For more information see ↗ *Creating and configuring a profile, page 144*.

<Calculate> button
Use the <Calculate> button to initiate a new calculation. The button lights up yellow when the profile or another parameter of the computation module has been changed. To obtain updated results, you must trigger a recalculation manually.

6.24.2 Creating and configuring a profile

To create a profile, click on the button with the gearwheel icon.



The *Manage profiles* dialog opens up.

On the side, there are buttons with the following functions:

	Add profile
	Copy selected profile
	Delete selected profile
	Import profiles from a **.computationProfile* file
	Export selected profile to a **.computationProfile* file

Add a profile, and enter a meaningful name so that the function of the calculation is identifiable.

If you already know that the profile is going to be saved as a global profile, you can enable the option here. You can also do that later, however. The advantage of a global profile is that other software, such as *ibaDatCoordinator* or *ibaPDA*, can also access it.

Then click on <OK>.

→ The new profile is now entered at the top of the computation module.

Then make the settings for the calculation in the *Profile* tab. Start with the placeholders.

Configuring a profile – Placeholders

To formulate your calculation, you must first define the placeholders for the input variables of the calculation.

Enter the placeholders one after the other in the *Placeholders* tab. As you fill in each line, the next vacant line is generated automatically.

Name
Name of the placeholder as it will subsequently be used in the calculation function.

Default constant value
Enter a value here that the placeholder is to assume if no signal is linked, or the signal is invalid. Note that this value must match the value type.

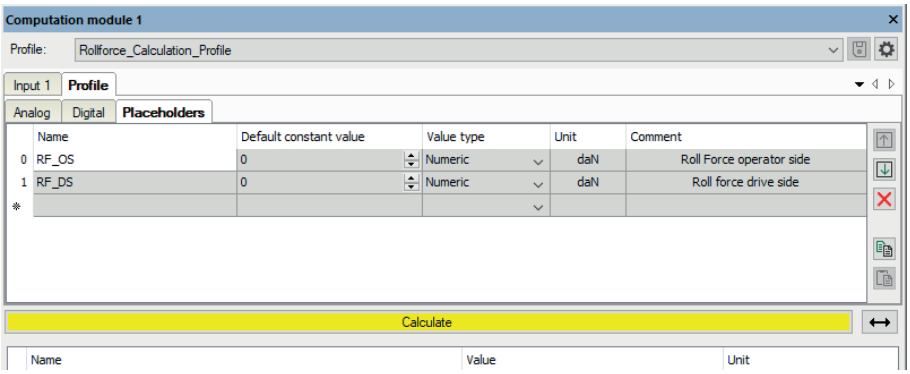
Value type
Select the value type from the drop-down list here. It is the type of the signal, or constant. The available options are:

- Numeric: a numerical value
- Digital: a purely digital signal (True/False, 0/1)
- Text: a text signal
- Any (default numeric): Value type is detected automatically
- Any (default digital): Value type is detected automatically
- Any (default text): Value type is detected automatically

Comment
You can enter a brief description of the placeholder here.

Example: Placeholder in profile
To calculate the total and differential rolling force on a rolling stand, enter the following placeholders:

- RF_OS (for operating side rolling force)
- RF_DS (for drive side rolling force)



In the next step, formulate the expressions in the *Analog* and *Digital* tabs.

Configuring a profile – Analog and digital expressions

Now formulate the desired calculations in the *Analog* and *Digital* tabs.

When you click on the <fx> button in the *Expression* column, all the functions in the expression builder are available to you.

In the formulas, you can use placeholders as well as the results of other expressions in the same profile or constant values. If you want to use the result of a line in a different expression, use the predefined placeholders {analog:X} or {digital:Y}, where X and Y stand for the name of the expression in the *Analog* or *Digital* tab of the same profile.

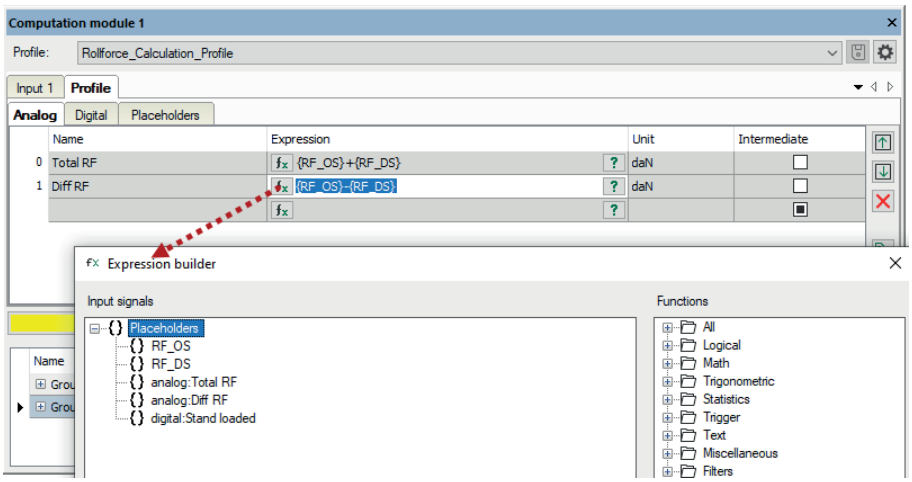
If you fill in multiple lines, you can manipulate the lines using the buttons on the right. The order of the lines is not relevant for calculation of the expressions.

	Move selected rows up
	Move selected rows down
	Delete selected rows
	Copy all rows
	Paste from clipboard, starting from the selected row

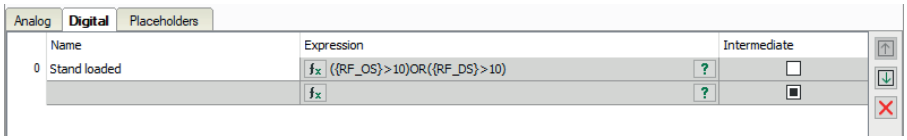
In the *Intermediate value* column, you can classify any expression as an internal intermediate value. These expressions or signals are then not available later in the signal tree.

Then save the profile.

Example: Calculation of the analog signals and digital signals
The total and differential rolling forces are calculated as analog values. Both the defined placeholders and the calculated values are shown in the Expression builder.



A digital signal is designed to indicate when one of the two forces is greater than a minimum value.



Switch to the *Input* tab and select the desired input signals.

6.24.3 Linking inputs to the computation module

In the final step of configuring the computation module, you link the real signals with the input parameters.

Select the *Input 1* tab.

Settings

File
The name of the data file currently open in *ibaAnalyzer* is shown on this line. If you have multiple data files open at the same time, they are shown here in a drop-down list. The calculation only works if the signals from the selected data file are also linked to the placeholders below under *Placeholders*.

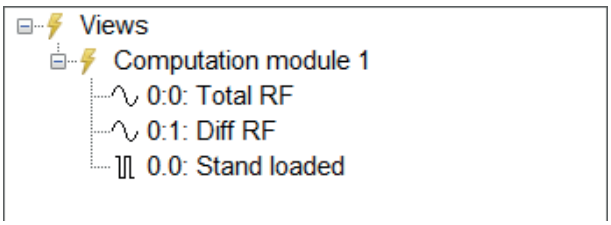
Timebase
Use this time base to determine the sampling for the calculation or the time resolution of the calculation results. The default is 10 ms. You can change the value of the time base to adapt it to the time base of the data file.

If the measured values in the data file were recorded with a time base of 100 ms, for example, with a time base of 10 ms in the computation module you would get tenfold oversampling, which would not be necessary. However, if the measured values in the data file were recorded at a higher sampling rate, such as 1 ms, then it may also be necessary to reduce the time base of the computation module to 1 ms in order to obtain accurate results.

Placeholders

Here you will find the placeholders previously defined in the profile. Click in the fields in the right-hand column to open the signal tree of the data file and assign the corresponding signal in each case.

When you perform the calculation for the first time after this assignment, the result signals also appear in the signal tree.

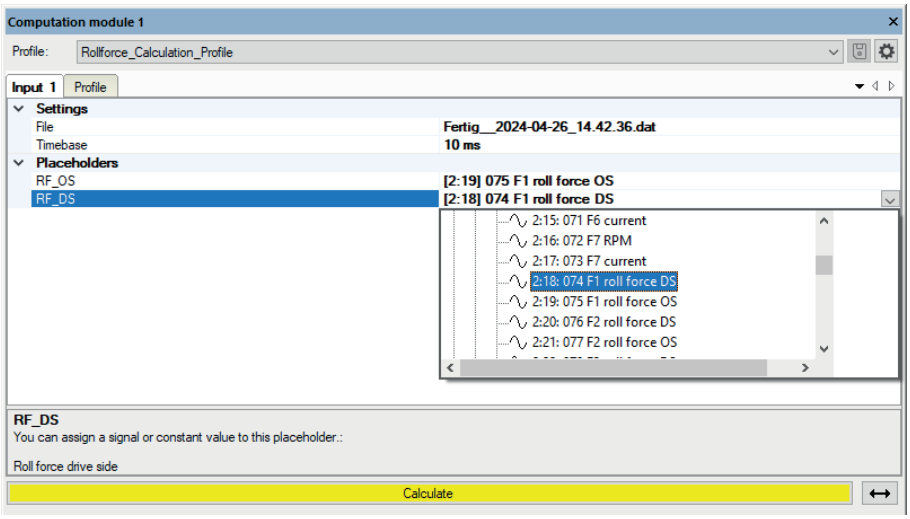


The result signals are numbered according to the following scheme:

Analog and text signals: `Input index:Signal index`

Digital signals: `Input index.Signal index`

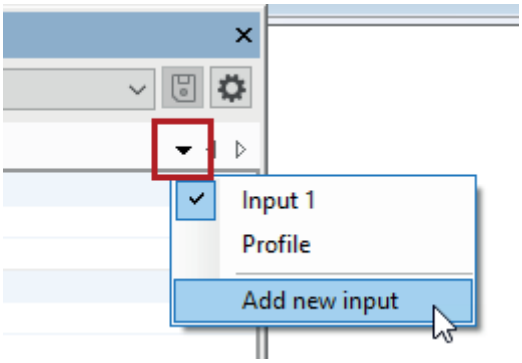
Example: Assignment of signals
The two rolling force signals for the operating and drive sides are assigned to the placeholders.



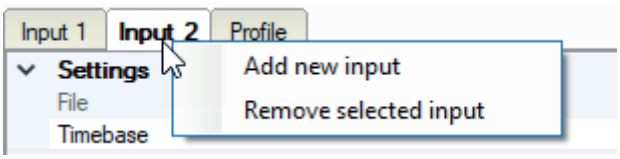
Parallel processing of multiple inputs

One advantage of the computation module is the option of running the calculation defined in the profile in parallel for multiple inputs.

To add another input, click on the small black arrow on the right in the tab row and select *Add input*.



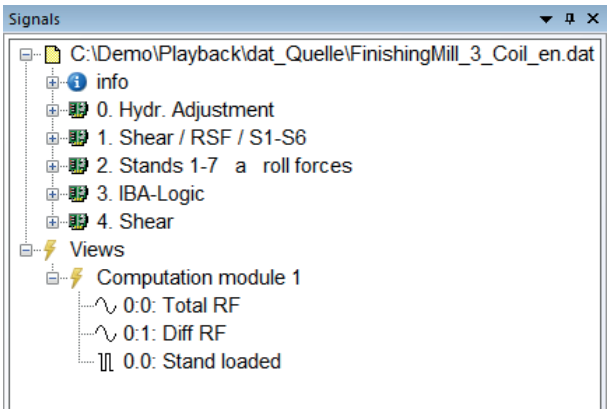
Alternatively, you can open the context menu via right-click on an input tab.



You can also remove inputs again via the menus.

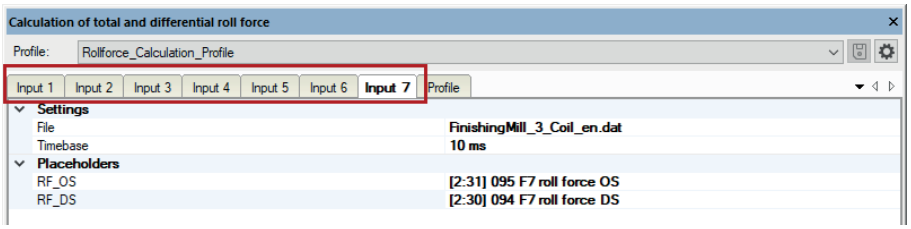
In the new input, you can then link the desired input signals from the data file again under *Placeholders*.

When you perform the calculation for the first time after this assignment, the result signals also appear in the signal tree.

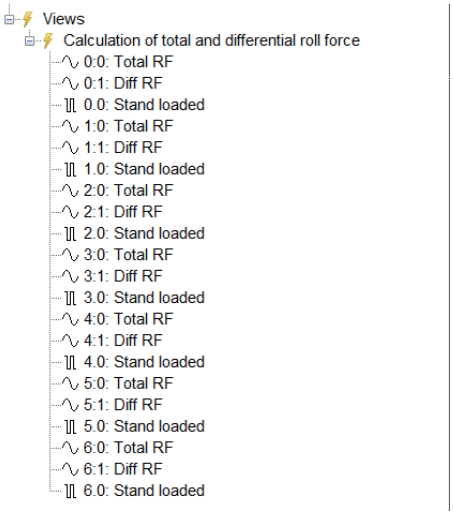


Example: Calculation for all rolling stands
In the example of the rolling mill, the calculation module can now calculate the rolling force for all 7 stands.

One input per stand



Signal tree



6.24.4 Placing and using a computation module

The computation module is a freely movable and dockable window. Use the mouse to move the window to the desired position. If you want to place the window in a fixed position without it covering other areas, drag it onto one of the docking fields shown.

Performing calculations

If you change the time base of the computation module, or make changes to the profile, the <Calculate> button lights up yellow. Then click the button to obtain updated calculation results.

Calculations are performed each time the <Calculate> button is clicked, not just when it is yellow.

To the left of the <Calculate> button is another button with a double arrow.

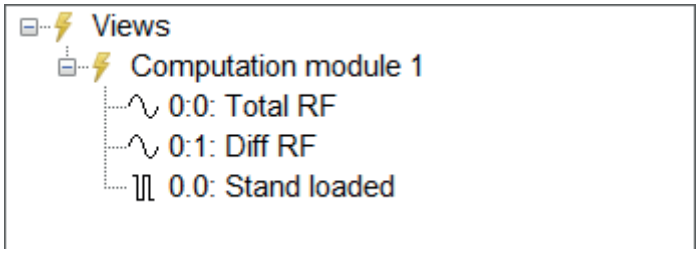
Use this button to turn the limitation to the current zoom range on or off.

	Zoom range irrelevant. All samples in the data file are used for the calculation.
	Calculation limited to zoom range (X-direction). Only the samples within the current zoom range are included in the calculation.

You can check this by zooming out again after a calculation in the zoom range. If you also have the results in the graph, the result values only appear in the previously zoomed area.

Showing results

The results are provided as signals in the *ibaAnalyzer* signal tree.



From there, you can drag and drop the result signals into the display area.

6.25 Audio player

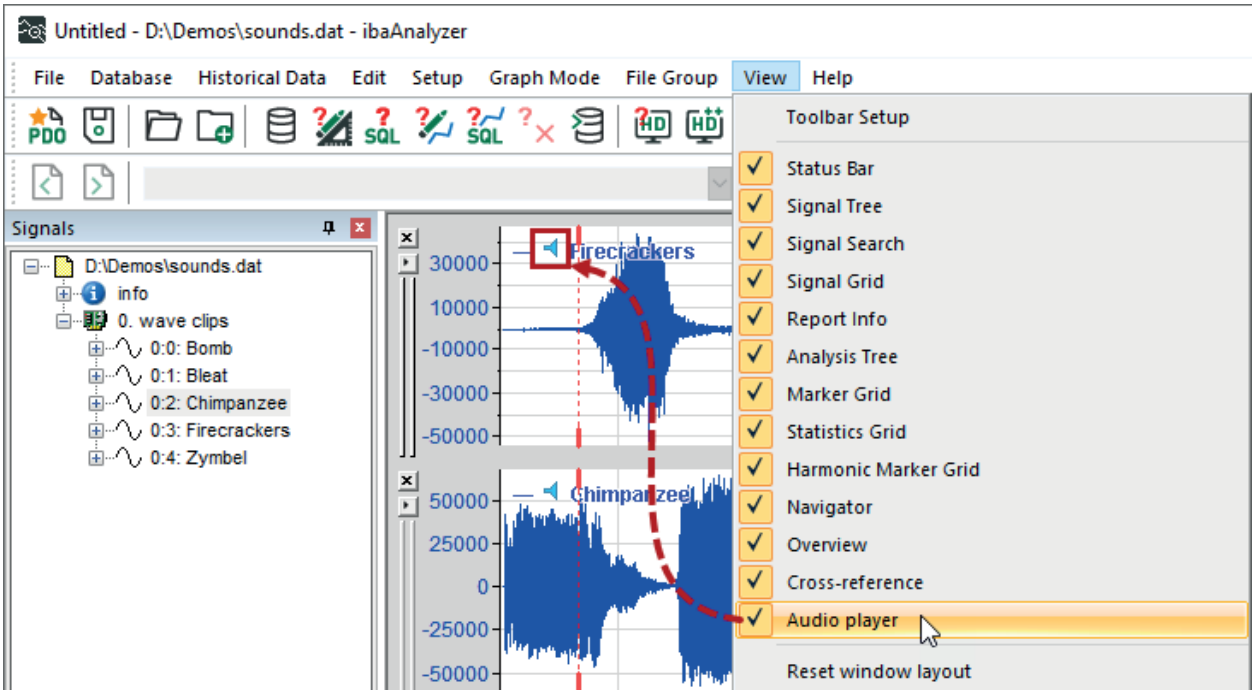
The audio player in *ibaAnalyzer* interprets time-based signals as sound signals. You can play the sound via the system's standard speakers.

The playback function is enabled for signals that were recorded with at least 100 samples/s, i.e. with a sampling rate of at least 100 Hz.

The acoustic playback can be helpful when analyzing vibration phenomena.

6.25.1 Turning on the audio player

By default, the audio player of *ibaAnalyzer* is turned off. You can turn it on in the *View* menu – *Audio player*.



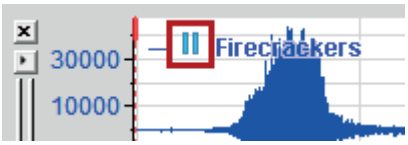
If the audio player is turned on, the signal legends shows a speaker icon for the signals suitable for audio output.

You can use the same menu to turn off the audio player again.

6.25.2 Audio player playback

To play back a signal via the audio player, click on the speaker icon in the signal legend. The playback starts from the position of marker X1. The marker moves along with the playback until the end of the data file.

During playback, the volume icon changes into a pause icon so that you can stop and continue the playback at any time by clicking on the pause icon.



The audio playback pauses in the following cases:

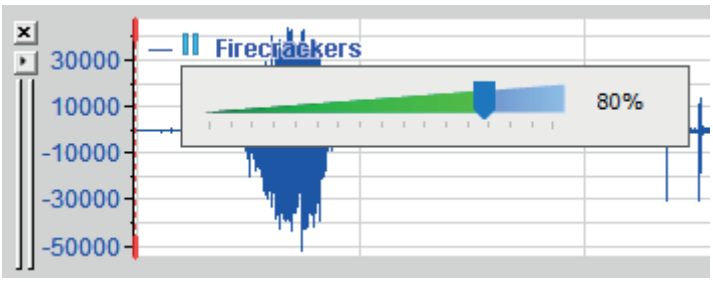
- You click on the pause icon in the signal legend.
- You move the markers.
- You change the signals present in the display window or an expression. You add or remove a graph, etc.
- A data file or other data source (trend or HD query) is loaded/reloaded.
- You play a video.
- You turn off the audio player in the menu *View*.
- The playback has reached the end of the data file.
- No more data is present.

Tip

If you want to listen to more than one signal at the same time, then use an expression (additional signal in the signal table or logical expressions) in which you add the respective signals. The audio tracks are then superimposed.

6.25.3 Audio player volume

A slide control appears below the signal legend during playback, which you can use to set the volume.



You can also change the volume via the Windows sound settings.

You cannot change the volume by scaling the signal with a factor. The signal data is normalized before it gets to the audio output.

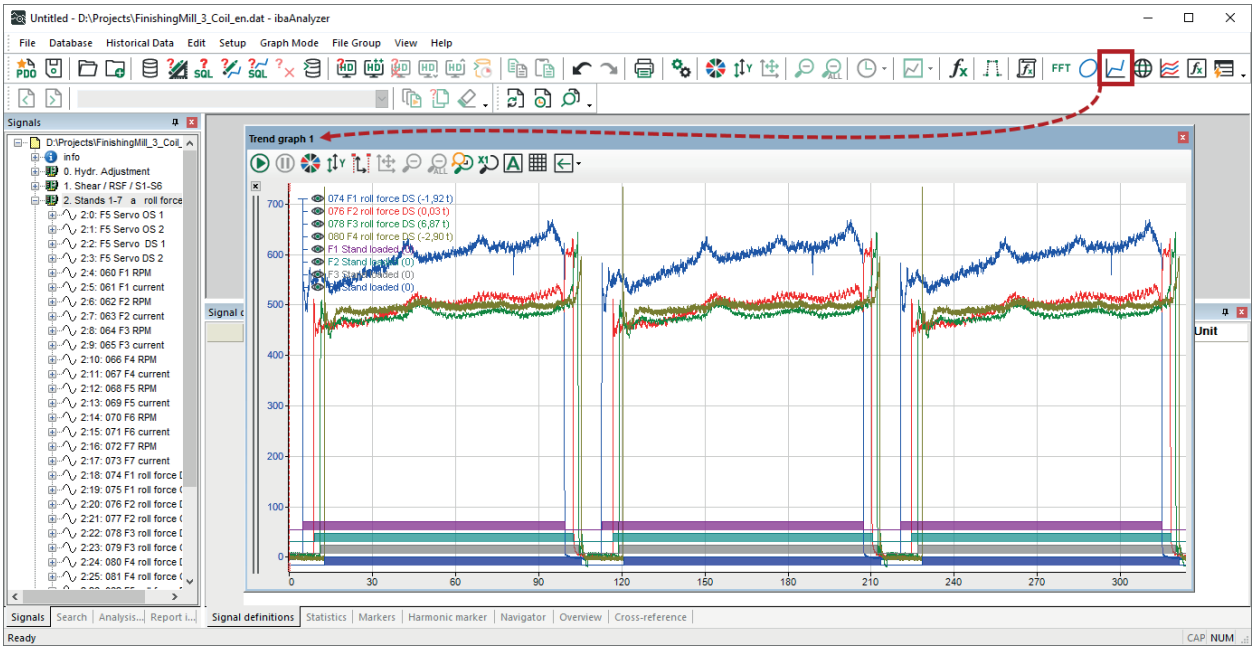
6.25.4 Audio player synchronization with video

If a data file contains *ibaCapture* videos, then the videos are played back together with the audio player when the audio player is started. The replay speed of the video is set to the original speed (1x) in this case.

Conversely, video playback does not start audio playback.

6.26 PDA trend view

A more flexible kind of signal presentation is the PDA trend view, which you can open from the toolbar or via the *View* menu. Every click on this tool button opens a new trend view.



In the style of the trend view in the data acquisition software *ibaPDA*, the view has the following properties:

- Display of all time-based signals and expressions, incl. vectors, text signals, analog and digital signals
- Each PDA trend view with its own X-axis and marker grid
- Relative or absolute time for the X-axis
- Free floating and dockable windows, which can also be arranged as piled tabs or set to auto-hide mode
- Individual zooming or adapting to the zoom area in the recorder window or navigator for each view
- Nearly the same settings as for the trend view in *ibaPDA*

6.26.1 Functions of the PDA trend view

Basically, the PDA trend view offers the same functions and operations as in *ibaPDA*.

Other documentation

You can find a detailed description of operation and setup of the *ibaPDA* trend graph in the *ibaPDA* manual, part 6.

The following buttons are in the toolbar for special *ibaAnalyzer* functions:



	Apply the same zoom area as set in the recorder window/navigator pane to the PDA trend view.
	Zoom in on the X1 marker from the *ibaAnalyzer* recorder window You can define the zoom on the X-axis in the settings, see ↗ *Properties of the PDA trend view, page 155*.
	Show or hide marker grid (toggle).

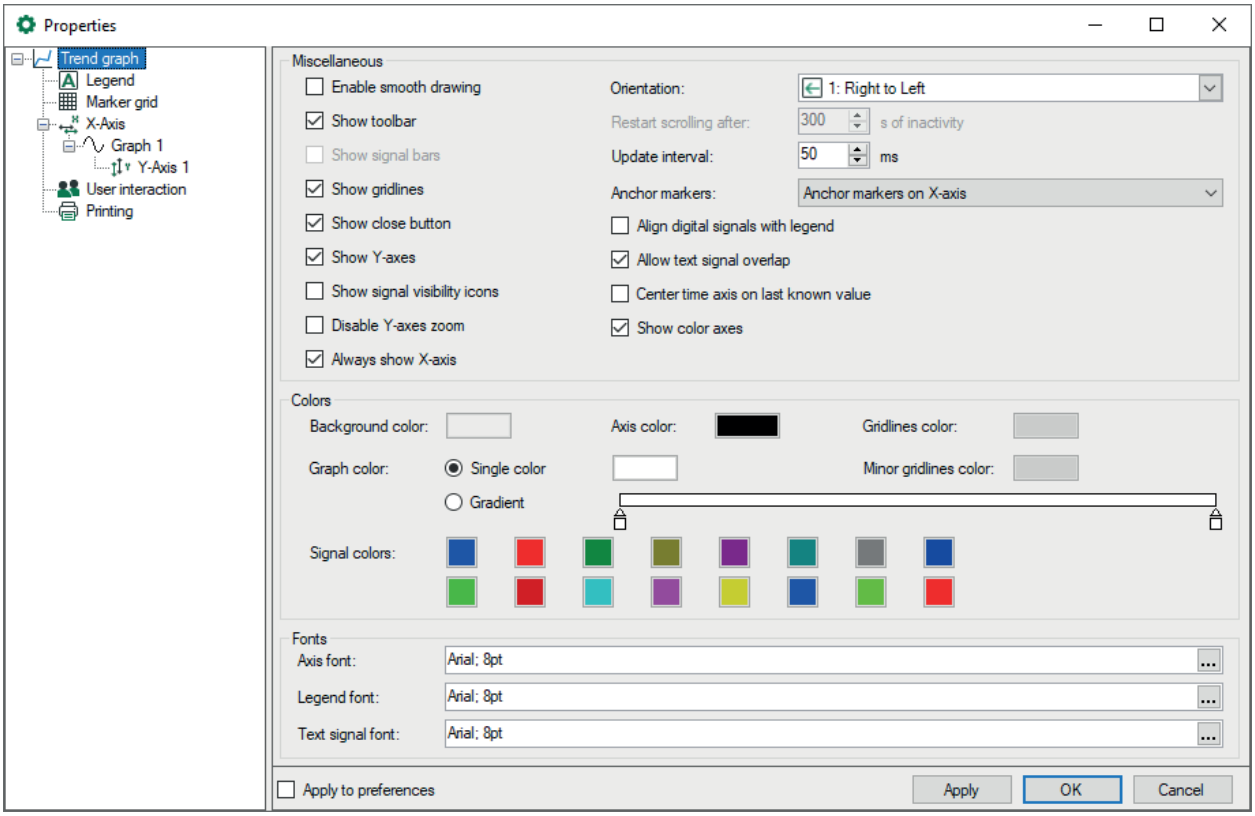
Note

You can add signals from the signal tree to the PDA trend view only via drag & drop. Combined shortcuts like double-click + <Ctrl> or <Shift> do not work.

6.26.2 Properties of the PDA trend view

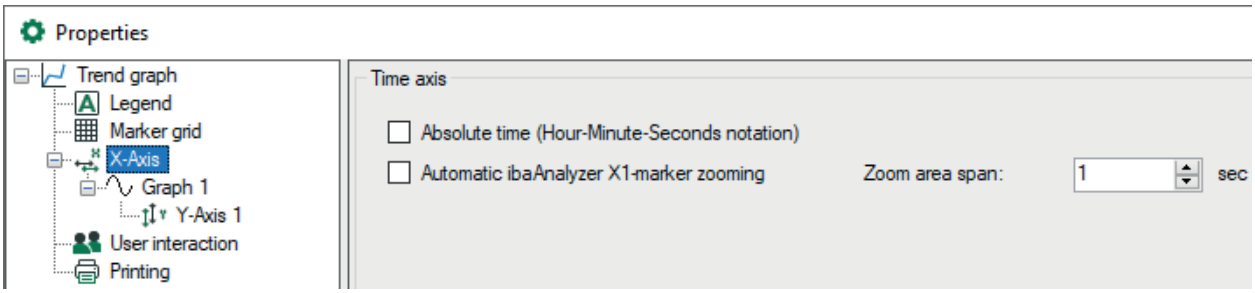
The dialog for the properties and settings of the PDA trend view is similar to the properties in *ibaPDA* however without the options for live view.

You can open the properties dialog using the context menu in the graph area of the PDA trend view and the command *Properties*.



Settings for the X-axis

As an additional settings, you can choose between absolute or relative time specification on the X-axis. Also, you can set a symmetrical X-axis zoom in seconds for the X1 marker zooming.



6.27 Map view

The map view can display geographical positions and movements based on GPS data. You can use the recorded geographical longitude and latitude data to evaluate the positions or routes of goods or plant units, and correlate them with high-resolution measurement data from industrial processes.

The map preview offers the following functions:

- Display configurable routes in different map types
- Analyze geographical positions together with high-resolution measurement data
- Synchronized bidirectional linking of marker positions
- Integrated playback function
- Free-floating and dockable windows, which can also be arranged as piled tabs or set to "auto-hide" mode
- Usage of maps in *ibaAnalyzer-Reportgenerator*

Requirements

The following requirements must be met to display maps and positions:

- Active Internet connection

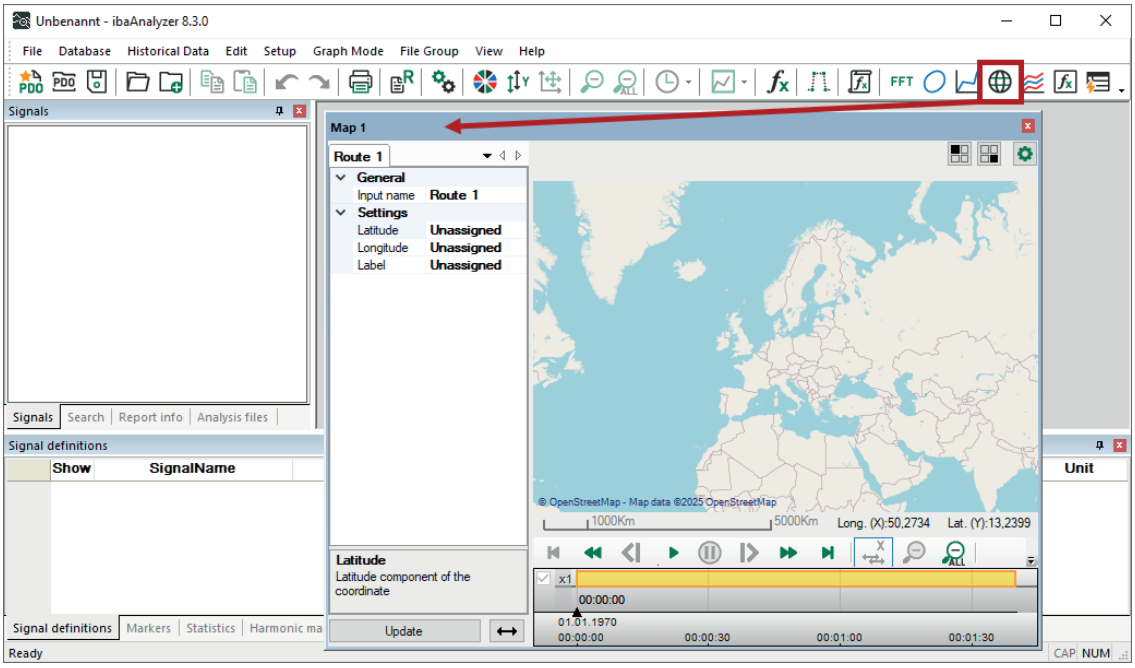
  The map view requires an active Internet connection in order to access map services.

- Geographical data

  The data file or HD recording must contain two signals with the values for longitude and latitude.
  The values must be in degrees as floating-point values, i.e. in decimal degrees. Additional minutes and seconds as separate signals are not supported. If necessary, you can use the expression builder to generate suitable signals.

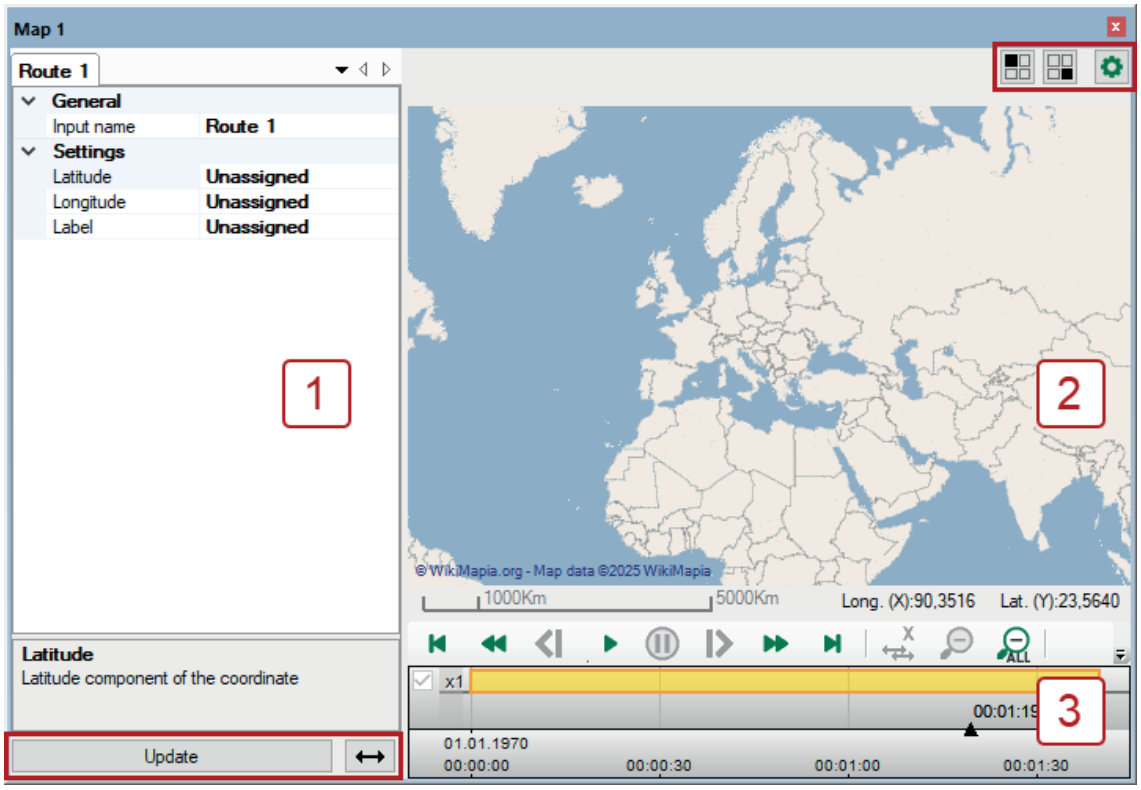
Open map view

You can open the map view from the toolbar or via the *View* menu. Each click on the button opens a new map view.



User interface of the map view

The map view features various areas that you can show and hide.



1	Configuration area, see ↗ *Configuring routes, page 158*
2	Map area, see ↗ *Map area, page 160*
3	Playback area, see ↗ *Playback area of the map view, page 161*

The following control elements are available in the view:

	Show/hide configuration area
	Show/hide playback area
	Open map view properties
	Apply zoom range of trend graph to map view in recorder window
<Update>	Apply changes in the configuration area to the map view

If the configuration area is hidden, the buttons <Update> and <↔> (Use *ibaAnalyzer* zoom range) are above the map.

6.27.1 Configuring routes

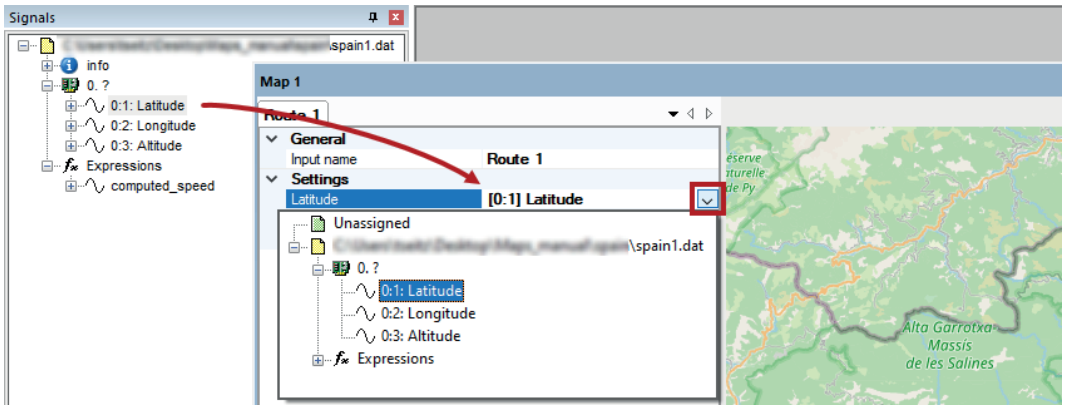
You set the input data for the routes in the configuration area. For each route, you need at least one latitude signal and one longitude signal from the open data file.

1. Enter a name for the route in the *Input name* field.

→ The name is adopted for the tab.

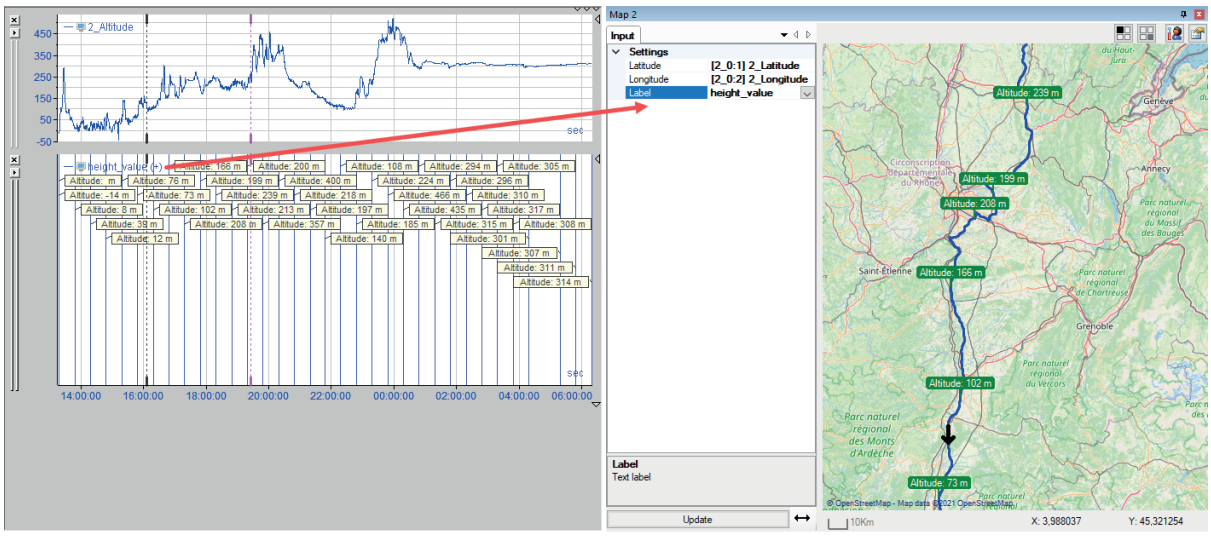
2. Select a latitude signal and a longitude signal.

   You can use the relevant drop-down list for this, or add the signal from the signal tree using drag & drop.



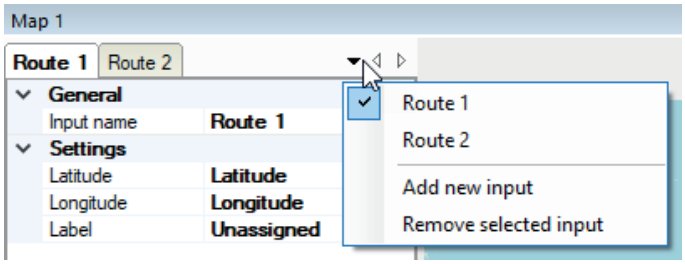
3. Optionally: To insert a label, select a text signal under *Label*.

→ Each entry of a text signal in the signal trend creates a label along the route.



4. Optionally: To add further routes, open the context menu of a tab or click on the small black triangle in the header.

   Choose *Add new input*.



5. Click the <Update> button to apply the changes for the map view.

   If there are changes that have not been applied, the <Update> button is highlighted in yellow.

6.27.2 Map area

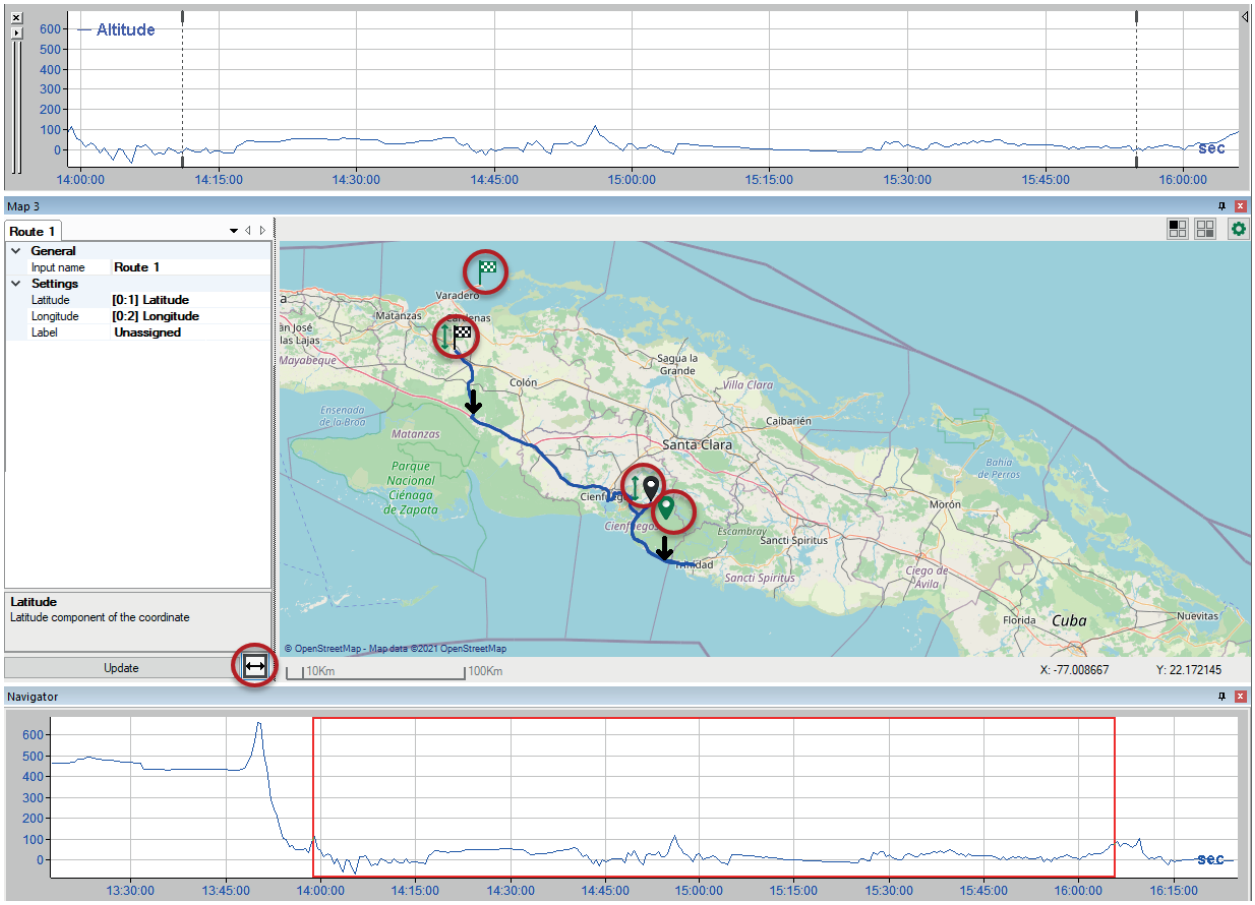
This area shows the configured routes on a map. You can select the map type in the properties of the map view; see ↗ *Map view properties, page 163*.

Operation

You can use the map like other maps on the web. Move the map view area by holding down the mouse button, and zoom with the mouse wheel.

The level of detailing when zoomed depends on the map type.

Click the <↔> button (Use *ibaAnalyzer* zoom range) to limit the map view to the section of the route corresponding to the data in the signal trend in the recorder window.
The beginning and end of this section are marked in addition to the route start and end icons.



Route icons

	Starting point of the complete route
	Starting point of the route section corresponding to the zoomed trend graph area
	End point of the route section corresponding to the zoomed trend graph area
	End point of the overall route

Markers X1 and X2

The markers X1 and X2 from the signal trend are linked by two markers shown on the route. During playback, the X1 marker also shows the current position on the route in the signal trend.

You can also move the markers manually to adjust the position and signal trend.

The color settings from Graph Setup (under *Colors*) for the markers also apply to the markers in the map view.

Context menu in the map area

You can use the context menu in the map area to determine the display and position of the route legend, and to access the map properties dialog; see ↗ *Map view properties, page 163*.

Other commands

Show full route
Reset the map scaling so that the entire route is visible on the map (Auto-zoom).

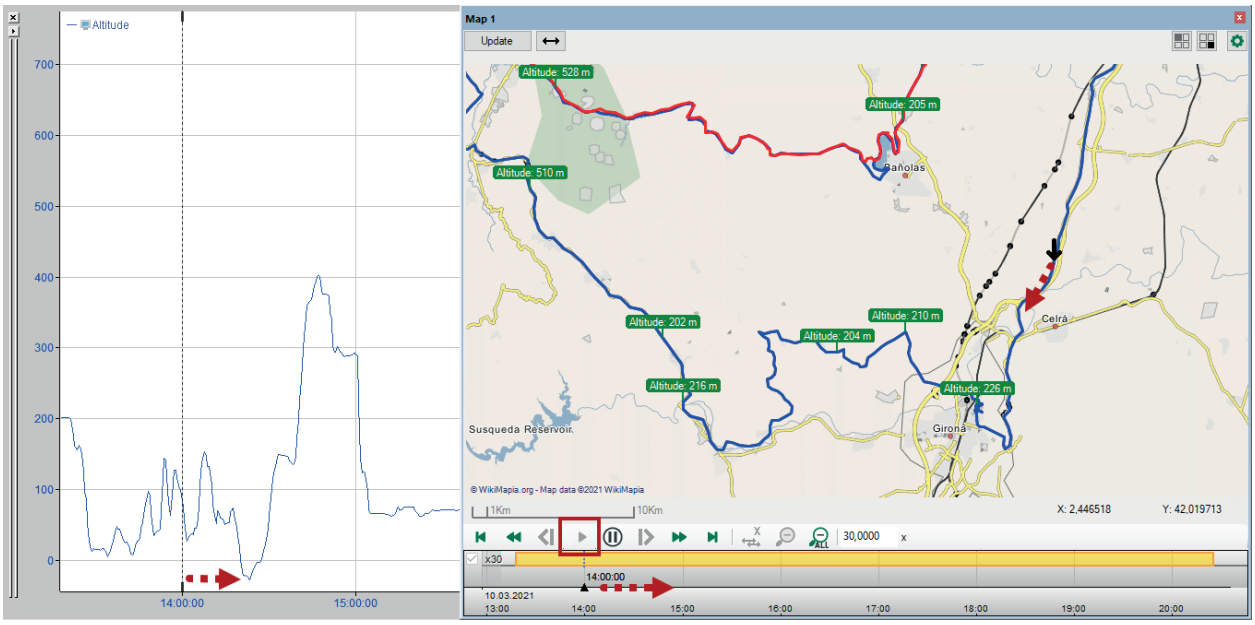
Center map on X1 marker
This command moves the map section along the route according to the X1 position during playback, or when the X1 marker is moved manually within the signal trend.

When you move the map or the marker on the map manually, this centering mode is canceled.

6.27.3 Playback area of the map view

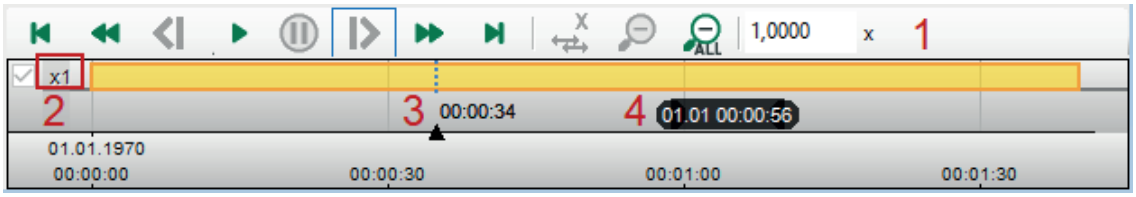
You can use the playback function to play the position on the map and the associated measured values synchronously.

The markers X1 and X2 from the signal trend are linked by two markers shown on the route. During playback, the X1 marker also shows the current position on the route in the signal trend. You can also move the markers manually to adjust the position and signal trend.



Control elements

In the playback area, you can control playback of the data file using the buttons and the slider.



1 Enter playback speed
You can enter the factor of the playback speed here. Press <Enter> to apply the new speed.
The playback speed is specified relative to the normal speed. 2.00x means, for example, that the current playback speed corresponds to twice the normal speed.

2 Display playback speed

3 Time marker
A black triangle on the time line represents the current time stamp. If you move the time marker, the X1 marker follows accordingly in the trend graph and the associated marker on the map moves to the corresponding position along the route. You can move the time marker by clicking on it and dragging it with the mouse. When you click anywhere on the time line, the marker will jump to that position.

4 Tooltip
When you move the mouse over the time line, the time stamp of the mouse position is shown in the tooltip.

Meaning of the buttons:


Start/stop playback


Skip to start/end


Reduce/increase playback speed
(the set playback speed is displayed on the left (2))


No function in the map view (not available)


Show entire time range


Zoom one level/all levels back in the playback area (does not apply to the map area)

You can also control playback from the keyboard:

Key	Function
<Up>	Increase playback speed
<Down>	Reduce playback speed
<Space bar>	Play/Pause

Zoom and shift the time scale
You can zoom in the time scale by drawing a rectangle on the time line while holding down the mouse button.

You can shift the visible time range by clicking (holding) on the time scale and moving the mouse horizontally. The cursor then appears as a double arrow.

Zooming the time line has no effect on the zoomed map view.

6.27.4 Map view properties

The *Properties* dialog offers various options for displaying the route.

You can open the dialog via the context menu of the map area or by the gearwheel button at the top right.

Confirm your changes with <OK>.

Key properties

You can set the desired map type on the *Map view* node. Various open-source maps can be selected.



Map type
You can use the following card types as standard:

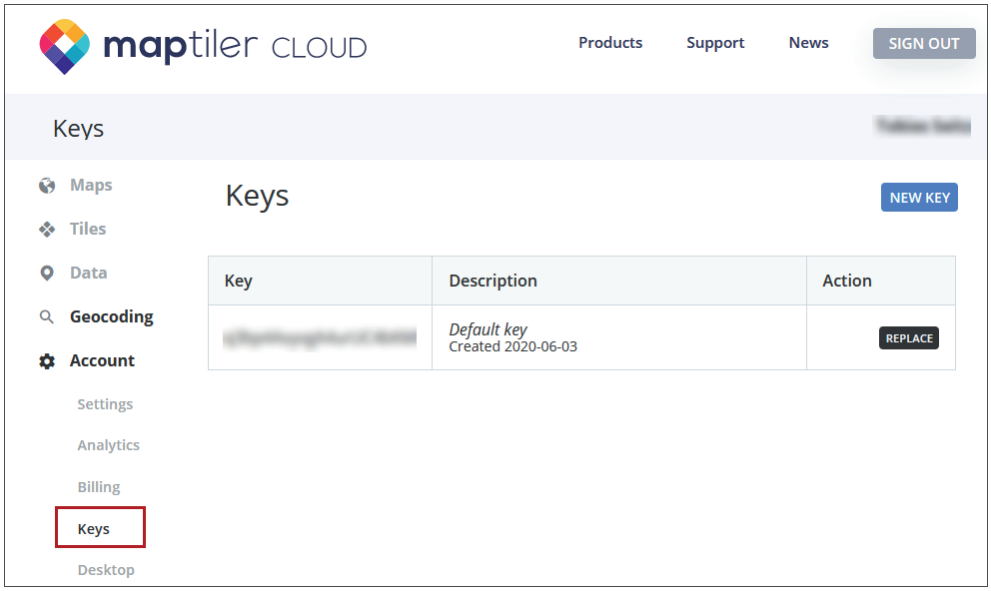
- OpenStreetMap
- OpenSeaMap
- WikiMapia

Use MapTiler
If you enable this option, you can use additional map types from our partner MapTiler.

For this you need an account with MapTiler, for which additional fees may apply. To test the function, you can initially set up a free account with MapTiler.

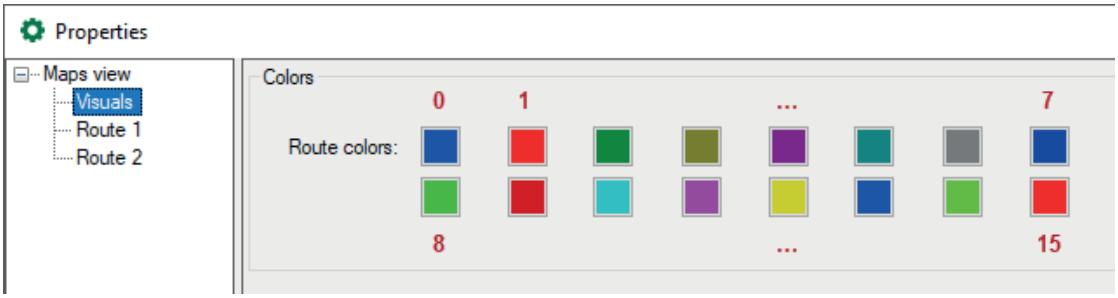
See https://www.maptiler.com/cloud/ for more details.

You must enter a valid API key in order to use MapTiler maps. To get a valid API key, sign in to your MapTiler account and go to *Account – Keys*.



Visuals

On the *Visuals* node, you can define the standard colors to be used when adding more routes to the map view and for dynamic color control. The integer numbers from 0 to 15 are assigned to the 16 colors, and so determine the sequencing.

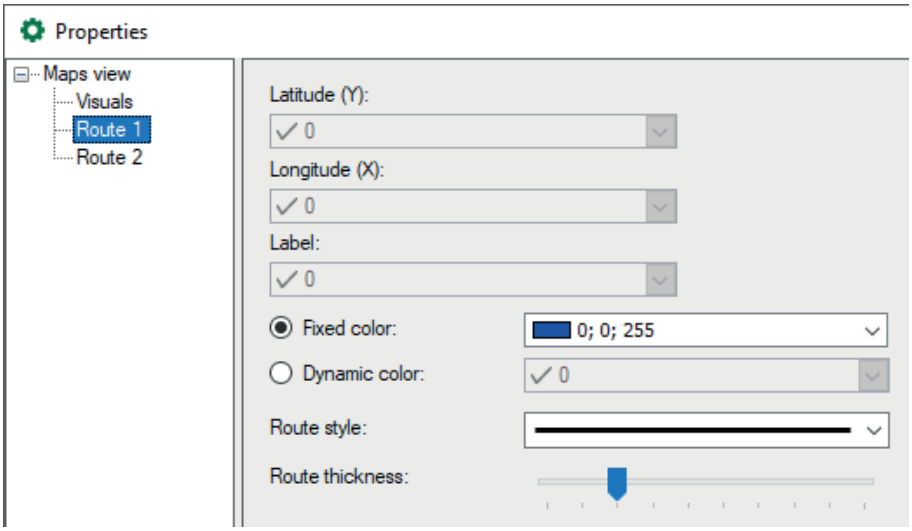


To change a color, click on the color field and select a color from the color selection dialog.

Route settings

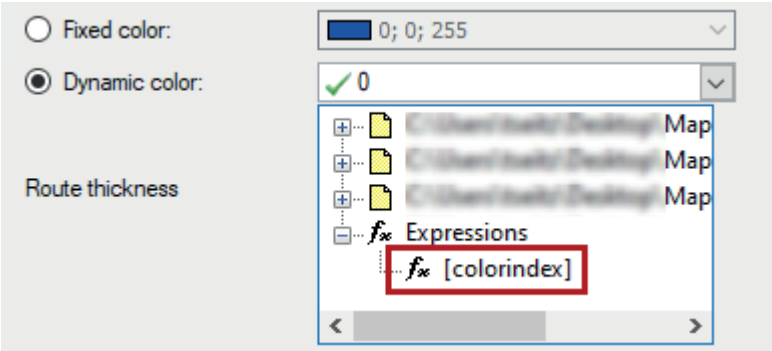
Each route has its own node in which you can set the color and line thickness on the map.

The selected signals for longitude, latitude and labeling are merely displayed, you cannot edit them.



For the color, you can choose between a fixed color, which you set in the field next to it, and dynamic color control.

With dynamic color control, the colors are determined by the values of an integer signal. This signal can be either an input signal or a virtual signal you generated with the expression editor. Only integer numbers from 0 to 15 are included. The numerical values correspond to the color definition in the *Visuals* node.



6.28 InCycle-Expert view (ibaAnalyzer-InCycle)

The view is mainly used visualize and analyze InCycle-Expert modules. The InCycle Expert module makes it possible to divide process cycles into ranges and define characteristic values. In the InCycle-Expert view, you can switch between cycle view and circle view.

The structure and operation of the InCycle-Expert view are similar to the view in *ibaPDA*.

You can create calculation profiles for the analysis in *ibaPDA* and use them with *ibaAnalyzer* as well. Conversely, you can also determine calculation profiles offline with *ibaAnalyzer* and then import them in *ibaPDA* in order to execute the process-synchronous analysis in the InCycle module.

Other documentation

For detailed information about the InCycle-Expert view, see the manual for the product *ibaAnalyzer-InCycle*.

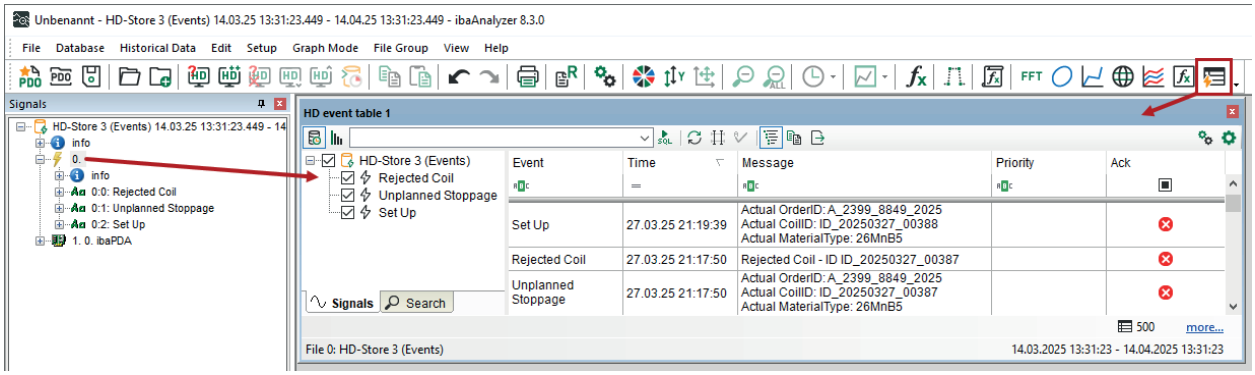
6.29 Event table

Similarly to *ibaPDA*, you can also use the event table for HD data in *ibaAnalyzer*.

The basis for the table is an HD query of an event-based HD store, see ↗ *Access to HD data with ibaAnalyzer, page 219*.

Because the events originate from an HD store, no live interactions, such as resetting alarms, are possible in *ibaAnalyzer*.

You can open the event table from the toolbar or via the *View* menu.



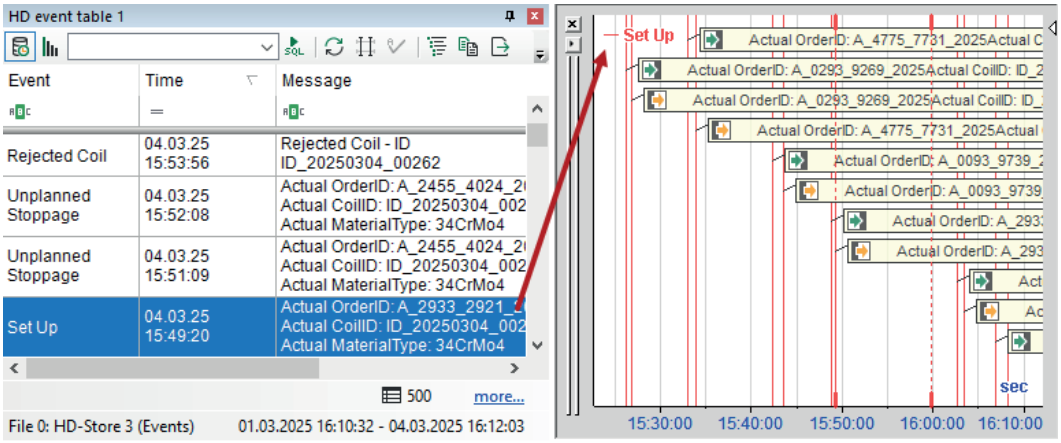
To populate the table, drag events or event groups from the signal tree into the event table. You can use events from multiple HD stores at the same time. In the event table, you can select the available events in a separate HD store tree.

You can customize the event table in Preferences or Graph setup, see ↗ *HD server*, page 94.

Events as text signals

Similar to the query result of an event-based HD store, you can also display the events from the event table as text signals, see ↗ *HD query results of an event-based HD store, page 243*.

To do this, drag the event from the table into the recorder window. Depending on which column you drag the event from, either the event with the message, the trigger signal or the relevant subchannel is displayed.

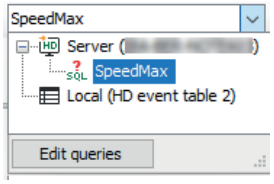


The events are marked by icons as incoming or outgoing events.

6.29.1 Use of the event table

The event table shows the selected events.

The event table toolbar includes the following control elements:

	Switches back to the display of events after a condition query.
	Displays the number of events in a time range. In the dialog window, select the events from the event table and the time range. The table that is shown contains a list indicating the number of events.
	Drop-down list for selecting the desired event query. When you expand the list, you can see all existing event queries and select the one you want. The <Edit queries> button opens the configuration dialog for conditional queries within the table. You can edit existing queries or create new ones in it.
	Executes the event query selected from the drop-down list.
	Updates the display according to the query being executed.
	Shows the events from the marker area of the recorder window.
	Function not available
	Show/hide signal tree
	Copies the rows selected in the event table to the clipboard. Only possible in pause mode.
	Exports the rows selected in the event table to an Excel or text file. Only possible in pause mode.

Status bar
The status bar is located at the bottom of the table. It provides information about SQL queries carried out (e.g. status of the running query, number of results, error messages) and whether a filter is currently active.

Sort and filter

Sort
You can sort the table by each column in ascending or descending order. Click on the header of the column which you want to use as the sort criterion. The arrow shown (down or up) indicates the sort order (descending or ascending). By default, the table is sorted by the *Time* column so that the most recent event is at the top.

Filter line
A filter line is located directly below the column headers. This allows you to filter based on text input or other options.

All columns, except *Trigger* and *Priority*, are filtered according to the character string entered. It is not possible to use placeholders. In the *Time* column, filtering is possible with the operators <, >, <= and >=.

Select a search operation using the <ABC> or <=> button. Then enter the search string or a value. Press <Enter> after making your entry, and the table will be sorted.



The following filters can be assigned for the *Trigger* column:

No filter active.

Incoming Only incoming events are displayed.

Outgoing Only outgoing events are displayed.

The following filters can be assigned for the *Ack* column:

No filter active.

Only events that have not been acknowledged are displayed.

Only events that have been acknowledged are displayed.

Note

As soon as a filter is set, a filter icon is displayed in the column header and the filter is highlighted in green. The set filter is displayed below the table.

You can enter a filter in a field of the filter line and then hide the corresponding column. In this way you can, for example, show only incoming messages without displaying the *Trigger* column.

Click on the filter icon to edit the filter for the column in question.

A detailed description of the filter functions can be found in the *ibaPDA* documentation.

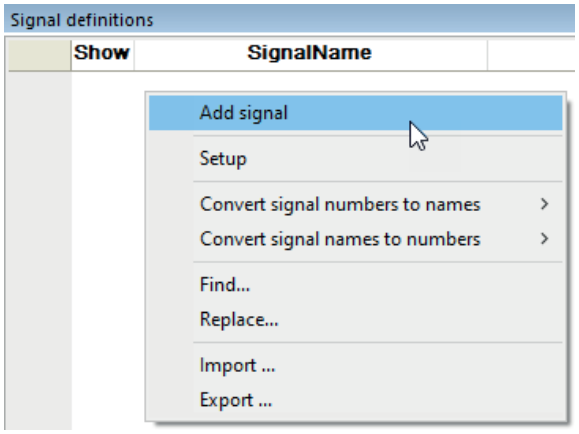
7 Creating new signals

In *ibaAnalyzer*, you can use not only the original signals (raw data) but also create new signals and incorporate them into further calculations. This opens up extensive analysis possibilities that go far beyond simply using the raw data. *ibaAnalyzer* provides two different methods for creating new signals.

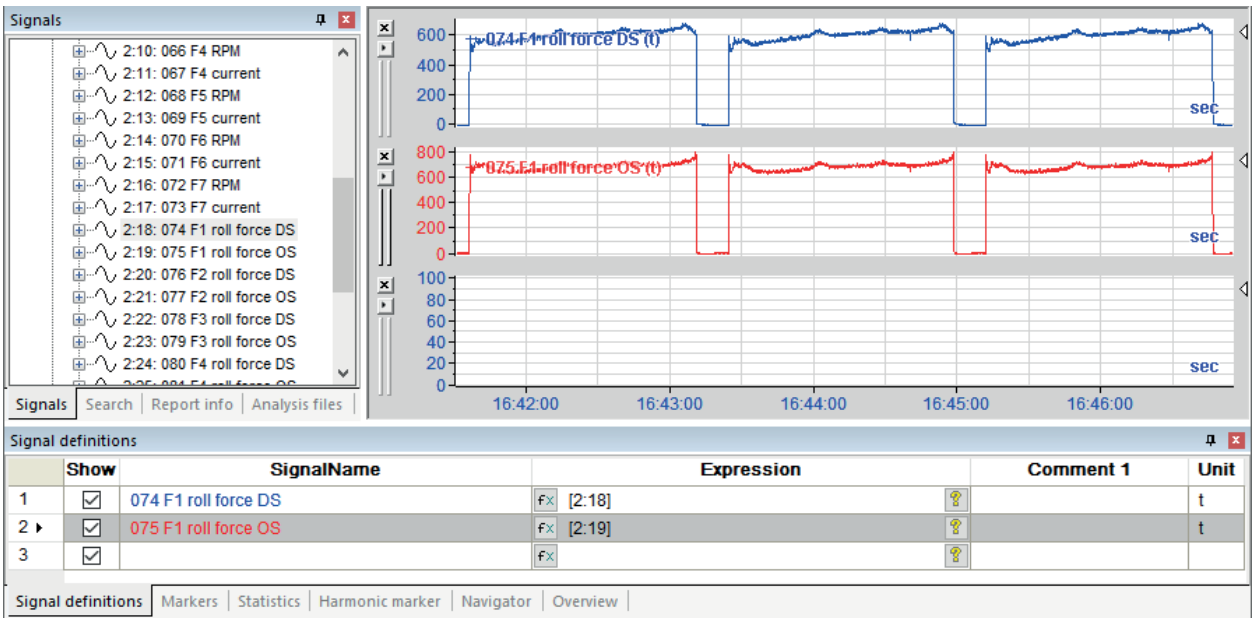
7.1 Adding signal in the signal table

You can add new signals at any time in the signal table on the *Signal definitions* tab. No data file needs to be open for this.

Open the context menu in the signal definitions and select *Add signal*. This even works if there are already signals in the table.



The command inserts an empty row in the table and a corresponding graph in the recorder window.

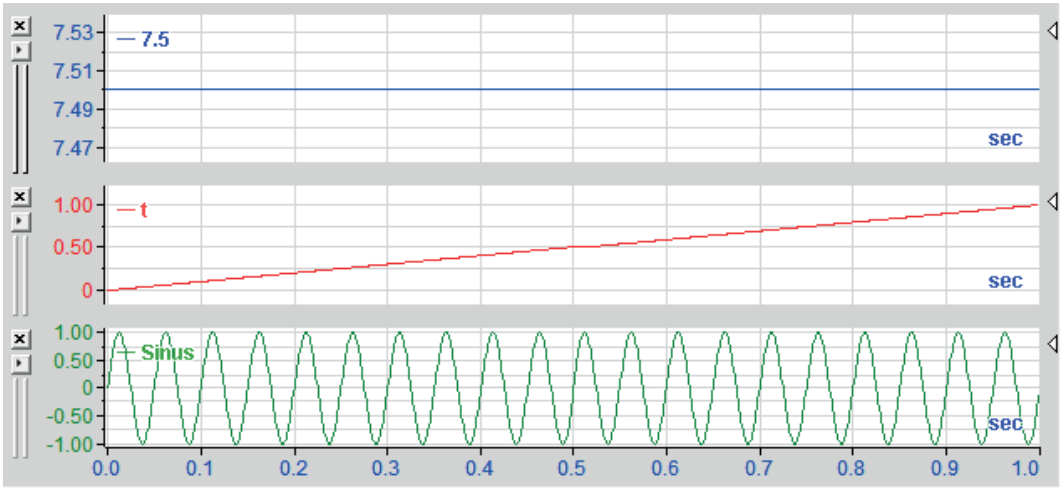


You can then enter any expressions and calculations you want in the new row in the *Expression* column. These include:

- raw data (original signals)
- constant values
- expressions for generating artificial signals using the functions of the expression builder
- mathematical operations with artificially generated signals and raw data as operands

Expressions as signals
You can also use simple expressions directly as signals. The figure below shows some examples:

- constant value (7.5)
- generation of a time line with the TIME function
- generation of a sine signal based on the time line



Signal definitions

	Show	SignalName	Expression	Unit
1	☑	7.5	fx 7.5	
2	☑	t	fx Time(1000, 0.001)	
3	☑	Sinus	fx Sin(2*Pi()*20*[t])	

For an explanation of the functions, see *ibaAnalyzer* manual part 3 *Expression builder*.

New signals from original signals

The combination of original signals also creates new signals, as the following figure illustrates.



Duplicate signals

When you select *Duplicate signal* from the context menu of a signal in the table, the content of the selected signal is copied to a new signal. That is useful if you only want to make minor modifications to an existing lengthy expression.

Use of the new signals

Newly generated signals are saved in the analysis (PDO file). If the analysis is opened without a data file, these expressions are present, but have no values. The expressions are only assigned values when you open a data file.

These newly added expressions can themselves be operands in other expressions. So the new expressions are also included in the signal tree of the expression builder. They are not displayed in the signal tree window.

Note

When you delete a graph with these signals, such as if you close a graph with <x>, the expressions are also deleted.

If you have disabled the Undo/Redo function in the Preferences, the deletion is final.

7.2 Logical expressions

The logical expressions are used to generate virtual signals, which are then stored as a fixed component of the signal tree in the analysis.

Compared to the definition of virtual signals in the signal table, logical expressions have the advantage that they cannot be deleted by closing a graph in which they are used.

You can use logical expressions to generate analog signals, digital signals and vectors (arrays). The functions of the expression builder are available for definition purposes.

An import/export function is available to aid the configuration of large numbers of logical expressions, ↗ *Export and import of logical expressions, page 184*.

You can also use the logical expressions as a source for *Computed columns* in the extraction functions and the report generator. As the calculations or expressions in these areas are often the same, you can reference logical expressions there, so avoiding duplication and errors.

For more information on calculated columns in database extraction, see *ibaAnalyzer* manual part 4, chapter *Computed columns*.

Tip



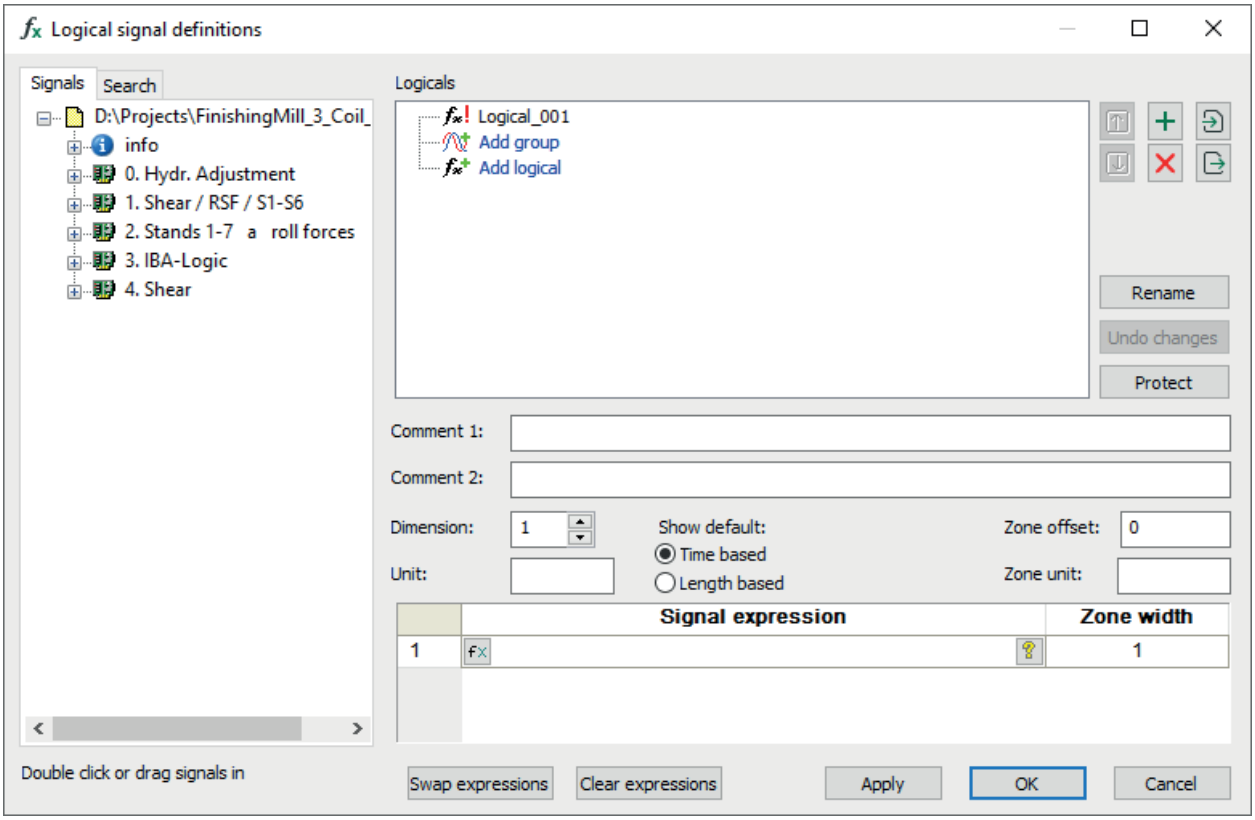
If you have already made extensive configuration settings under *Computed columns*, you can use a matching function in the relevant dialogs to automatically define the logical expressions.

7.2.1 Logical expressions dialog window

Open the dialog window for defining logical expressions using the toolbar button.



In the *Logical expressions* dialog, you can create new logical expressions and edit existing expressions.



Signals tab
On the left-hand side of the dialog window is a signal tree where you will find the original signals from the data file as well as the generated expressions.

Search tab
You can search for signals in this tab as usual. You can drag the search results directly into the configuration table for the signal definition.

Logicals
This field shows the logical signals that have already been generated (in the image above there is still an empty standard signal).

Buttons for signal editing

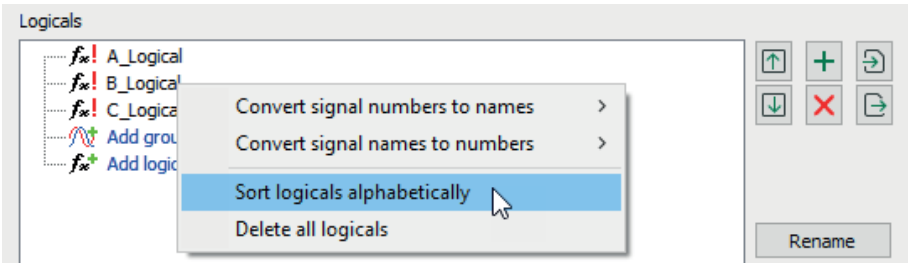
	Move the selected logical signal up or down in the list.
	Add a new logical signal.
	Delete the selected logical signal.
	Import a signal list or export the current list, see ↗ *Export and import of logical expressions, page 184*.
<Rename>	Rename the selected logical signal. For more information, see ↗ *Renaming logical expressions, page 187*.
<Undo>	Undo actions on the selected signal before clicking <Apply> or <OK>.
<Protect>	Protect all logical signals with a password.

Red symbols on the icon of each signal indicate upcoming changes to the signal (!) or deletion of a signal (x). Confirm these actions with the <Apply> or <OK> button.

The information shown below refers to the structure and contents of the signal.

Context menu in the Logicals area

You can use the context menu to change the signal reference (signal name or number), sort the logical signals alphabetically, or delete all definitions.



Dimension

The dimension describes the number of related, continuous-time or continuous-length expressions that can then be displayed in a 3D view. You can enter a numerical value from 1 to 2048.

Normal, simple signals have the dimension 1.

To create a three-dimensional profile view, however, a set of measurement series for a physical measurement variable must exist which are assigned to the third spatial coordinate. For more information, see ↗ *Creating vector signals (array), page 177*.

Comment 1 and 2

Like for measuring signals of a data file, you can enter two comments for logical expressions which provide more information and can be used in the legend, for example.

Unit

The unit is shown in the legend and in the signal table.

Show default: Time based/Length based

This option allows you to preset whether the logical signal is to be time based or length based.

Zone offset and Zone unit
In the case of multi-dimensional signals (vector signals), these two settings are used to provide a realistic representation of profile measurements.

For more information, see ↗ *Zone control for vector signals, page 181*.

Signal expressions table
Enter an expression representing the desired signal in the rows in this table. If you want to transfer raw signals or existing expressions, you can transfer them from the signal tree in the dialog window to the expression table using drag & drop or by double-clicking.

For more extensive expressions using mathematical functions, open the expression builder using the <fx> button inside the table row. Use of the expression builder is described in part 3 *Expression builder*.

<Swap expressions>
Use this button to reverse the order of the rows of a multidimensional signal (vector).

<Reset expression>
Use this button to clear the content from the fields in the *Signal expressions* column. The other settings of the logical signal are retained, such as name, dimension, unit etc.

<Apply> and <OK>
Use the <Apply> button to apply changes or deletion commands without closing the dialog. The <OK> button does the same, but closes the dialog.

7.2.2 Generating a simple signal

The following example shows you how to generate a simple signal with a logical expression.

Example

1. Open the *Logical expression*s dialog.

→ The standard signal "Logical001" is suggested.

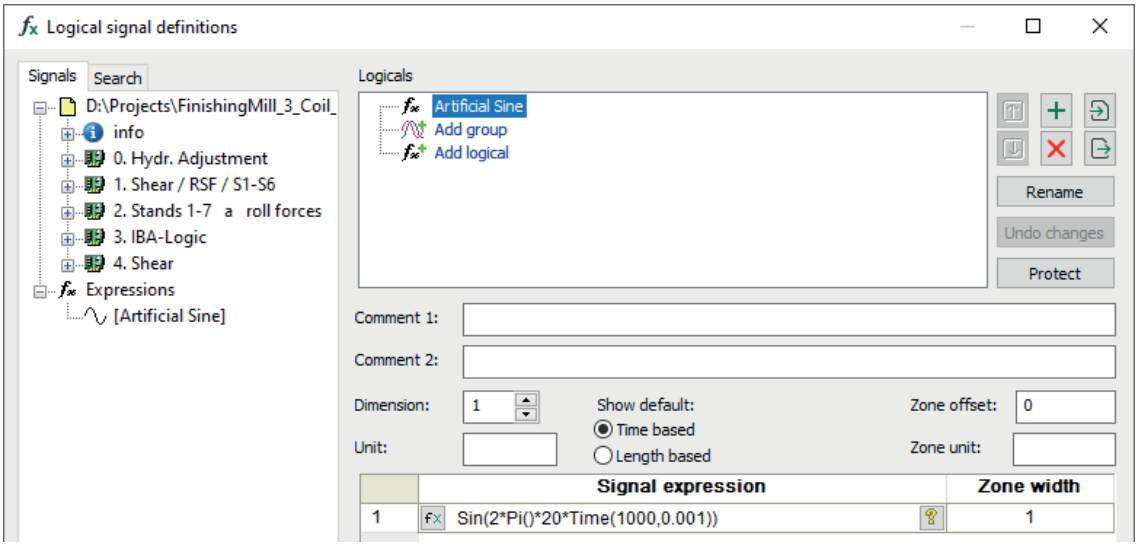
2. Rename the standard signal:

   Select the signal "Logical001", click on <Rename>, and enter a signal name (e.g. "Artificial Sine").

3. Make the following settings:

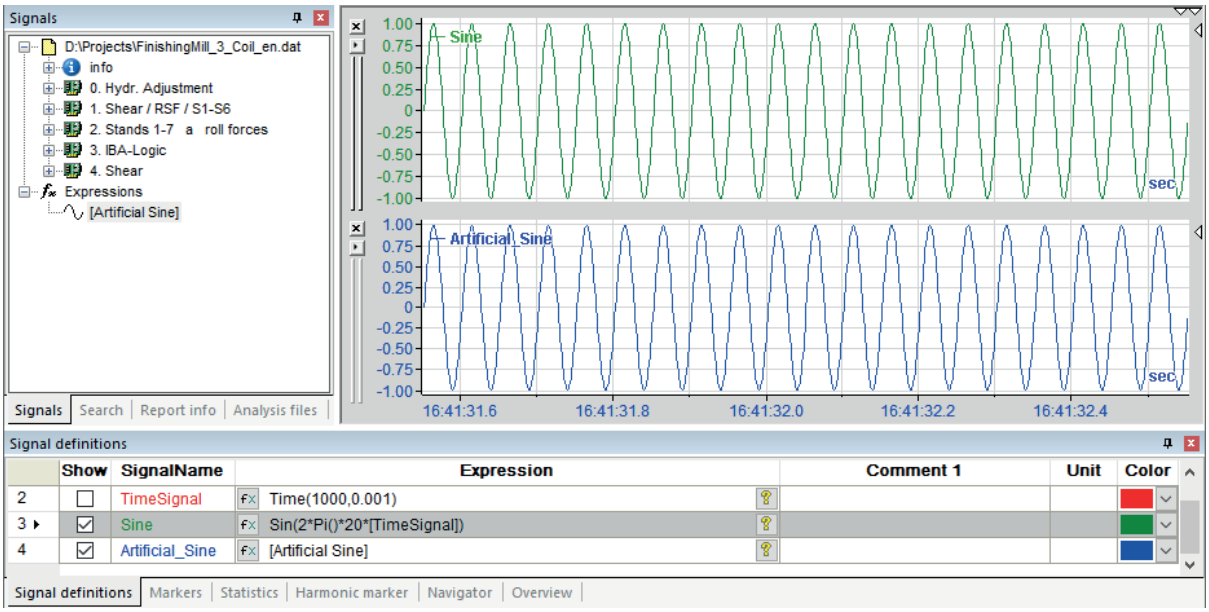
   Dimension = 1, no unit, time-based.

4. Now enter the expression for a sine curve in the "Signal expressions" table row, or use the expression builder as an aid.
   Example: A sinusoidal signal with a frequency of 20 Hz and a duration of 1 s is generated with this expression: `Sin (2*Pi()*20(Time(1000,0.001)))`



5. Save the new signal with <OK>.

→ The new "Artificial Sine" signal is now available in the signal tree window and in all other signal trees. You can use it like a "real" signal.



The image also shows that the logical signal "Artificial Sine" from the logical expressions is included in the signal tree. But the expression that was generated only in the signal table is not listed in the signal tree, see ↗ *Adding signal in the signal table, page 169*.

7.2.3 Creating vector signals (array)

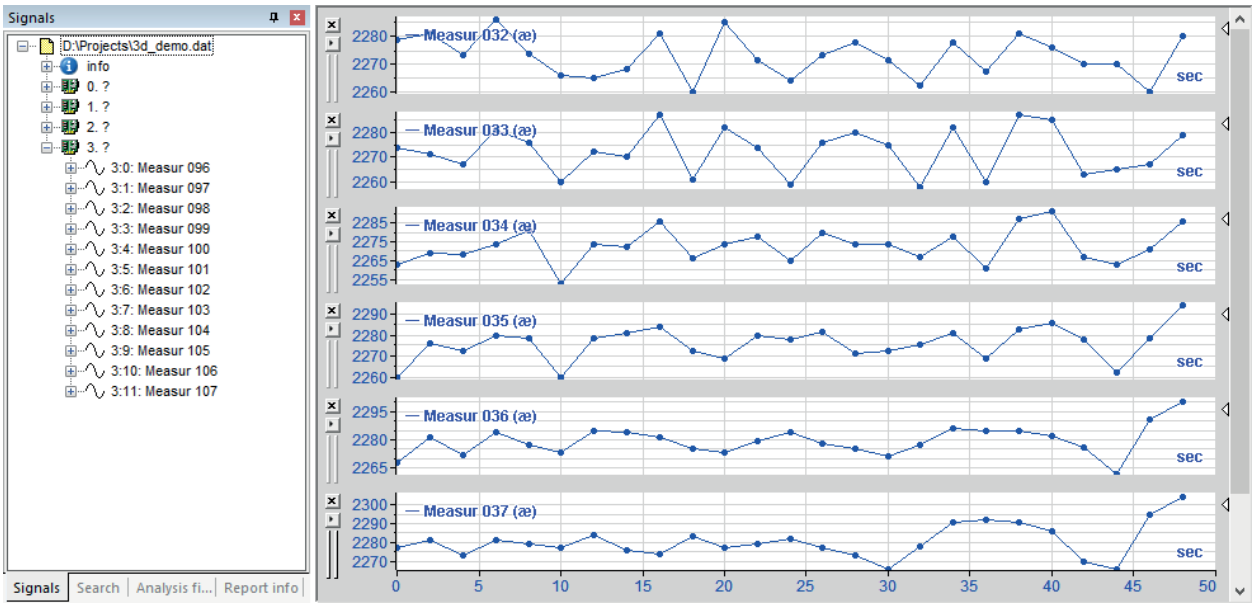
To enable 3-dimensional views, a signal of the ARRAY type is required (also vector signal).

The following example shows you how to generate a vector signal.

Example of a multi-dimensional signal for a strip thickness profile

The thickness of rolled strip is measured in a rolling mill. In order to achieve a good strip quality, the strip thickness should, of course, be the same at all points. The thickness is therefore measured not just at one point, but over the entire width and length of the strip. In this example, the thickness gage provides 108 measurement signals distributed across the strip width. This means that the strip width is divided into 108 measuring zones, with each measuring zone providing thickness measurement values for as long as the strip passes below the measuring device. The duration of all the signals is the same, because they are all distributed over the entire strip length.

The data file in *ibaAnalyzer* only shows a series of modules and signals that are of little significance when displayed individually.



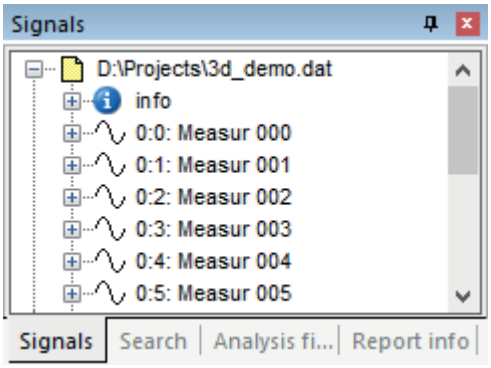
The screenshot above shows that the last signal is numbered 107. Modules 0 to 3 thus contain the signals "Measur 000" to "Measur 107".

Procedure

1. The consecutive listing of signals in the signal tree window is more suitable for further analysis purposes.

   So open the context menu in the signal tree and choose *Linear numbering*.

→ The signals are now shown without modules in the signal tree. Furthermore, the signals are no longer identified by [module number:channel number], but by consecutive numbers from 0 to 107.



2. Open the *Logical expressions* dialog.

→ The standard signal "Logical001" is suggested.

3. Rename the standard signal:

   Select the signal "Logical001", click on <Rename>, and enter a signal name (e.g. "VectorSignal").

4. Make the following settings:

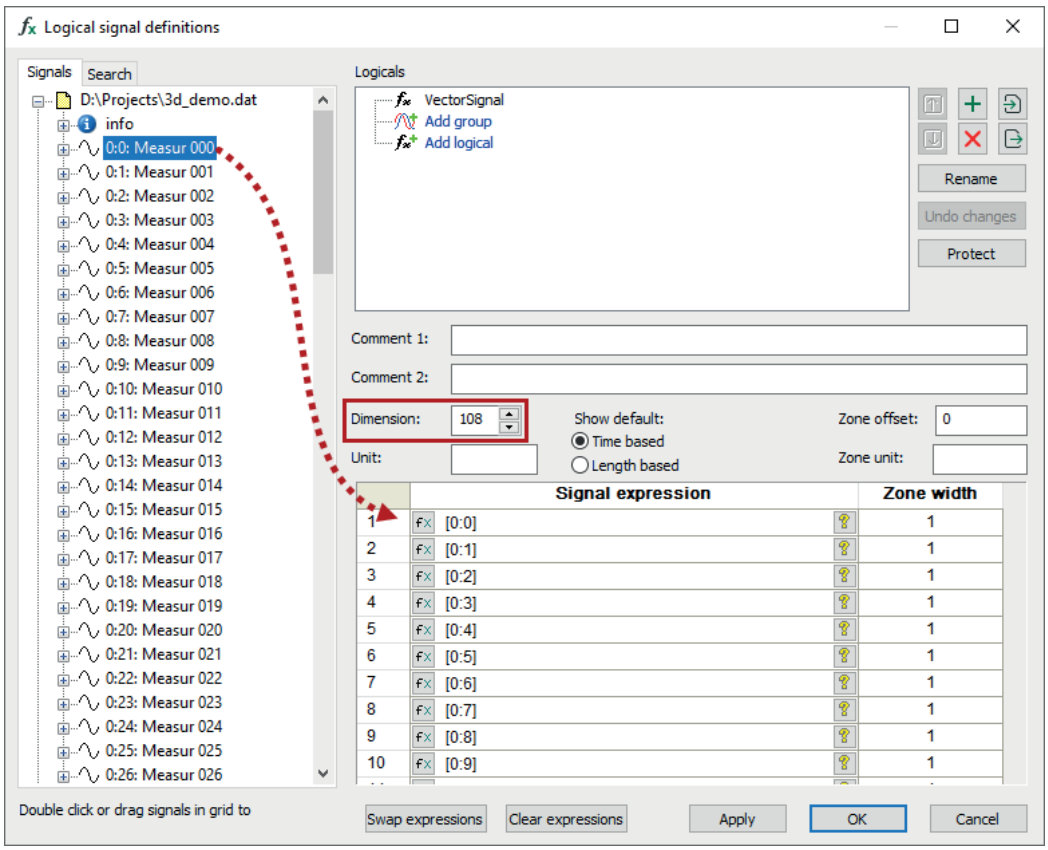
   Dimension = 108, no unit, time-based.

→ The table contains 108 rows (0…107).

5. Select the first row in the table.

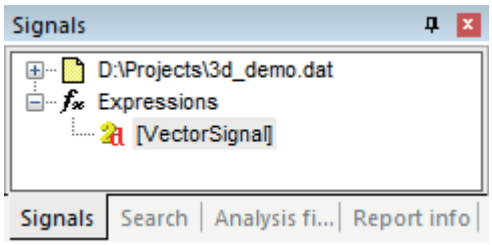
6. In the signal tree in the dialog window, double-click on the first signal ("Measur 000").

→ All 108 signals are included in the table.



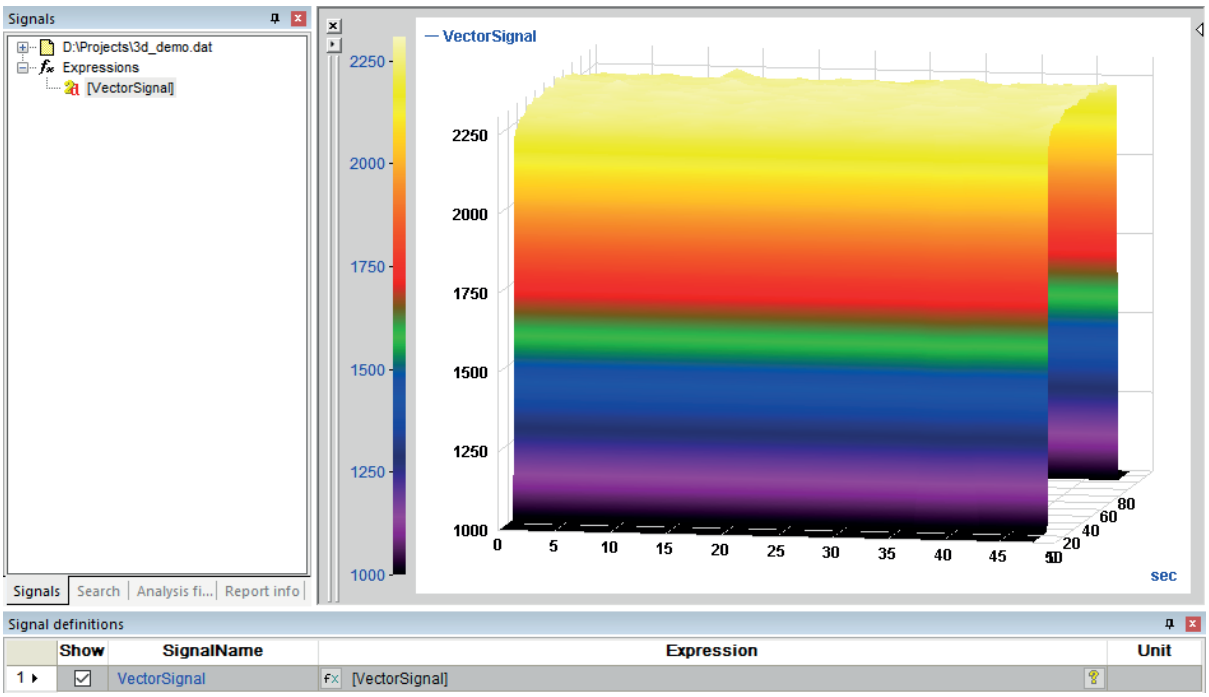
7. Save the new signal with <OK>.

→ If you close the signal tree of the data file using the <+> button, you will find the new vector signal under the *Expressions* node.



8. Open the vector signal in the recorder window (by double-clicking or using drag & drop).

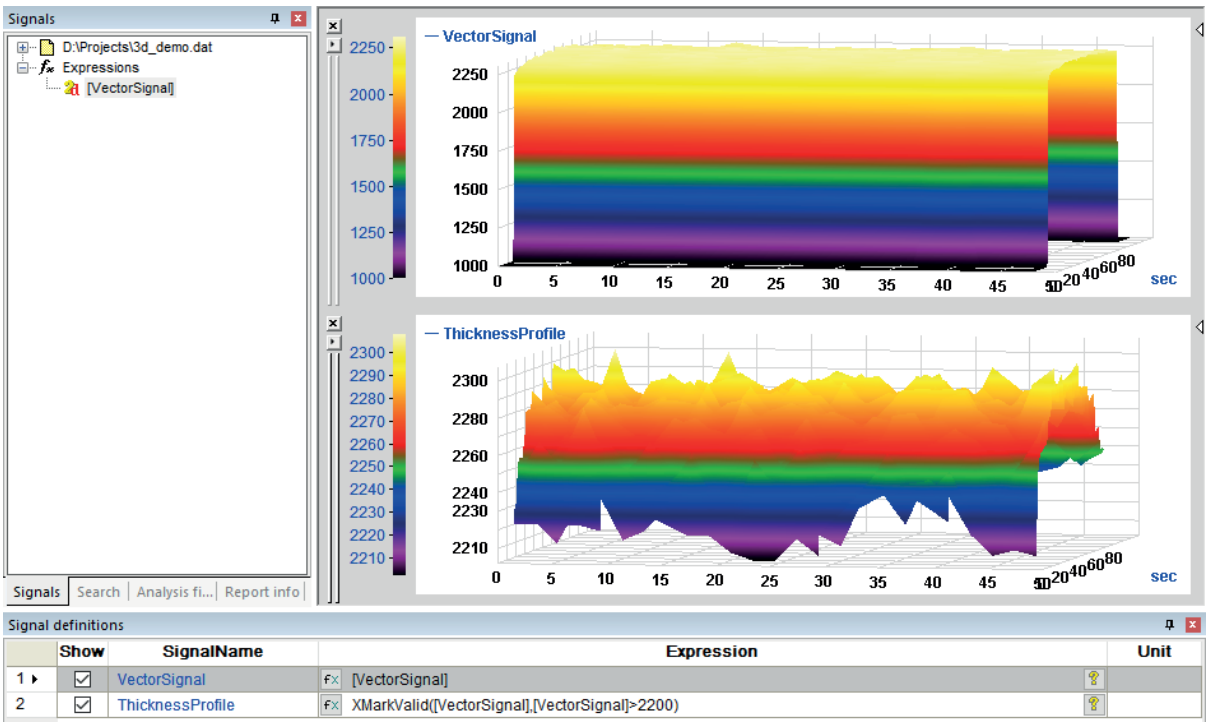
→ Due to the multi-dimensionality of the signal, *ibaAnalyzer* selects a 3D view for the graph.



The entire value range is displayed. In practice, it is often more interesting to check the fluctuations in thickness within the range of the target value. In the graphic, this corresponds to the horizontal surface area at the top.

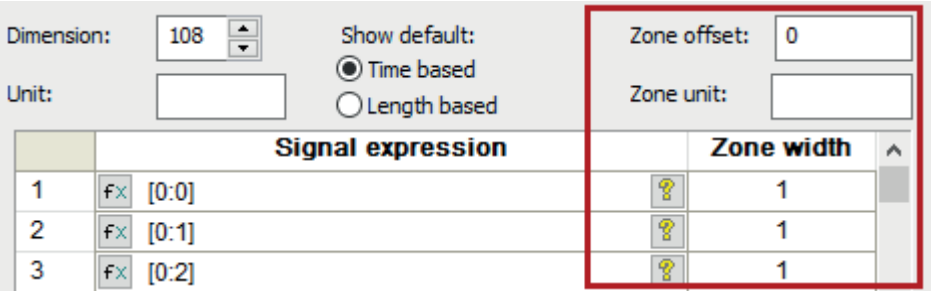
9. In order to obtain a more informative visualization in this respect, you can use the "XMarkValid" function from the expression builder to cut out the relevant portion of the measured values, as shown in the graphic below.

   For more information, see *ibaAnalyzer* manual part 3, chapter *X-axis operations*.



7.2.4 Zone control for vector signals

By default, only "1" is entered in the *Zone width* column. This means that all zones have the same width and the measured values are always at equal distance, respectively. The geometrical width is not important for this and the scale at the Y-axis in the 2D top view only shows the number of zones.



You can achieve a geometrically correct representation on the Y-axis if you specify the zone widths according to the geometric width and the corresponding physical unit, such as mm or cm.

This is also helpful for measuring devices with measurement zones of different widths. To display the measured values realistically, you can specify the zone width for each zone individually. As a result, the measured values in the 2D top view and the 3D view are distributed geometrically correctly along the Y-axis.

If you adjust the zone width, the measured values are in the middle of the zone.

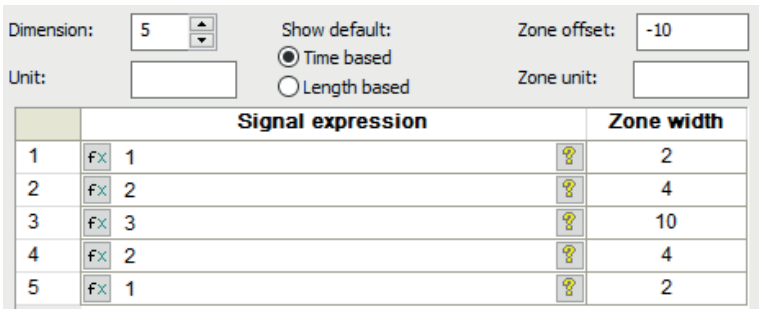
Specify zone width and zone offset
You can only enter positive, numerical values for the zone width.

To assign a large number of zones the same value, enter the value and then click on the header of the *Zone width* column. All cells below the cell where the cursor is placed are filled with this value.

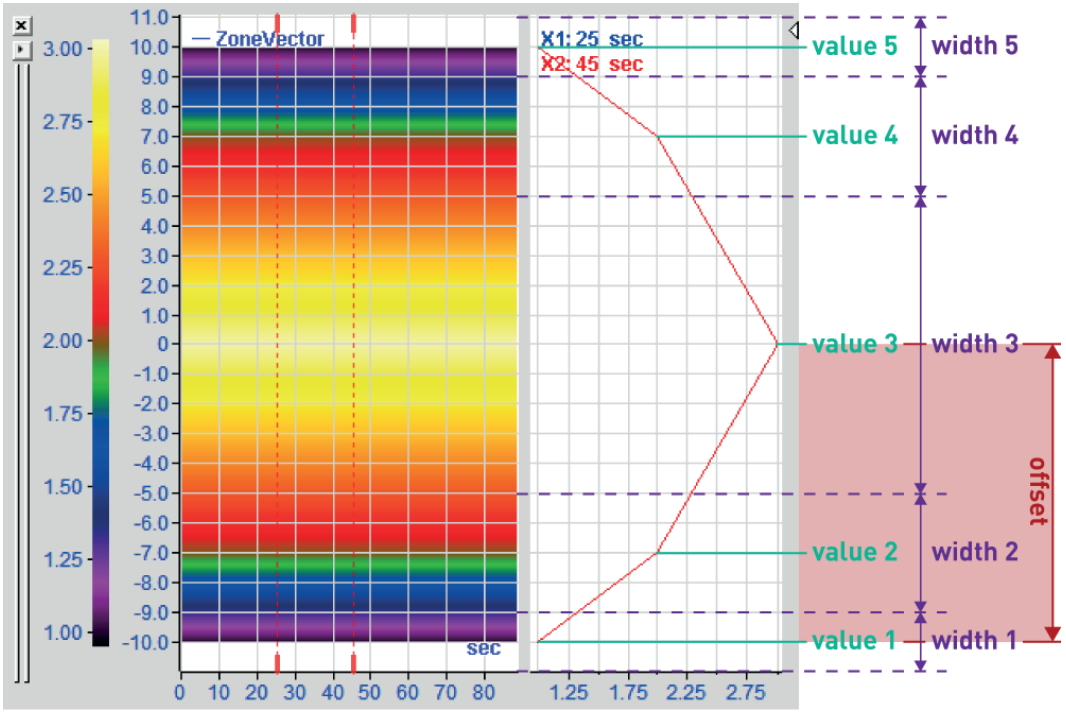
To correct the position relative to the zero line, you can enter an offset between the zero line and the middle of the first zone. The offset can be a positive or negative value.

Example of a vector signal with five zones

The following example shows the relationships using a simple vector signal with five zones:



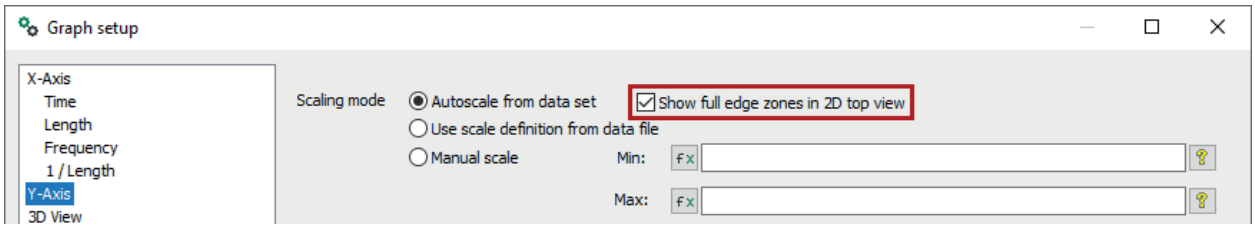
These settings generate a vector signal that is displayed in the graph.



The first signal is positioned according to the offset (here at -10). The sum of the zone widths amounts to 22 so that the last signal is displayed at +10. The distance of the other signals to each other results from the average of the respective zone widths.

ibaAnalyzer calculates the color gradients by means of linear interpolation between adjacent signals. In the outer half of the outermost signals, there are no colors, as interpolation is not possible.

In the Y-axis settings, you can decide whether or not to display the empty ranges when autoscaling the graph.



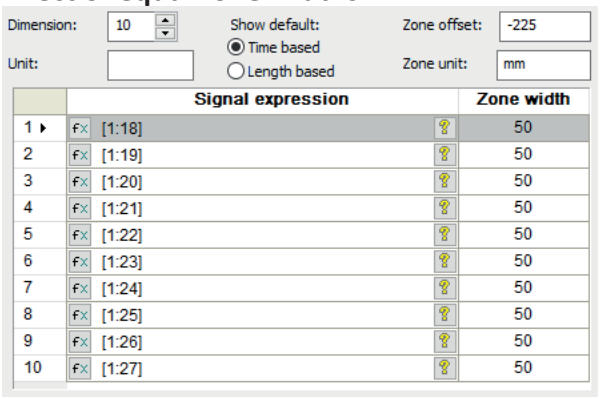
Comparison of views with different zone widths

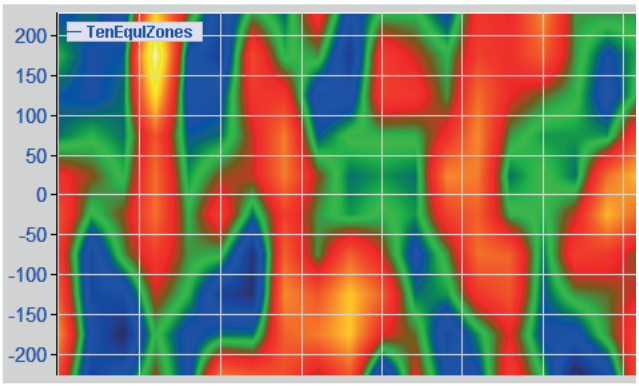
This example shows the difference in appearance when using various zone widths.

A vector signal with 10 zones is configured for a measuring width of 500 mm:

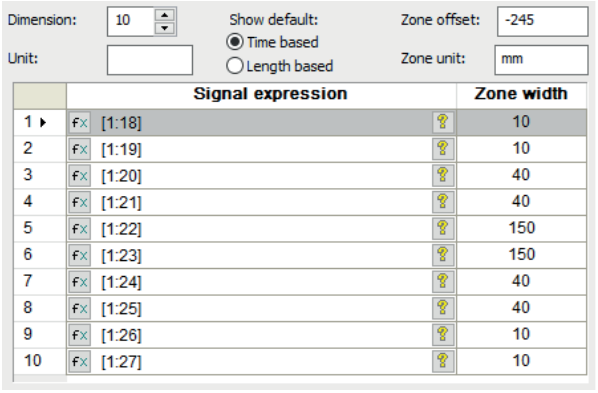
- 10 equally sized zones 50 mm wide
- 10 differently sized zones between 10 mm and 150 mm wide

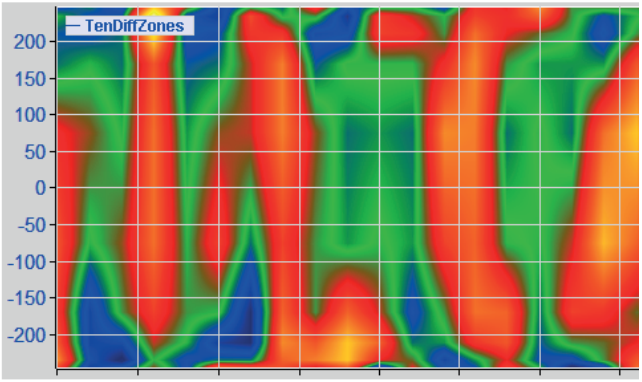
Effect of equal zone widths





Effect of different zone widths

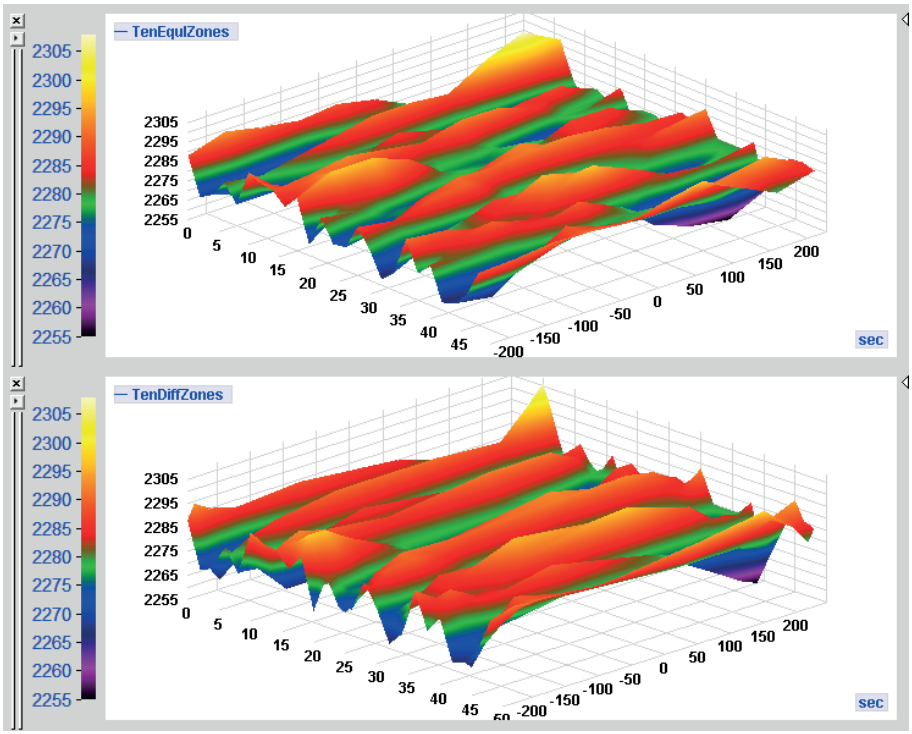




The illustration with different zone widths clearly shows the widening of the middle range (-150 to 150).

Differences in the 3D view

The 3D view also incorporates the zone widths. The following illustration shows the vector signal with the same zone widths (top graph) and different zone widths (bottom graph).



7.2.5 Export and import of logical expressions

Importing and exporting logical expressions is useful when handling many or very complex logical signals. You can also use the Export function to back up your work, and make the logical signal definitions available to other users or on other computers.



Import button



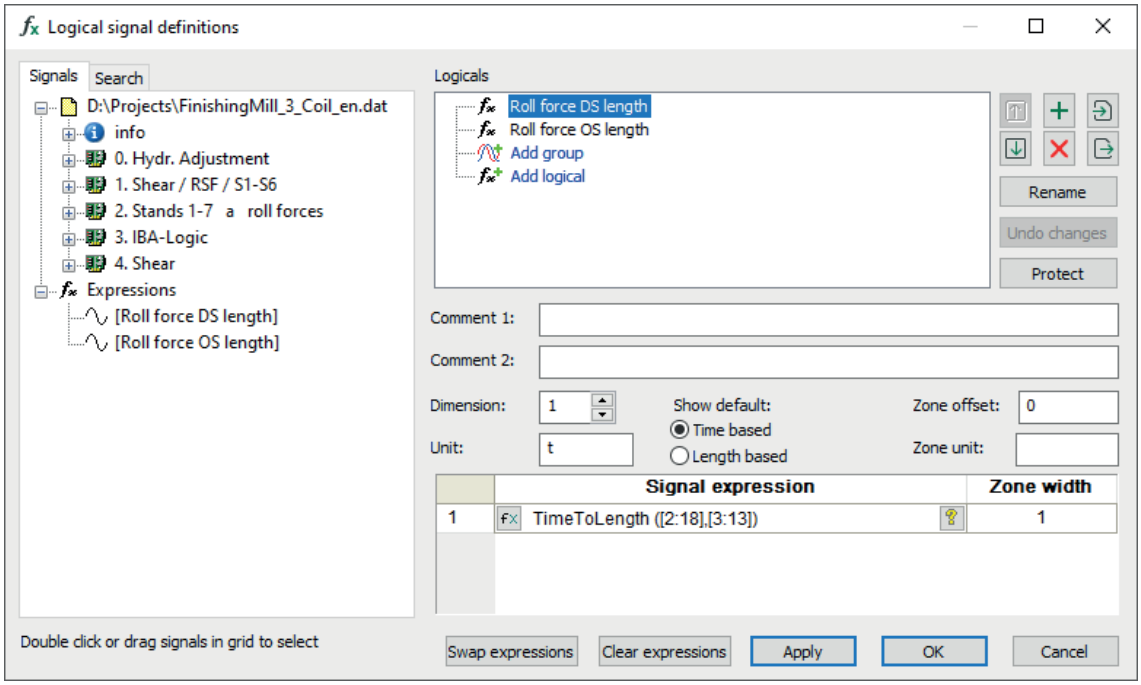
Export button

Similarly to the signal definition table, you can export the logical signals to a text file. You can then edit the text file in a text editor or MS Excel. It is sometimes easier or more efficient to configure large volumes of data in a spreadsheet program than in a configuration dialog.

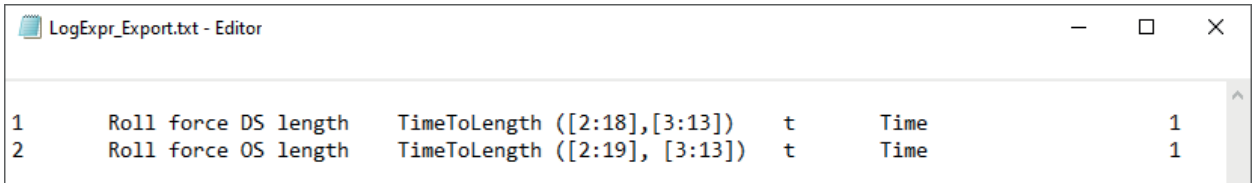
Define one or two example signals in the *Logical expressions* dialog in order to obtain the correct formatting of the text file. *ibaAnalyzer* generates text files with tabs as separators between the data points.

Example
Define one or two signals and then export them.

The Import/Export buttons always apply to all logical signals.



In a text editor, such as Notepad, the exported example file looks like this:



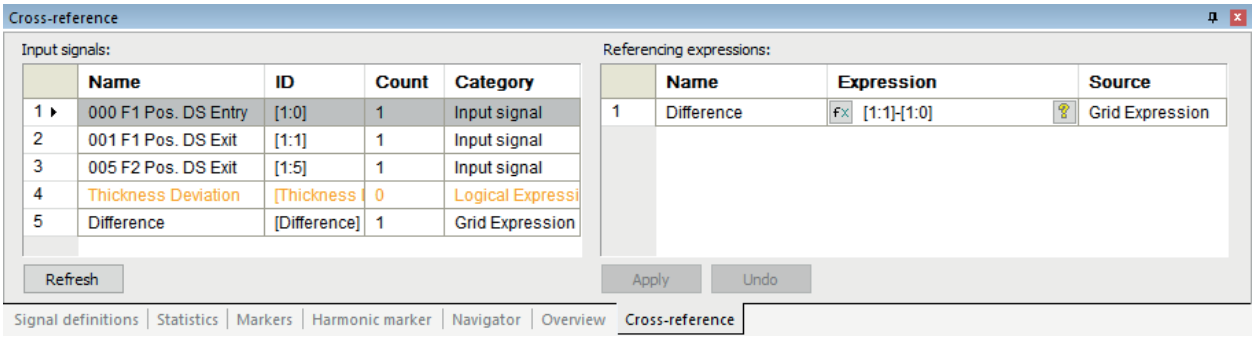
In MS Excel, the columns are distributed like this:

	A	B	C	D	E	F	G	H
1	1	Roll force DS length	TimeToLength ([2:18],[3:13])	t	Time			1
2	2	Roll force OS length	TimeToLength ([2:19], [3:13])	t	Time			1
3								

If you keep the correct file formatting, you can edit more data sets using these tools. Save the file again as a text file to import it into the logical expressions.

7.2.6 References of logical expressions

On the *Cross-reference* tab of the signal table, you can review the usage and referencing of input signals and logical expressions. The tab shows a list of all input signals and expressions, and their usage in expressions and calculations. You can edit the expressions directly in the *Referencing expressions* table.



For more information see *ibaAnalyzer* manual part 1, chapter *Cross-reference tab*.

7.2.7 Groups for logical expressions

Grouping of logical expressions enables clearly structured management in *ibaAnalyzer*. Groups and subgroups allow you to organize logical expressions hierarchically, and edit them efficiently.

Each logical expression can only belong to one group, but can have multiple higher-level groups.

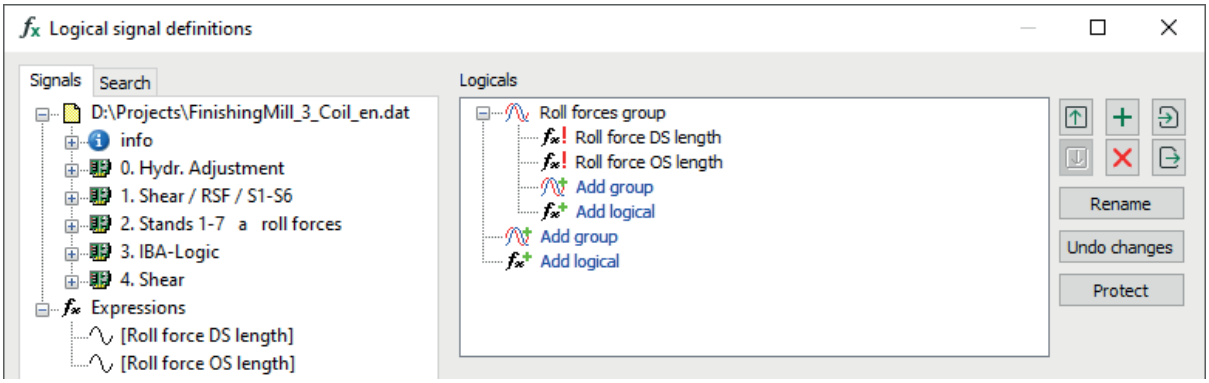
This feature is also backward-compatible. When you open an analysis with groups in older *ibaAnalyzer* versions, backslashes in the expression names indicate their group membership in a flat list.

An expression is renamed internally in *ibaAnalyzer* when you assign it to a group. For more information, see ↗ *Renaming logical expressions, page 187*.

Creating groups

The *Logical Expressions* dialog provides two options for creating groups.

- In the *Logicals* area, click on the blue *Add group* link. Enter a name for the group. In this way, you can also create further subgroups in this group.



- You create a logical expression in the *Logicals* area. Rename the expression by prefixing the desired group names in front of the new expression name, separated by a backslash ("\"). You can likewise include subgroups, also with backslashes, within the group name.

  You do not need to repeat the names of higher-level groups if you have already created the expression in the group.

  Example: The logical expression "TemperatureMax" belongs to the "Sensors" group and the "Temperature" subgroup. You can name the expression like this: "Sensors\Temperature\TemperatureMax".

Editing logical expressions and groups

To add logical expressions to a selected group, click on the blue *Add logical* link or on the green <+> button.

You can use drag & drop to move expressions and groups you previously created to other groups. Multiple selection is possible.

To delete groups or expressions, click on the red <X> button. The group or expression is then marked for deletion. If a group has subgroups, the subgroups and all expressions in them are deleted.

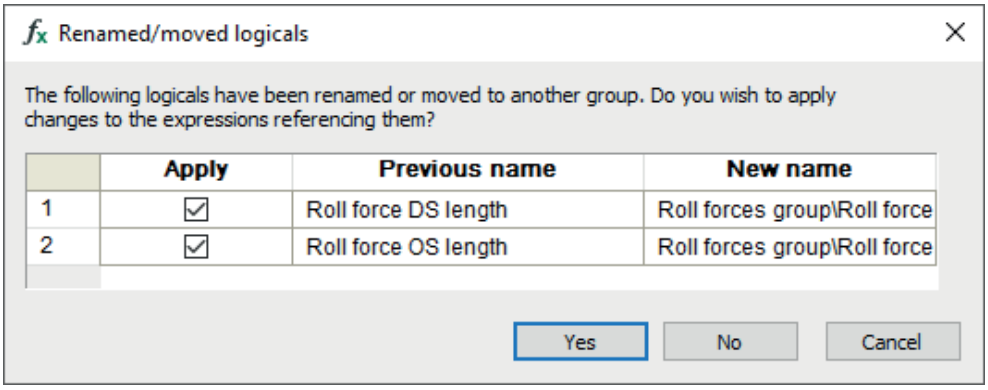
Groups in the report generator

The grouping of report fields in the report generator works in a similar way to the grouping of logical expressions. However, the report designer uses dots (".") instead of backslashes. *ibaAnalyzer* automatically converts the backslashes into dots before the group structure is displayed in the report designer.

7.2.8 Renaming logical expressions

You can rename expressions and groups using the <Rename> button in the *Logical Expressions* dialog. An expression is also renamed internally in *ibaAnalyzer* when you assign it to a group.

When you confirm the changes with <Apply> or <OK> in the *Logical Expressions* dialog, a dialog opens up in which you can automatically adjust all referenced expressions. This dialog shows a list of the old and new names of the logical expressions together with checkboxes. By default, all renamed or regrouped expressions are selected to apply the adjustment.



Yes: All referenced expressions of the selected logical expressions are adjusted.

No: No changes are made elsewhere. The change at the selected position remains.

Cancel: Return to the Logical Expressions dialog without making changes elsewhere. The change at the selected position remains.

This function makes management easier, and ensures that all references to logical expressions remain valid after renaming or regrouping.

8 Macros

The macro function allows you to define and save complex and standardized analysis functions as macros. You create macros using the familiar functions from the expression builder. They can be used universally because placeholders replace the input and output parameters. Macros make analyses clearer and easier to understand.

Macros belong to the analysis and so are saved in the analysis file. You can also save macros globally, and so make them available for other analyses. You can export and import macros in order to swap them.

Use the macro designer to create and edit macros. You can open the macro designer from the toolbar or via the *Setup* menu.



All functions of the expression builder are available in the macro designer. Using macros offers the following advantages:

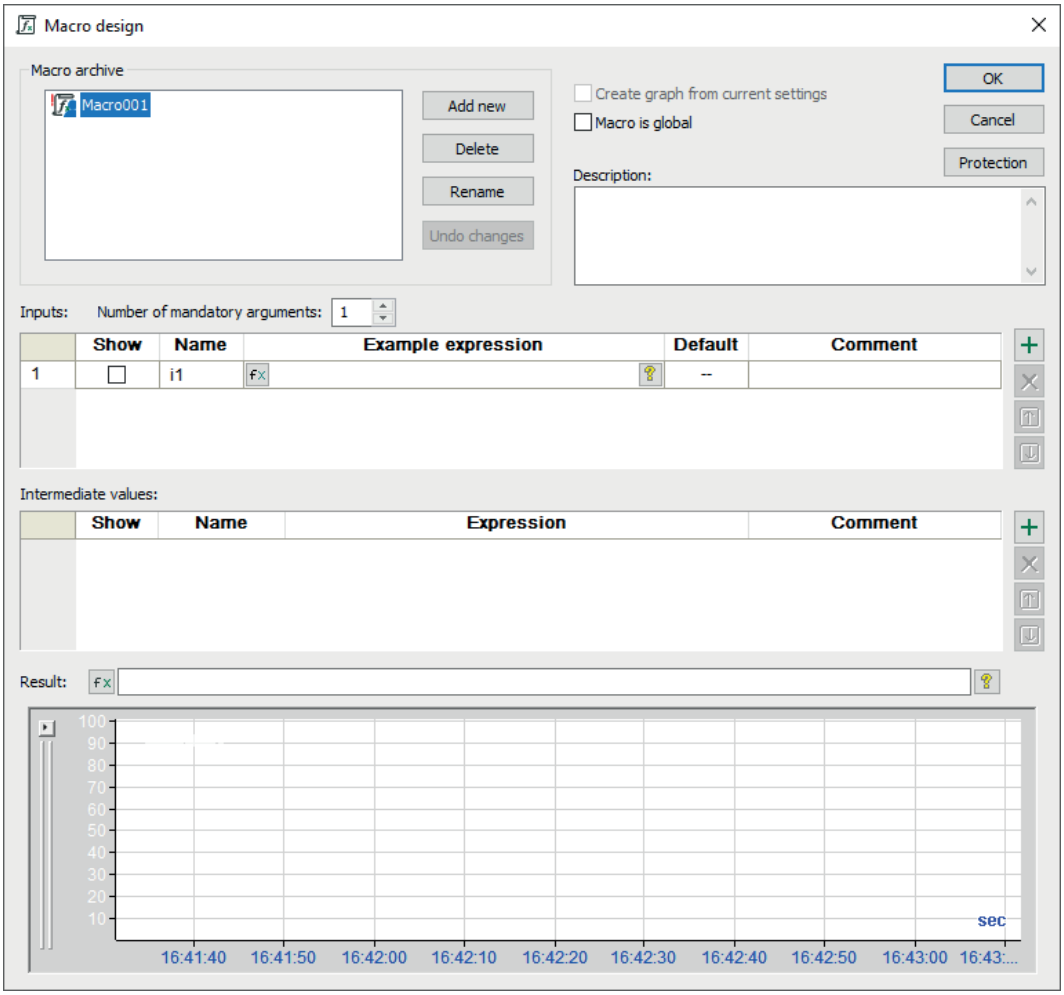
- If you repeatedly perform calculations on different input signals, macros significantly reduce the effort required to create the expression functions.
- You can hide complex calculations in a macro so that the *ibaAnalyzer* signal table only displays short, easily comprehensible expressions.
- You can store general calculations as macros in a library so that they are also available for other analyses.
- You can export and import macros. Other users can then also use the macros. Because analyses are often specific to a data file, it can be difficult to exchange entire analyses.
- Using macros does not require any programming skills.
- You can protect macros from unauthorized changes with a password.

8.1 Macro designer

Use the macro designer to create and edit macros. You can open the macro designer from the toolbar or via the *Setup* menu.



The macro designer looks like this:



Macro archive

The macro archive lists all existing macros for the selected analysis. You can create new macros, change or delete existing ones or change their names.



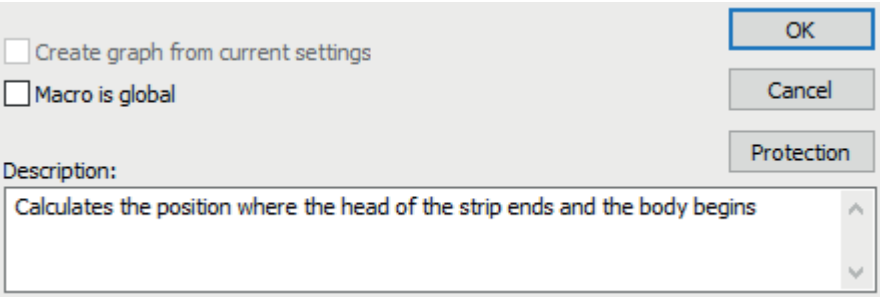
You can export and import the macros; see ↗ *Export and import of macros, page 199*.

Create graph from current settings
If you enable this option, the created macro is executed immediately and displayed as a new graph in the *ibaAnalyzer* recorder window. This option is not available if the entries in the expression builder are invalid, or if you have not defined arguments in the *Inputs* area.

Macro is global
ibaAnalyzer features local and global macros:

- Local macros:
  The *Macro is global* option is disabled in the macro designer. These macros are saved in an analysis and are only available in that analysis.
- Global macros:
  The *Macro is global* option is enabled in the macro designer. These macros are saved on the computer and are available in all analyses that you open or create on that computer.

Description
Briefly describe the macro in this field. The description later appears in the expression builder.



Inputs
Enter the required parameters for the macro, see ↗ *Area for input variables, page 191*.

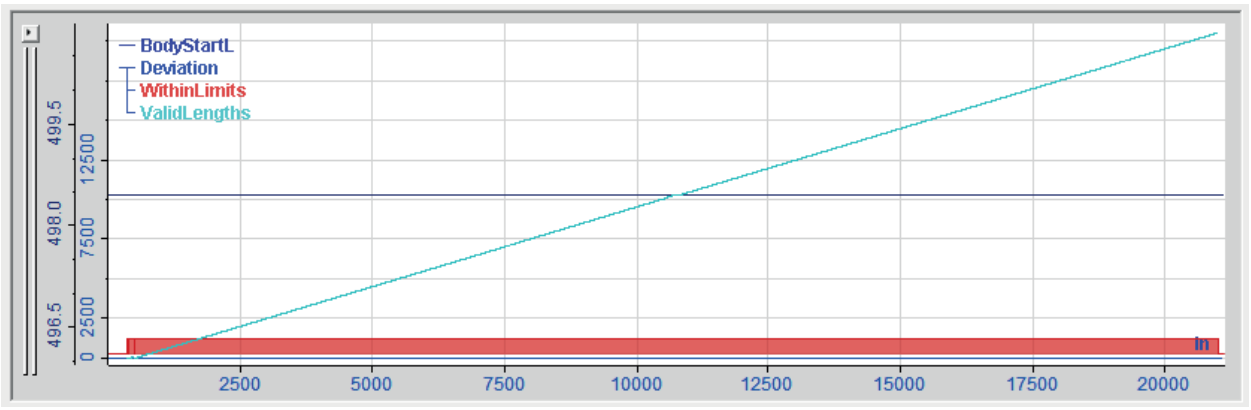
Intermediate values
Define expressions that serve as intermediate results and are available for further operations, see ↗ *Area for intermediate values, page 192*.

Result
Enter the function that produces the result of the macro. You can create the result function with the expressions from the expression builder or with other macros. If you want to use other macros, they must be valid. If you use intermediate results, put them in square brackets, e.g. "[intermediate value]". You can also use additional signals (other signals from the loaded data file, logical signals, results from database queries, etc.). However, the selected signals must always be present and valid, otherwise the macro cannot be executed.



Preview window

The preview window shows all valid input arguments, intermediate results, and the macro result itself. You can switch the preview of each signal on or off using the checkbox in the *Show* column.



The preview window has the same features as a graph in the recorder window.

You can change the X-axis mode using the triangle button on the graph header, see ↗ *X-axis modes (reference axes)*, page 129. By default, the X-axis is set as the time axis.

8.1.1 Area for input variables

In the *Input* area, you define all the input variables (arguments) that you need for the further macro design process.

Mandatory and optional arguments

You can create optional and mandatory arguments. Mandatory arguments are essential to the functioning of the macro. Optional arguments describe additional parameters for controlling the macro function.

Set the number of mandatory arguments in the selection field above the table. The first rows in the table are then reserved for the mandatory arguments, depending on the number of mandatory arguments you preset. Enter an expression for each mandatory argument in the *Example expression* column. You cannot specify default values for mandatory arguments.

Then enter the optional arguments. You can specify expressions and default values that can be used as fallback options if the function becomes invalid. You can also specify default values only.

Inputs: Number of mandatory arguments: 1

	Show	Name	Example expression	Default	Comment
1	☑	Deviation	TimeToLengthL([82:1], [67:3])	--	Actual deviation (in percent)
2	☐	Limit		1.75	Allowed deviation (in percent)
3	☐	ReqLength		1200	Required length (in length-units) that the deviation

Columns in the table

The area includes the following items:

Show
Select the respective line to show the result of the expression or the fixed value for optional arguments in the preview window.

Name
Enter a unique name for the argument. Do not use the same names for arguments or intermediate values.

Example expression
Enter an expression function for each argument. An entry is obligatory for mandatory arguments. You cannot link to other arguments or intermediate values in the expression builder.

Default
Enter a numerical value for optional arguments. If you reference the optional argument in the macro, this value is used.

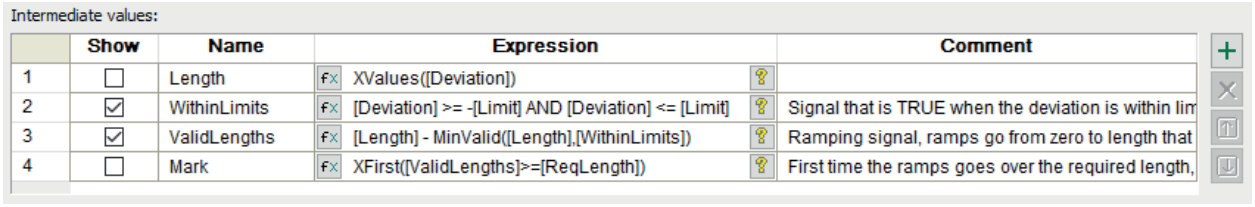
Comment
Briefly describe the argument.

Buttons

	Insert an empty row for a new argument
	Delete row and argument
	Move argument up in the table
	Move argument down in the table

8.1.2 Area for intermediate values

In this area, you can perform intermediate calculations or obtain partial results that you need for further macro design.



Note

Make sure that there are no contradictions between the individual intermediate values, and avoid circular references. Otherwise the macro cannot be executed correctly.

There is no check or warning in relation to circular references.

The area includes the following items:

Show
Select the respective line to show the result of the selected expression in the preview window. The following conditions must be met for this:

- The expression must be valid.
- If the intermediate value references mandatory arguments or other intermediate values, they must be valid.

Name
Enter a unique name for the argument. Do not use the same names for arguments or intermediate values.

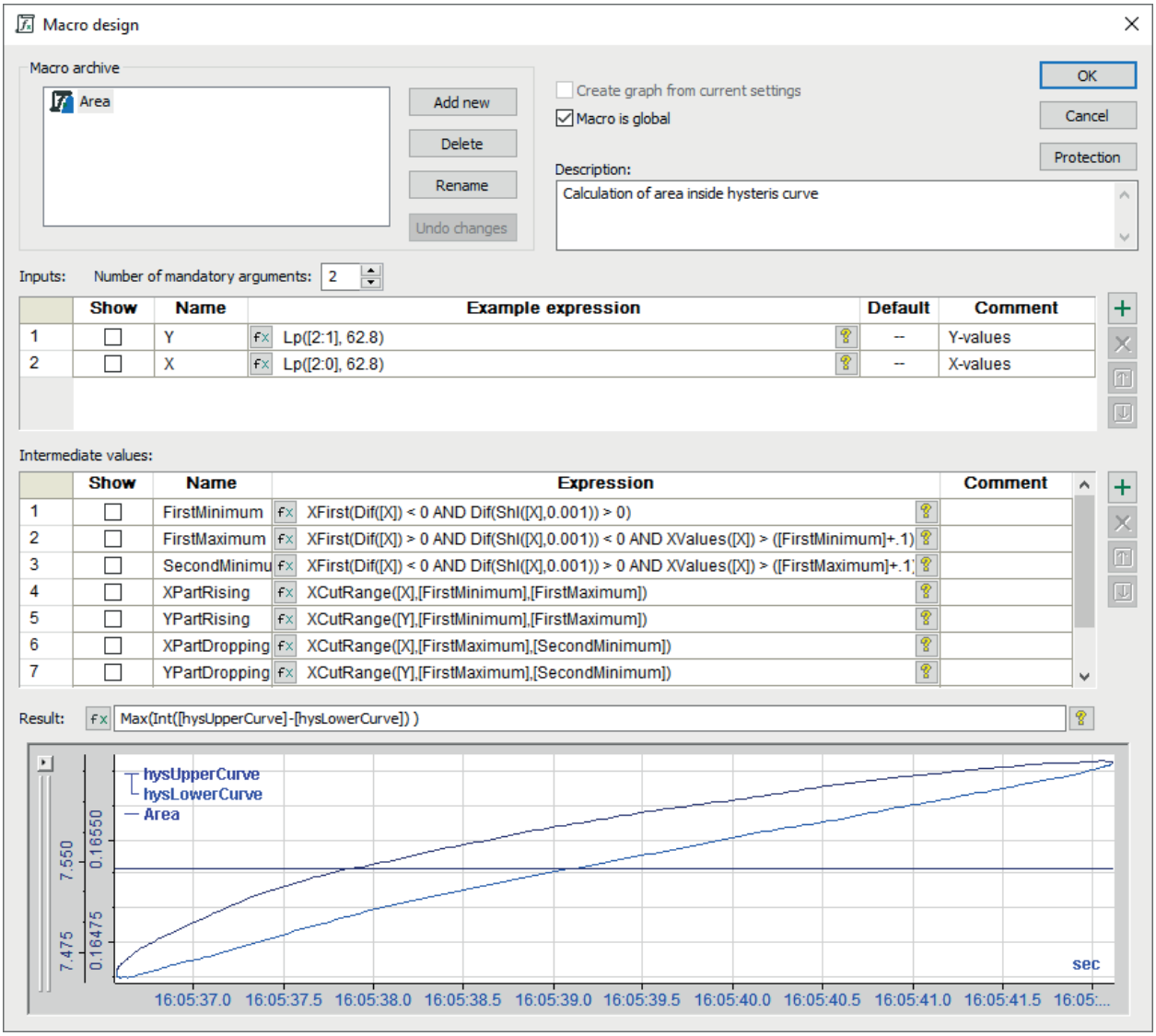
Expression
Enter the function with which you want to create the intermediate value. You can create the function with the expressions from the expression builder or with other macros. If you want to use other macros, they must be valid. If you use intermediate results, put them in square brackets, e.g. "[intermediate value]". You can also use additional signals (other signals from the loaded data file, logical signals, results from database queries, etc.). However, the selected signals must always be present and valid, otherwise the macro cannot be executed.

Comment
Briefly describe the intermediate value.

Buttons
The buttons on the right-hand side have the same functions as for the inputs, see ↗ *Area for input variables, page 191*.

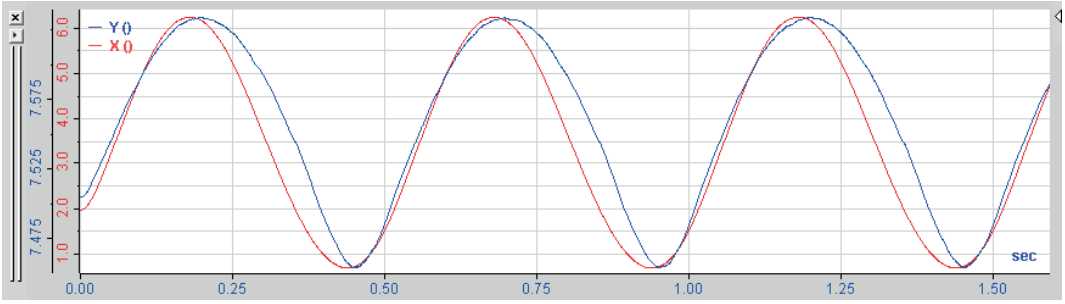
8.1.3 Example 1: Calculating the surface area within a hysteresis curve

This example shows the creation of a macro to calculate the surface area within a hysteresis curve (X-Y plot).



Procedure

1. Select the number of mandatory arguments, here "2".
2. Select the input signals (X-values and Y-values).
   In the example, these are 2 time-based signal trends of a position measurement ("Y") and a force measurement ("X").



3. Enter the intermediate values, see ↗ *Interim calculation, page 195*.
4. To determine the area between the curves, integrate the difference between the intermediate values *hysUpperCurve* and *hysLowerCurve*.

Interim calculation

Interim calculations, which you enter as intermediate values, are required to create the macro:

- FirstMinimum:
  Calculation of the signal point where the input signal reverses and starts falling again. This point is determined using the *XFirst* and *Diff* functions. The derivative changes from a negative to a positive value.
- FirstMaximum:
  Calculation of the signal point where the input signal reverses again, and starts rising again. This point is determined using the *XFirst* and *Diff* functions. The derivative changes from a positive to a negative value.
- SecondMinimum:
  Calculation of the signal point where the input signal reverse again. This point is determined as the signal point after *FirstMaximum* using the *XFirst* and *Diff* functions.
- XPartRising:
  The *XCutRange* function determines the range for the input signal between *FirstMinimum* and *FirstMaximum*.
- YPartRange:
  The *XCutRange* function determines the range for the output signal between *FirstMinimum* and *FirstMaximum*.
- XPartDropping:
  The *XCutRange* function calculates the range where the input signal falls between *FirstMaximum* and *SecondMinimum*.
- YPartDropping:
  The *XCutRange* function calculates the range where the output signal rises between *FirstMaximum* and *SecondMinimum*.
- hysLowerCurve:
  The lower curve of the hysteresis is determined using the *XY* function (here *YPartRising* and *XPartRising*).
- hysUpperCurve:
  The upper curve is determined using the *XY* function (here *YPartDropping* and *XPartDropping*). Usually, the *XY* function requires the X-argument (here *XPartDropping*) to always rise. Because this is not the case in the example, it is first corrected with the *XMirror* function for *XPartDropping* and *YPartDropping*.

8.1.4 Example 2: Calculating the head, fillet and tail of an aluminum strip

Rolled metal strips, or coils, are basically divided into three different sections: the head, fillet, and tail. The fillet is the largest part of the strip.

Preliminary consideration

The individual sections must be first be defined:

The head is the area where essential quality parameters (e.g. thickness, width, mechanical properties) are not yet homogeneous within permissible tolerances. The head is always at the start of the strip. This section is formed first during rolling.

The definition also applies to the tail, however this section is the last to leave the rolling stand. Reshaping is completed.

The fillet section is between the head and tail, meaning the key quality parameters are ideally homogeneous in this area.

The following two macros calculate the start and end of the fillet section.

Macro to calculate the start of the fillet: BodyStartL

The following input signals are defined:

- Deviation:
  This quality parameter (here the thickness deviation) must be within a specified tolerance. The signal is length-based using the *TimeToLengthL* function. The deviation is given as a percentage.
- Limit:
  Tolerance limit as a constant value. This optional argument has a preset percentage value (here 1.75%).
- ReqLength:
  Required length, where the thickness deviation must be within the specified tolerance. This is also an optional argument with a preset value (here 1.200).

Interim calculations are required for generating the macro:

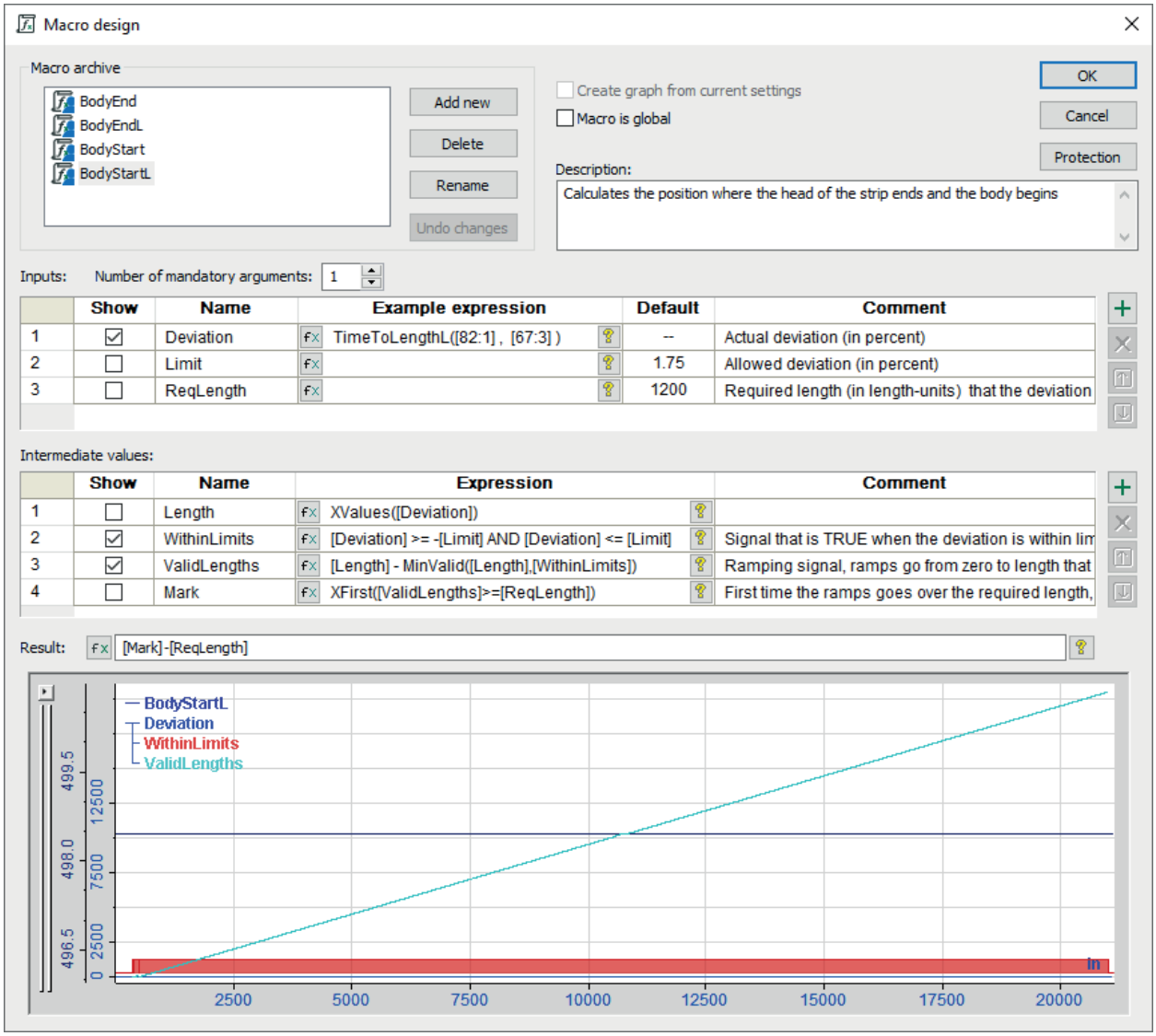
- Length:
  The *XValues* function determines the length-based signal points of the thickness deviation (*Deviation*).
- WithinLimits:
  This expression determines whether the thickness deviation (*Deviation*) is within the tolerance. The expression is TRUE if: *-Limit ≤ Deviation ≤ Limit.*
- ValidLengths:
  The *MinValid* function determines the smallest value of *Length* that fulfills the *WithinLimits* condition (where the tolerance conditions are met for the first time).

  This value is then subtracted from *Length*. *ValidLengths* thus receives the length-based signal points, which are less than 0 before the *WithinLimits* range, and so in the invalid range. *ValidLengths* therefore starts at y=0.

- Mark:
  The *XFirst* function determines the first value of *ValidLengths* that is greater than or equal to *ReqLength*. This is therefore the first length in the valid range that corresponds to or exceeds the required length.

The difference between *Mark* and *ReqLength* is then the result of the calculation, and the macro (the start of the fillet section).

To generate a preview, you must use the triangle button to change the X-axis mode to the length axis.



Macro to calculate the end of the fillet: BodyEndL

The input signals have the same parameters as selected in the previous example.

Interim calculations are required for generating the macro:

- Length:
  Same expression as in the previous example
- WithinLimits:
  Same expression as in the previous example

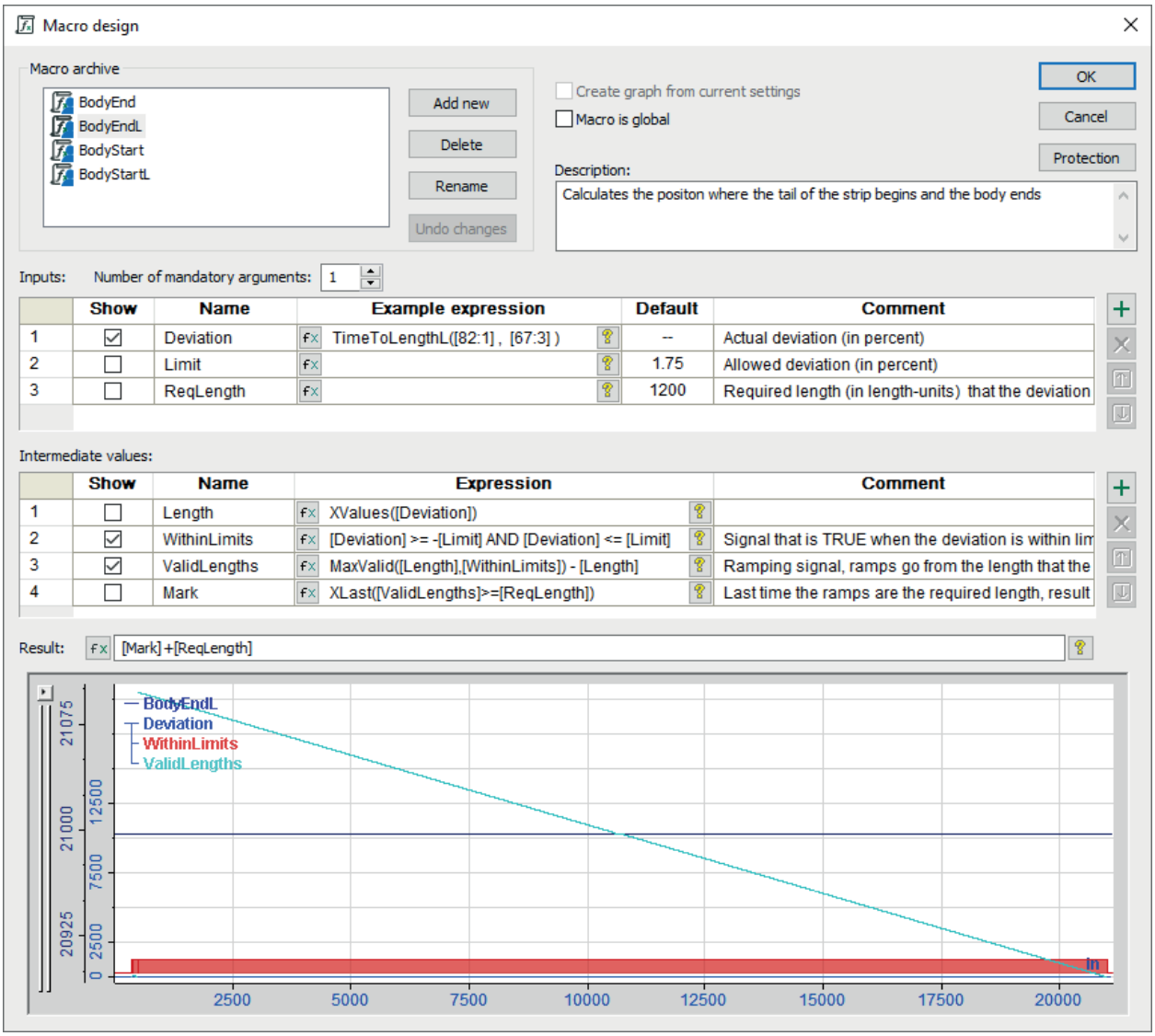
- ValidLengths:
  The *MaxValid* function determines the largest value of *Length* that fulfills the *WithinLimits* condition (where the tolerance conditions are met for the last time).

  This value is then subtracted from *Length*. *ValidLengths* thus receives the length-based signal points, which are less than 0 after the *WithinLimits* range, and so in the invalid range.

- Mark:
  The *XFirst* function determines the first value of *ValidLengths* that is greater than or equal to *ReqLength*. This is therefore the first length in the valid range that corresponds to or exceeds the required length.

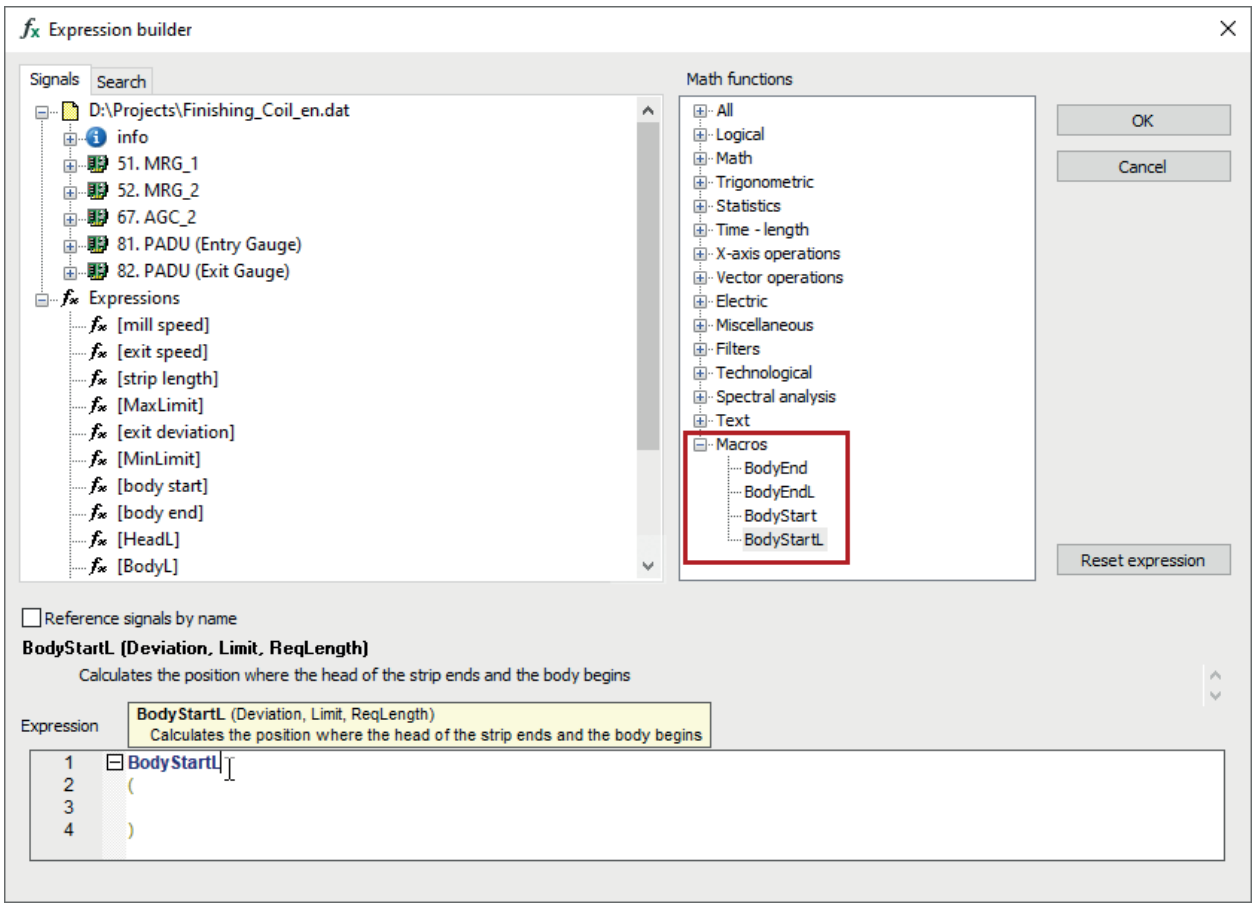
The sum of *Mark* and *ReqLength* is then the result of the calculation, and the macro (the end of fillet section).

To generate a preview, you must use the triangle button to change the X-axis mode to the length axis.



8.2 Applying macros in the expression builder

All macros created are available in the expression builder. You can use them there like conventional functions. The macros appear in the function tree in the *Macros* node.



As with other functions, you see a tooltip text when you enter the macro name or click on the macro name in the command line. The description that is shown in the expression builder is the description that you entered in the macro designer.

8.3 Export and import of macros

Macros are part of the analysis (PDO file) and are also available in the Expression builder. You can also export and import macros.

When exporting, *ibaAnalyzer* saves the macros in an MCR file.

ibaAnalyzer features local and global macros:

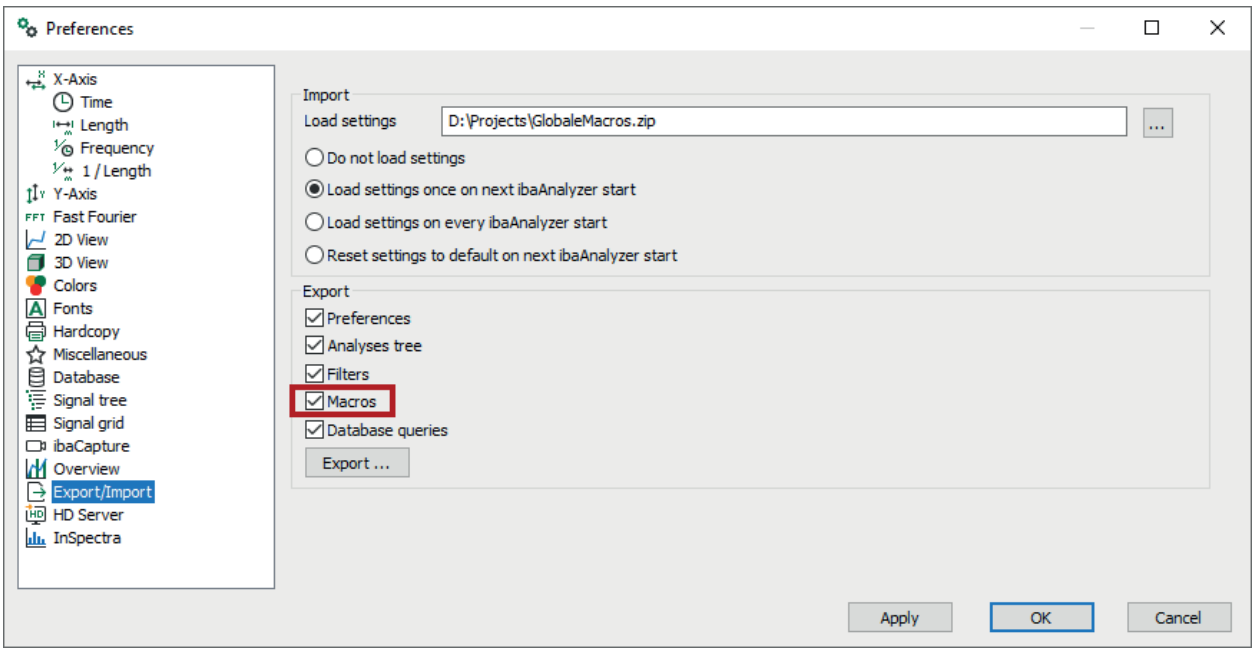
- **Local macros:**
  These macros are saved in an analysis and are only available in that analysis. The *Macro is global* option is disabled in the macro designer.
  You can export and import local macros via the context menu in the macro designer, see ↗ *Exporting and importing local macros, page 201*.
- **Global macros:**
  These macros are saved on the computer and are available in every analysis that you open or create on that computer. The *Macro is global* option is enabled in the macro designer.
  You can export and import global macros in the Preferences, see ↗ *Exporting and importing global macros, page 200*.

8.3.1 Exporting and importing global macros

You can export global macros in the *Export/Import* node in Preferences. This function exports various items and settings to a ZIP file. You only need to export and import global macros in order to use them on other computers.

For more information, see ↗ *Export/import settings*, page 93.

In addition, each global macro is stored as an MCR file in the following directory:
`C:\Users\[user]\AppData\Roaming\iba\ibaAnalyzer`
You can also import these macro files directly into an analysis or macro archive, see ↗ *Exporting and importing local macros, page 201*.



Exporting macros via Preferences

1. Open the *Export/Import* node in Preferences.
2. To include macros in the export, select the *Macros* option under *Export*.
3. Confirm with <Apply>.
4. Click on <Export>.
5. Enter a path and a file name.

→ *ibaAnalyzer* exports the selected settings as a ZIP file to the directory.

Importing macros via Preferences

After importing the file, macros contained in an export file are available as global macros.

1. Open the *Export/Import* node in Preferences.
2. In the *Import* area, select a path and a ZIP file with *ibaAnalyzer* settings using the browser button <...>.
3. Select the second or third option to load the settings on the next startup or on every startup of *ibaAnalyzer*.

4. Confirm your settings with <OK>.
5. Restart *ibaAnalyzer*.

→ The global macros and the other settings from the ZIP file are available in *ibaAnalyzer*.

8.3.2 Exporting and importing local macros

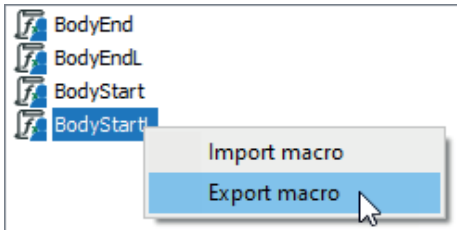
Local macros are initially only available in the analysis in which they were defined.

Individual macros can be specifically exported from the macro archive or imported into it. In this way, you can swap macros with other users or between different computers.

In principle, you can also import or export global macros in the macro archive. However, they lose the global status and are marked as local macros when imported into another analysis.

Exporting local macros

1. Select a macro in the macro archive of the macro designer.
2. Right-click to open the context menu of the macro.



3. Select *Export macro*.
4. Enter a path and a file name and confirm with <Save>.

→ *ibaAnalyzer* exports the selected macro as an MCR file to the directory.

Importing local macros

1. Open the context menu in the macro archive of the macro designer.
2. Select a path and an MCR file.

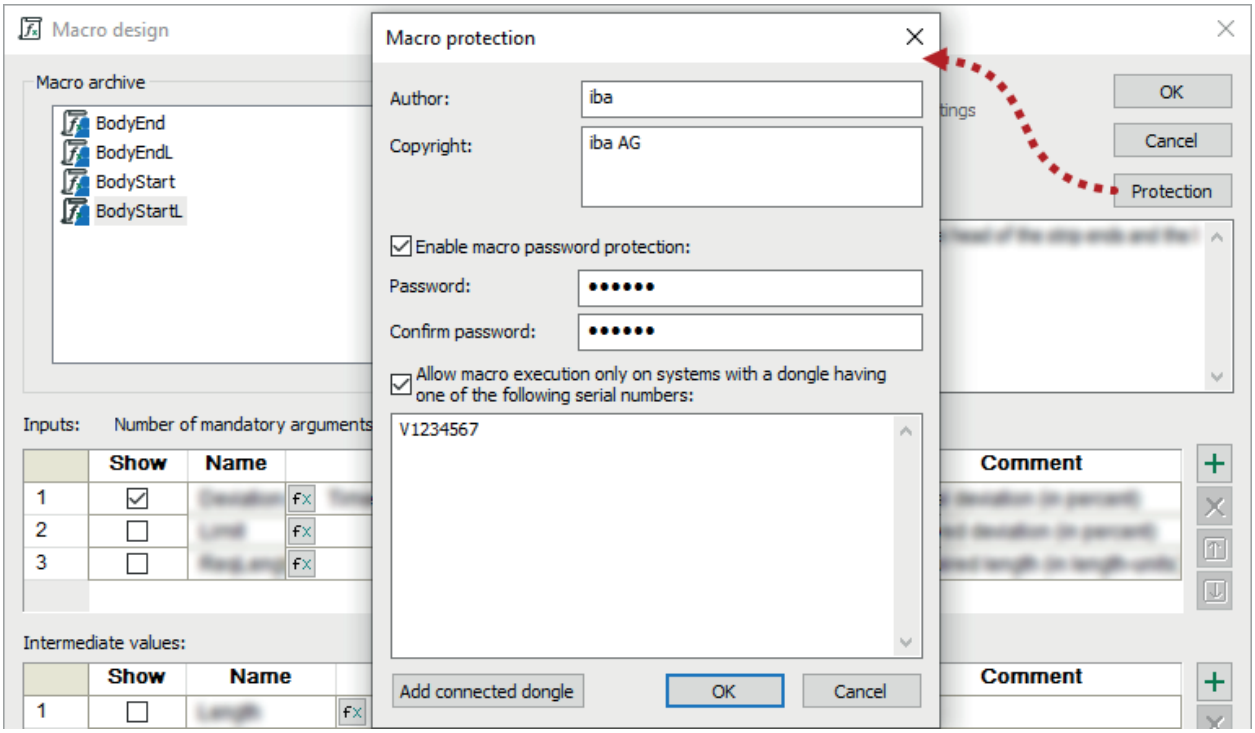
→ *ibaAnalyzer* imports the selected MCR file as a macro into the macro archive. You can use or edit the new macro in the same way as the other macros.

8.4 Password protection for macros

You can protect a macro against unauthorized or accidental changes by assigning a password. You can also make execution of the macro dependent on certain MARX dongle numbers.

Password assignment

Click on the <Protection> button in the macro designer to open the password protection dialog.



Enter the following in the dialog:

Author
Enter the name of the macro's author here.

Copyright
If required, enter copyright or other information, such as the author's contact details.

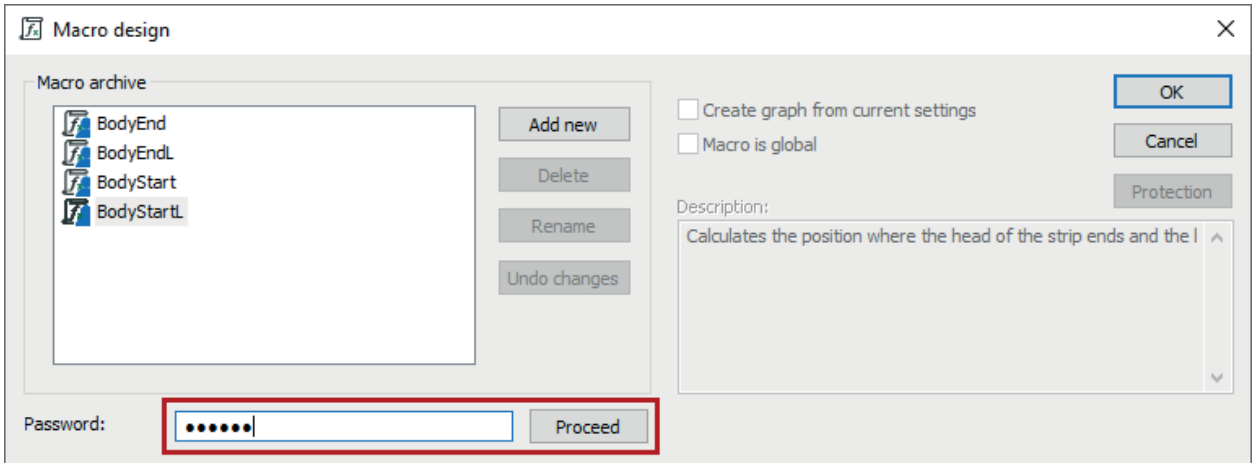
Enable macro password protection
After enabling this option, enter a password and confirm it.
Once a password has been enabled (by exiting the dialog with <OK>), the macro is protected against changes the next time the macro editor is opened. The calculations are then hidden until you enter the correct password.

Allow macro execution only on systems ...
If you enable this option, you can restrict the ability to execute a macro by entering MARX dongle numbers in the field below. You can then only run the macro on systems that have a dongle with a listed number.

<Add connected dongle>
Use this button to include the license of the MARX dongle currently connected to the computer you are using.

Opening protected macros

If you want to open a protected macro in the macro designer, enter the correct password in the *Password* field and click <Proceed>.



Removing password protection of macros

To cancel macro protection, open the protected macro and uncheck the *Enable macro password protection* option in the *Macro protection* dialog.

9 Filter editor

Filters are key tools in signal analysis, as they isolate specific parts of a signal, minimize interference, and improve the quality of data for analysis. Filters highlight relevant signal parts and suppress unwanted noise components. These functions not only improve the signal quality but also increase the efficiency of processing and facilitate the extraction of meaningful characteristic data.

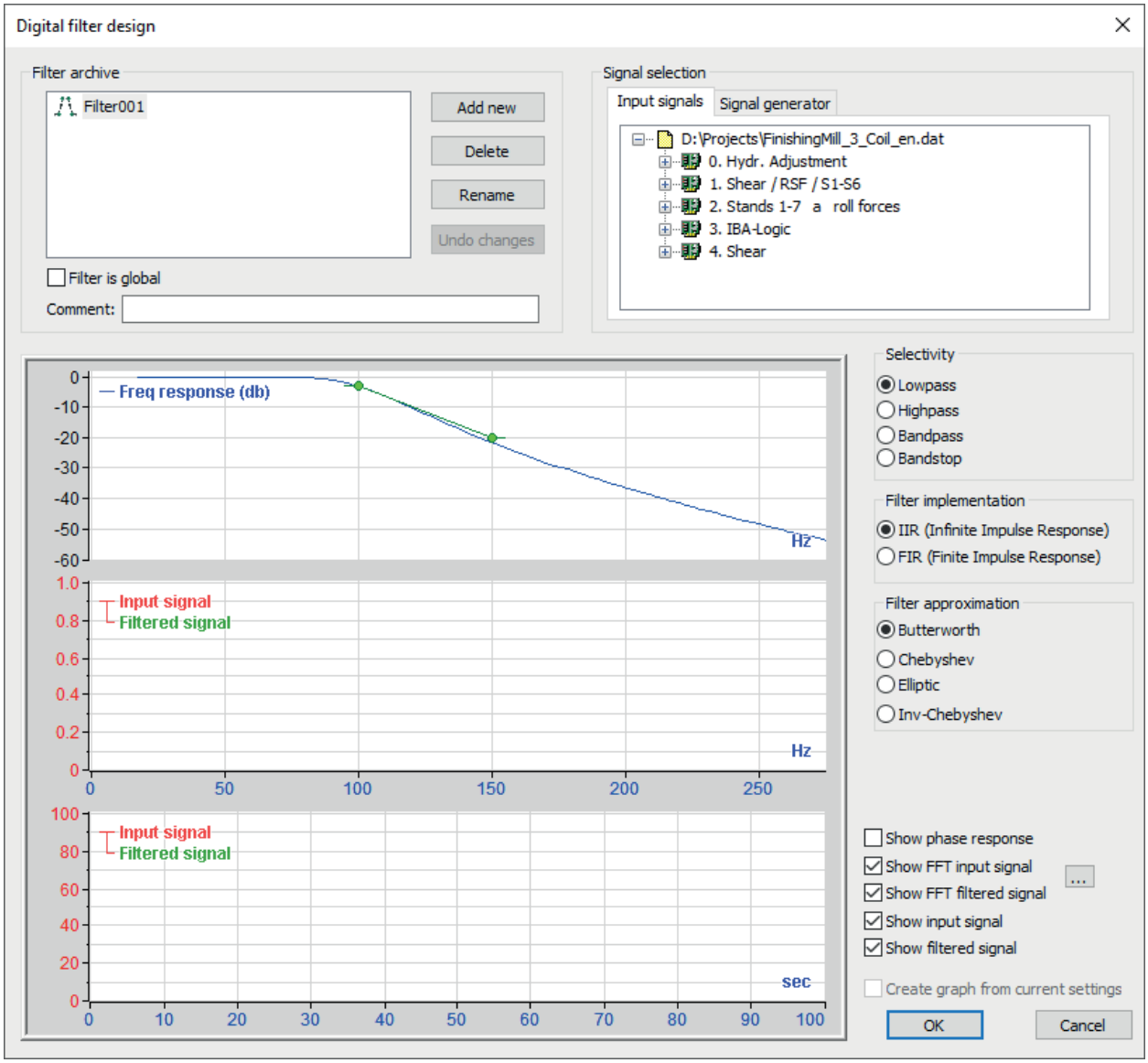
The filter editor offers versatile solutions for processing signals for various applications and achieving precise results.

9.1 Filter editor dialog window

Use the filter editor to create and edit the filters. You can open the filter editor from the toolbar or via the *Setup* menu.



The filter editor looks like this:



Filter archive
The filter archive lists all existing filters for the selected analysis. You can create new filters, or change, delete or rename existing filters. You can export and import the filters, see ↗ *Export and import of filters, page 213*.

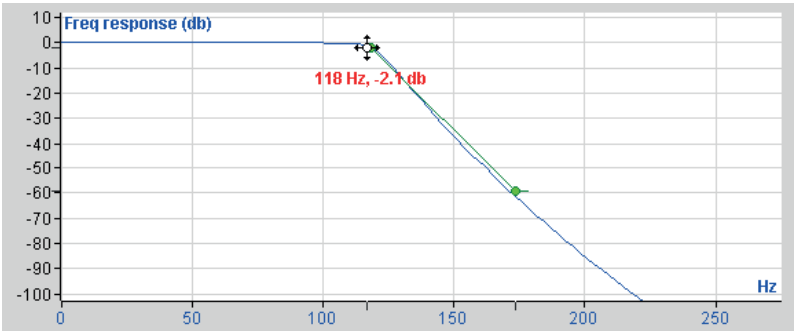
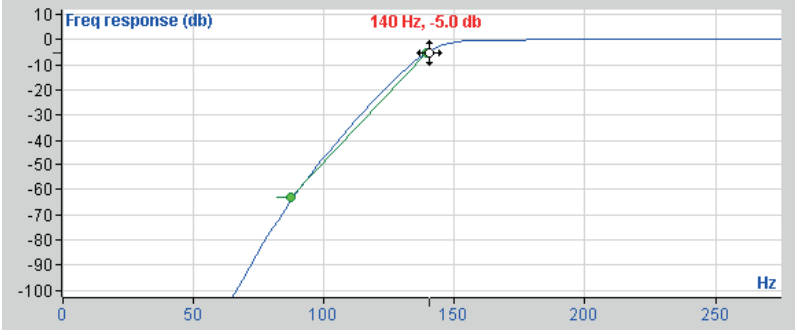
Filter is global
ibaAnalyzer features local and global filters:

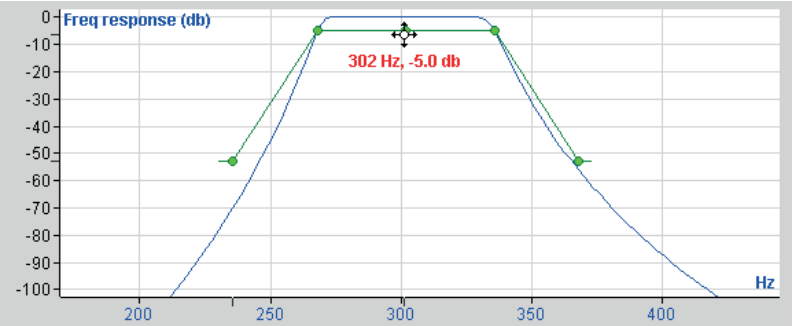
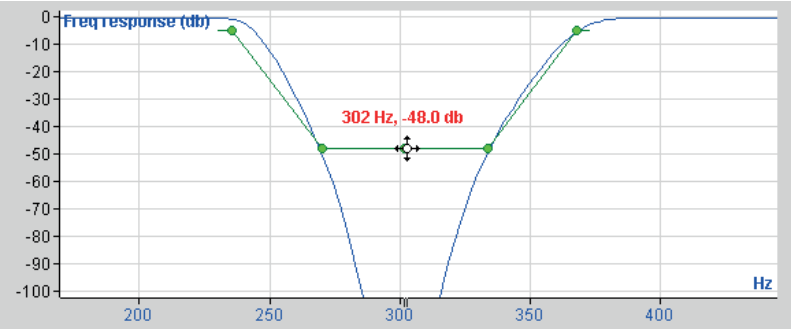
- Local filters:
  The *Filter is global* option is disabled in the filter editor. These filters are saved in an analysis and are only available in that analysis.
- Global filters:
  The *Filter is global* option is enabled in the filter editor. These filters are saved on the computer and are available in all analyses that you open or create on that computer.

Comment
Enter a brief description of the filter. The description later appears in the expression builder.

Signal selection
Select the signal for which you want to test the filter. You can also generate a signal. For more information, see ↗ *Signal selection, page 207*.

Selectivity
Select the type of filter. You have the following options.

Filter type and description	Example
Low pass Allows low frequencies to pass, and filters out high frequencies.	
High pass Filters out low frequencies, and allows high frequencies to pass.	

Filter type and description	Example
Bandpass Allows frequencies within the set frequency range to pass, and filters out the lower and higher frequencies.	
Bandstop Filters out the frequency components within the specified frequency range, and allows lower and higher frequencies to pass.	

Filter implementation
Select a filter implementation matching your application.

- *IIR (Infinite Impulse Response)*:
  IIR filters are often the preferred option in practical use because they calculate faster and require less memory.
- *FIR (Finite Impulse Response)*:
  FIR filters offer more control over the phase and amplitude shape than IIR.

Filter approximation
Select a filter characteristic (approximation) matching your application. The filter approximations differ in their calculation methods.

Curve field
You can customize the view for the curves.

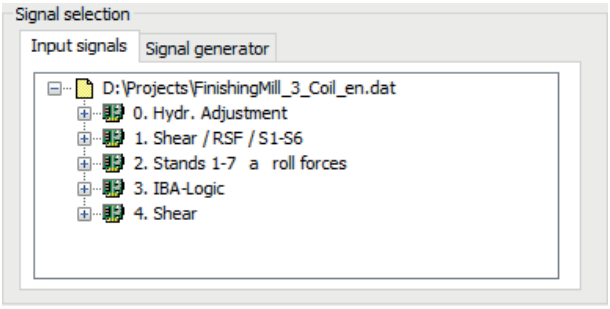
- *Show phase response*: Phase shift or phase response of the filter in degrees (deg)
- *Show FFT input signal*: FFT of the input signal as a red curve
  You can adjust the FFT settings using the <...> button.
- *Show FFT filtered signal*: FFT of the filtered signal as a green curve in the same graph as the FFT input signal
- *Show input signal*: Selected or generated input signal as a red curve
- *Show filtered signal*: Filtered input signal as a green curve in the same graph as the input signal
- *Create graph from current settings*: If you enable this option, a new graph is created in the *ibaAnalyzer* recorder window with the filtered signal when you exit the filter editor by clicking <OK>. Use this function to include filtered signals in the analysis.

For more information, see ↗ *Editing the curve field and filters, page 208*.

9.1.1 Signal selection

Select the signal for which you want to test the filter. You can also generate a signal.

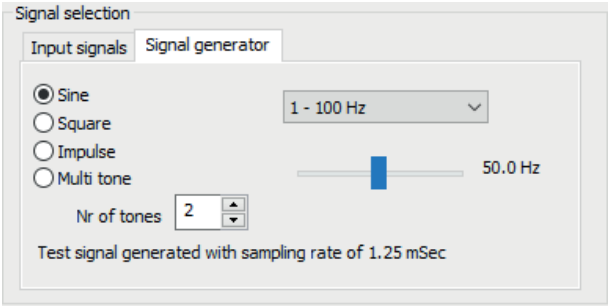
Input signals tab
When a data file is loaded, this tab displays the input signals in the signal tree.



You can use the filters for the following signal sources:

- Input signals, i.e. all signals in a data file
- Virtual signals created using the expression builder

Signal generator tab
The signal generator can generate a range of test signals, such as sine, square, pulses and multi-tones (frequency combinations). You can set the signal shape and the frequency range.



Multi-tone settings

The multi-tone signal is made up of individual base frequencies (tones). A multi-tone with one tone corresponds to a sine tone. A multi-tone with two tones corresponds to the combination of two sine signals, one base frequency (lower frequency) and a second sine with a higher frequency.

If you select more than one tone (*Nr. of tones* > 1), a second slider control is displayed. Use the upper slider to set the base frequency and the lower slider to set the higher frequency.
The higher frequencies result from the set bandwidth (upper frequency minus base frequency) divided by the number of tones. Each additional tone is added with half the amplitude of its predecessor. The lower slider cannot have a lower frequency than the upper slider.

9.1.2 Editing the curve field and filters

The dialog window of the filter editor shows two display graphs by default: Frequency response (db) and the phase response (deg). If you have selected other display graphs when last editing the filters, these graphs appear the next time when opening the filter editor.

These graphs behave in exactly the same way as the graphs in the recorder window of *ibaAnalyzer*: You can adjust the scales of the axes or open a zoom window as in the recorder window. You can reset the zoom or use auto-scaling via the context menu of the graph.

Setting the filter

You set the filter in the Frequency response graph.

Drag the filter to the desired position using the green dots. The mouse pointer turns into a compass symbol and the cut-off frequency and attenuation of the respective position are displayed.

Procedure for low-pass and high-pass filters

1. Move the upper point to the desired cut-off frequency.
2. Move the lower point so that the attenuation reaches the desired steepness and strength.

A steep connecting line between the two points attenuates frequencies that deviate slightly from the cut-off frequency. With flat connecting line, the filter works more smoothly.

Procedure for bandpass and bandstop filters

1. To set the desired frequency band and cut-off frequencies, move one of the points horizontally that are directly next to the center point. You only need to move one of the points because they behave symmetrically.
2. To position the frequency band at the frequency to be filtered, use the center point to move the entire frequency band along the frequency axis.
3. Move the outer points to determine the attenuation of the unwanted frequencies. You only need to move one of the points because they behave symmetrically.

Manually setting the filter

You can enter the exact frequencies of the respective filter points manually using the *Manual filter setup* command in the context menu in the Frequency response graph.

9.1.3 Example: Create a bandstop filter for 50 Hz

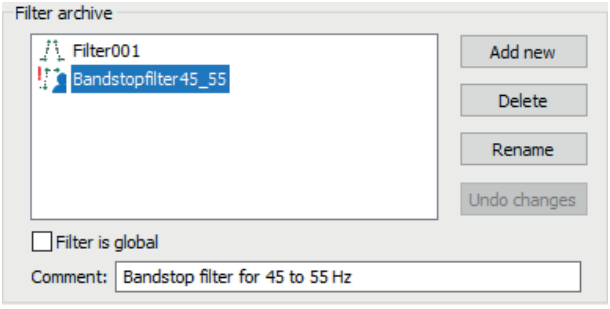
There are many ways to design a filter with the filter editor. The method depends on the application and on your knowledge about filter techniques. The following example is intended to help you familiarize yourself with the editor and gather practical experience.

The test signal is generated by the integrated signal generator. Alternatively, you can also use a clean input signal from the data file or an artificially generated signal from the expression builder for your example.

Which view you choose for the graphs depends on the requirements. Especially when filtering out frequencies from real signals, it is recommended to turn off the phase response display, as it is rarely needed in practice. Instead, enable both FFT displays and both signal displays.

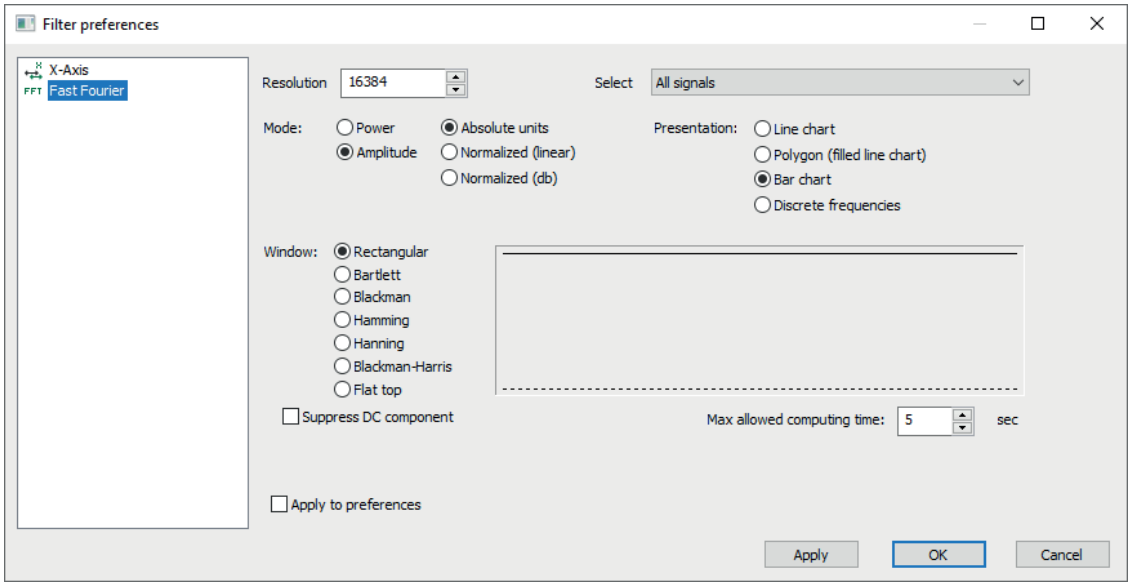
Procedure

1. Create a new filter using the <Add> button in the *Filter archive* area.
2. Use the <Rename> button to enter a descriptive name for the filter, for example "Bandstop45_55".
3. Enter a comment to briefly describe the filter.



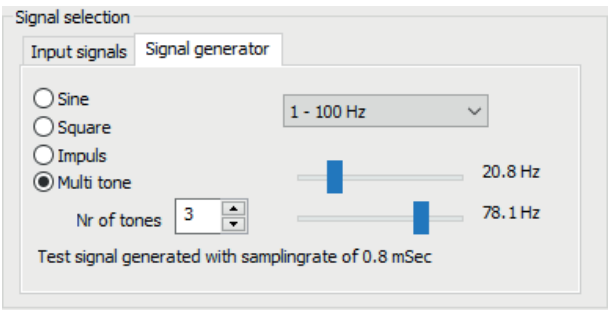
4. Turn on the *FFT input signal* and *Input signal* graphs.
   You can use these displays to check the shape of the input signal generated by the signal generator.
5. Customize the FFT display using the <...> button.

→ The Filter preferences dialog for the FFT view opens up.

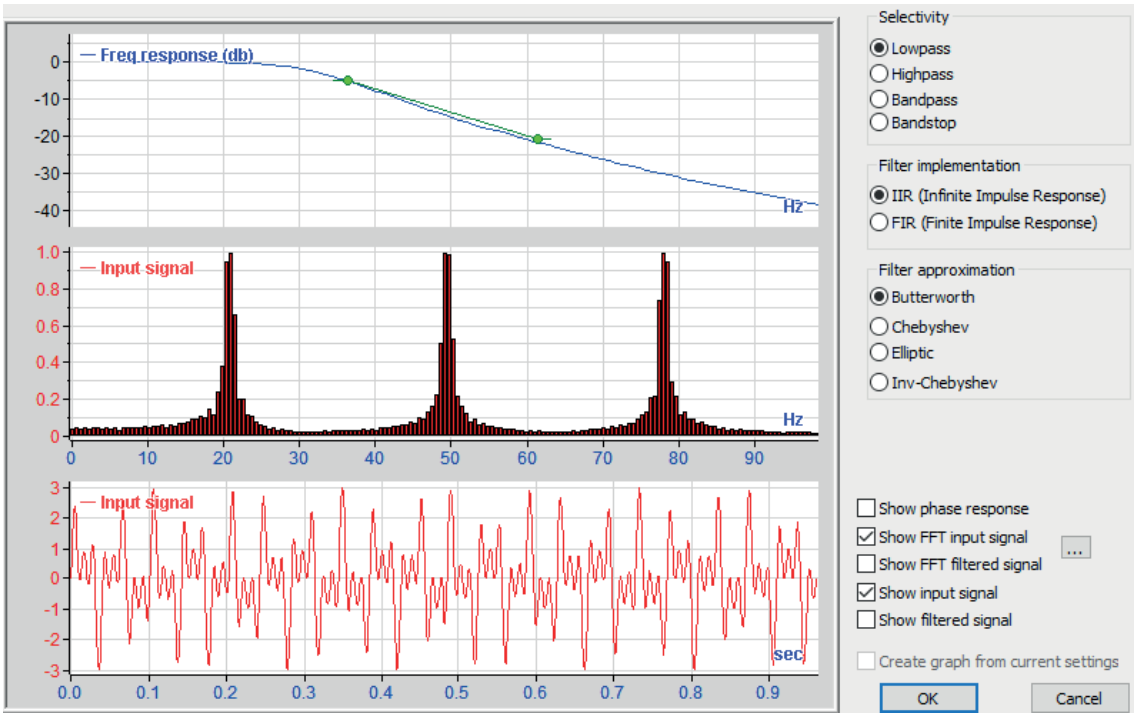


6. To generate a test signal, open the *Signal generator* tab in the *Signal selection* area.
7. Generate a test signal with multiple frequencies, one of which is approximately 50 Hz. You have many options for doing this.

   The following settings are used in the example:



Move the slider in the signal generator so that the amplitude of the input signal in the FFT view increases at 50 Hz.
If you cannot see the frequencies, or parts are missing, use the *Autoscale frequency axis in signal range* command in the context menu of the graph.



The test signal contains the frequencies 20 Hz, 50 Hz and 78 Hz. The bottom graph shows the progression of the signal over time.

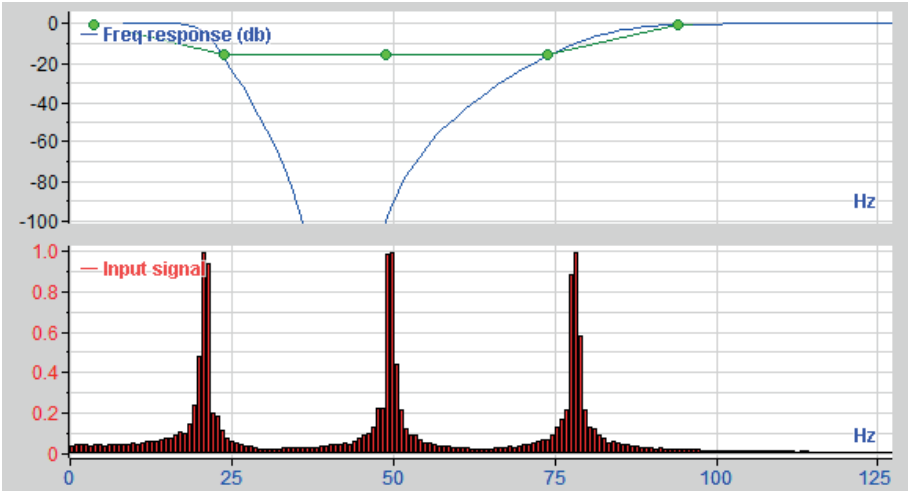
8. Select the *Bandstop* filter type.

   If the green dots of the filter are not completely visible in the frequency response graph, use the *Autoscale frequency axis in filter range* command in the context menu of the graph.

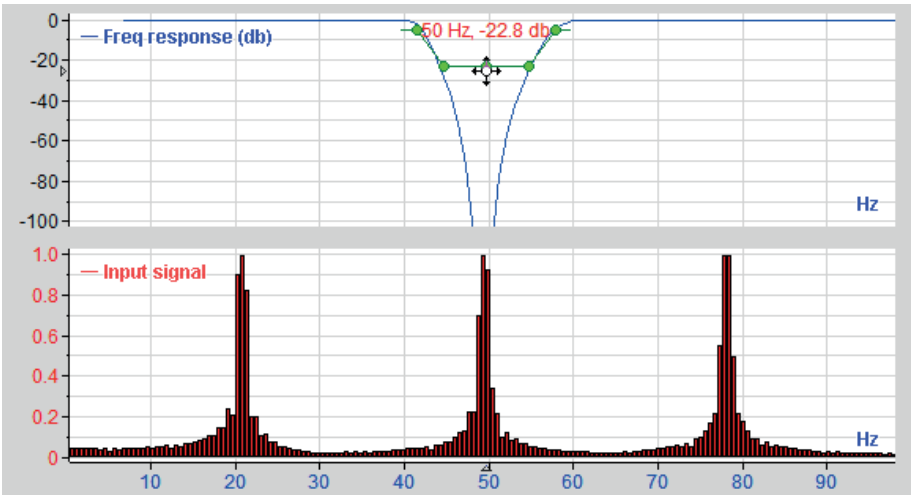
9. Slide the outermost dots of the frequency band together to create a compact and easily movable characteristic.

10. Shift the filter dots next to the center point toward the frequency amplitudes of the input signal.

    To provide a better resolution of the area of interest, use the *Autoscale frequency axis in filter range* command in the context menu of the graph.



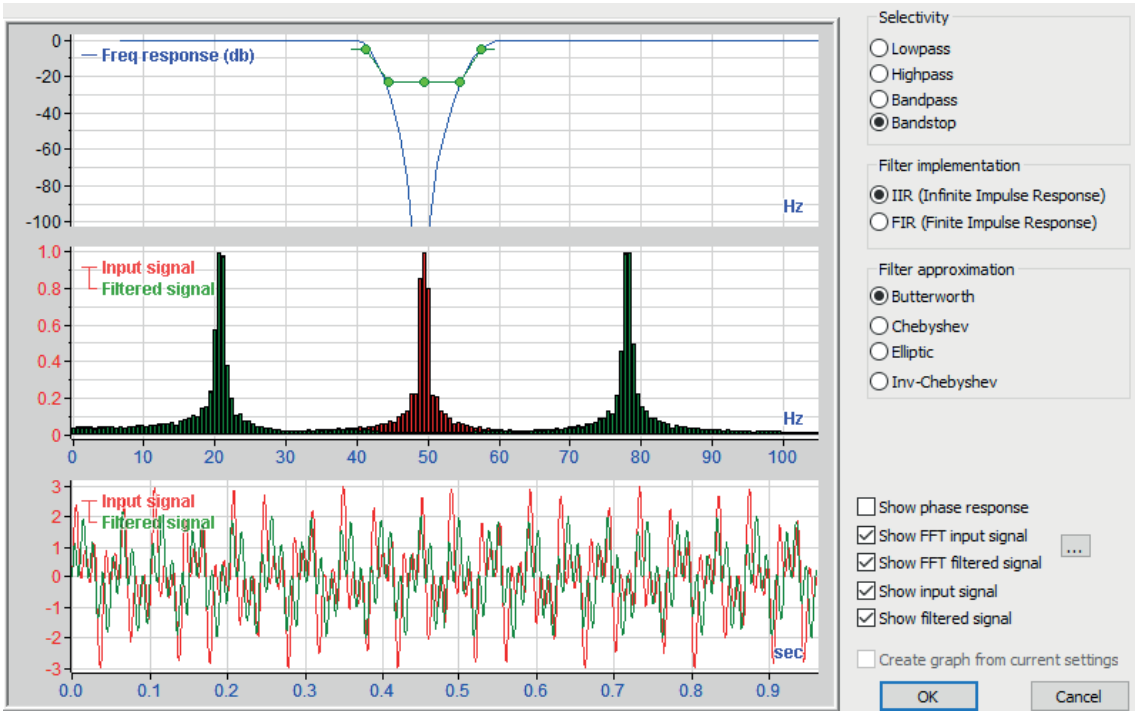
11. To set the corner frequencies, first move the center point to the 50 Hz position.
    Then place the dots next to the center point so that the corner frequencies are 45 Hz and 55 Hz.



12. Then turn on the FFT display for the filtered signal, and use the same FFT settings as for the other FFT view (see step 5).

→ The frequencies around 50 Hz are missing in the filtered signal.

→ You can easily change the filter behavior using the two outer green dots (attenuation).

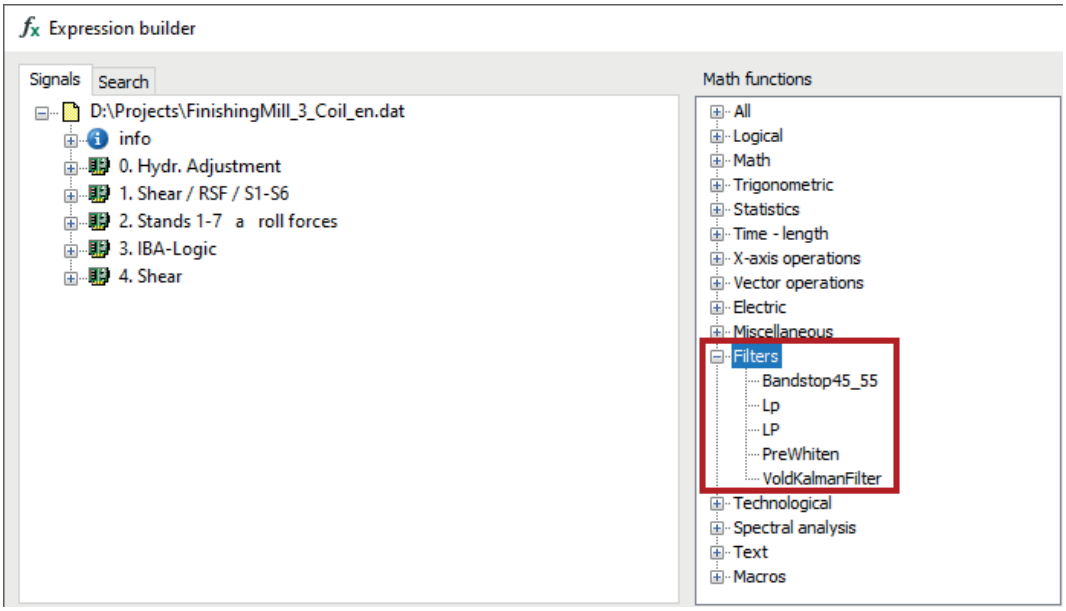


13. To save the filter for the current session, exit the dialog with <OK>.

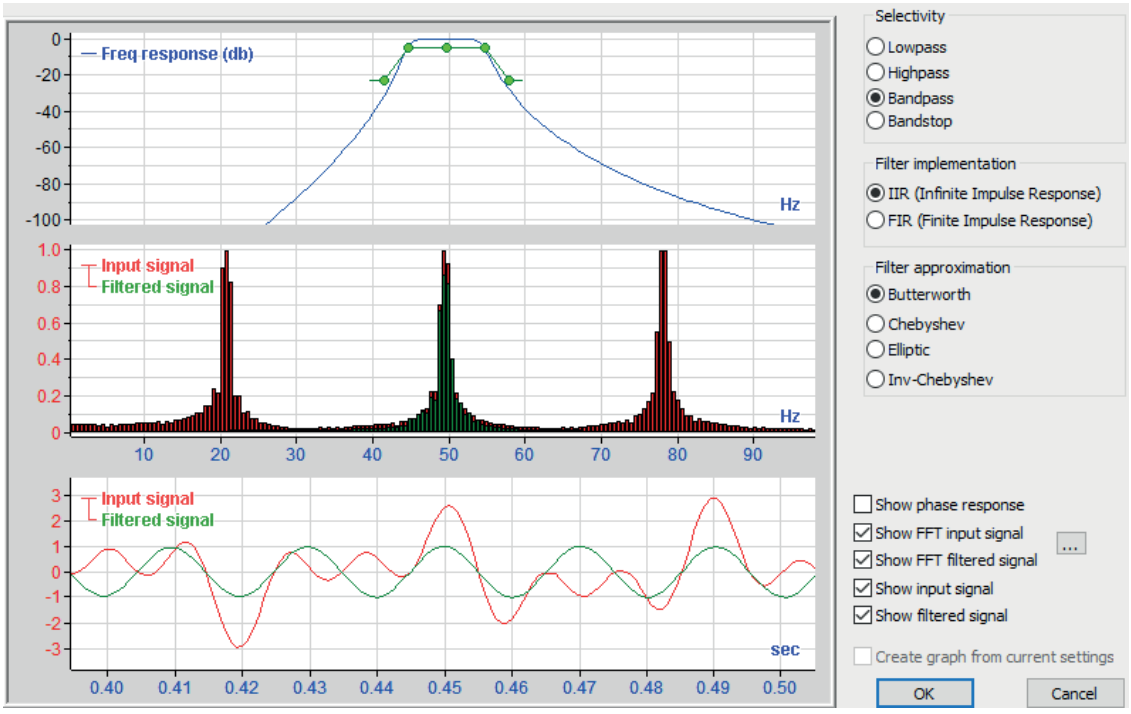
    To integrate the filter into the analysis, also save the analysis afterward.

    To make the filter available in all analyses with *ibaAnalyzer* on this computer, select the *Filter is global* option in the filter editor and exit the dialog with <OK>.

→ After saving, the filter is available in the Expression builder. You can use the filter to filter out 50 Hz oscillations from any signal.



→ If you switch the filter type, you can try out and create other filters. For example, if you switch to a bandpass filter, the frequency band is retained, and the frequencies outside 45 Hz to 55 Hz are filtered out.



9.2 Export and import of filters

Filters are part of the analysis (PDO file) and are also available in the Expression builder. You can also export and import filters.

When exporting, *ibaAnalyzer* saves the filters in a FIL file.

ibaAnalyzer features local and global filters:

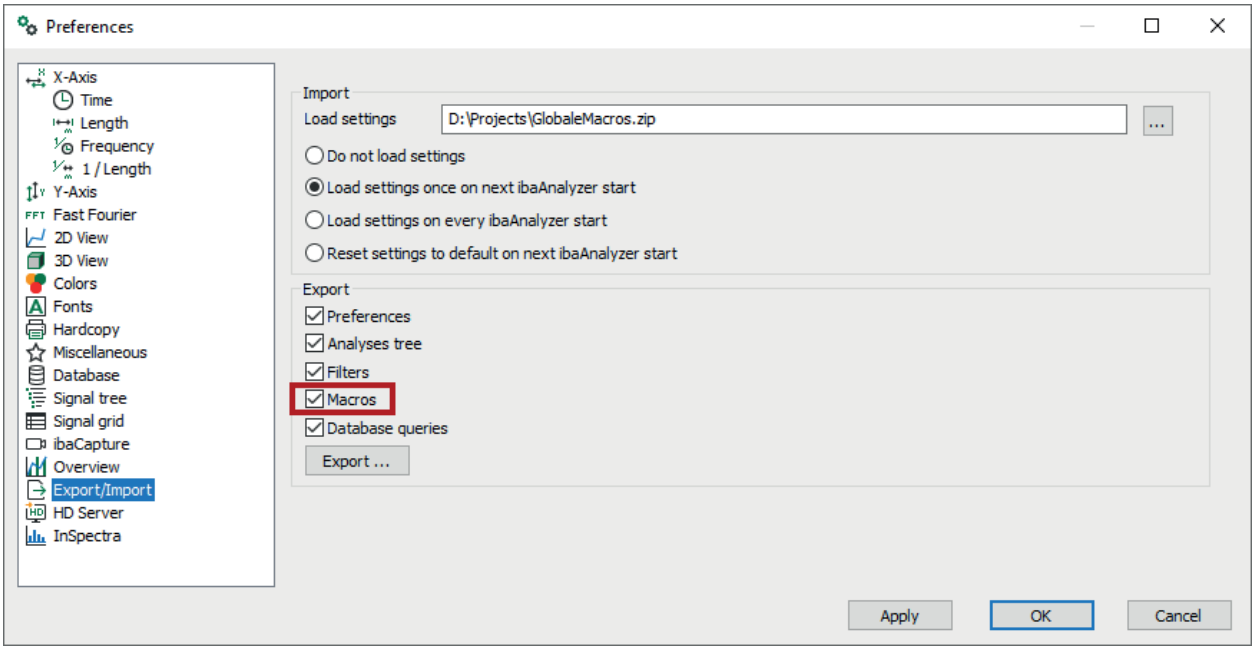
- **Local filters:**
  The *Filter is global* option is disabled in the filter editor. These filters are saved in an analysis and are only available in that analysis.
  You can export and import local filters via the context menu in the filter editor, see ↗ *Exporting and importing local filters, page 214*.
- **Global filters:**
  The *Filter is global* option is enabled in the filter editor. These filters are saved on the computer and are available in every analysis that you open or create on that computer.
  You can export and import global filters in the Preferences, see ↗ *Exporting and importing global filters, page 213*.

9.2.1 Exporting and importing global filters

You can export global filters in the *Export/Import* node in Preferences. This function exports various items and settings to a ZIP file. You only need to export and import global filters in order to use them on other computers.

For more information, see ↗ *Export/import settings*, page 93.

In addition, each global filter is stored as a FIL file in the following directory:
`C:\Users\[user]\AppData\Roaming\iba\ibaAnalyzer`
You can also import these filter files directly into an analysis or filter archive, see ↗ *Exporting and importing local filters, page 214*.



Exporting filters via Preferences

1. Open the *Export/Import* node in Preferences.
2. To include filters in the export, select the *Filters* option under *Export*.
3. Confirm with <Apply>.
4. Click on <Export>.
5. Enter a path and a file name.

→ *ibaAnalyzer* exports the selected settings as a ZIP file to the directory.

Importing filters via Preferences

After importing the file, filters contained in an export file are available as global filters.

1. Open the *Export/Import* node in Preferences.
2. In the *Import* area, select a path and a ZIP file with *ibaAnalyzer* settings using the browser button <...>.
3. Select the second or third option to load the settings on the next startup or on every startup of *ibaAnalyzer*.
4. Confirm your settings with <OK>.
5. Restart *ibaAnalyzer*.

→ The global filters and the other settings from the ZIP file are available in *ibaAnalyzer*.

9.2.2 Exporting and importing local filters

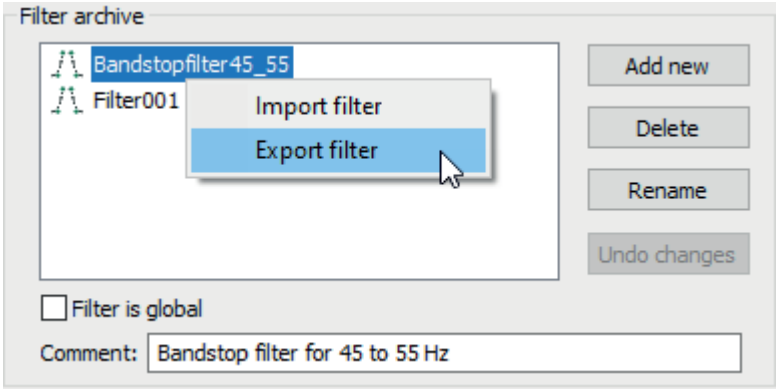
Local filters are initially only available in the analysis in which they were defined.

Individual filters can be specifically exported from the filter archive or imported into it. In this way, you can swap filters with other users or between different computers.

In principle, you can also import or export global filters in the filter archive. However, they lose the global status and are marked as local filters when imported into another analysis.

Exporting local filters

1. Select a filter in the filter archive of the filter editor.
2. Right-click to open the context menu of the filter.



3. Select *Export filter*.

4. Enter a path and a file name and confirm with <Save>.

→ *ibaAnalyzer* exports the selected filter as a FIL file to the directory.

Importing local filters

1. Open the context menu in the filter archive of the filter editor.
2. Select *Import filter*.
3. Select a path and a FIL file.

→ *ibaAnalyzer* imports the selected FIL file as a filter into the filter archive. You can use or edit the new filter in the same way as the other filters.

10 Text signals

In *ibaPDA*, you can configure text signals, which are saved in the data file. In *ibaAnalyzer*, you can display these text signals. Depending on the configuration of the data storage in *ibaPDA*, text signals in the signal tree or in the info area of the data file (info fields).

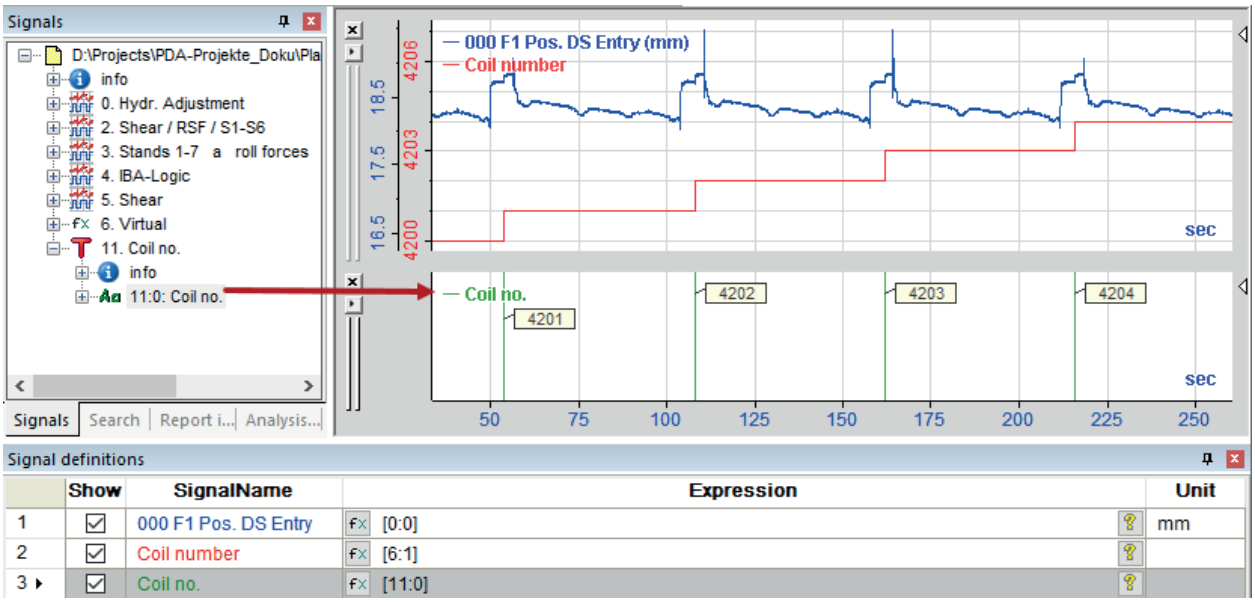
In *ibaAnalyzer*, you can also export text signals into iba data files.

10.1 Display of text signals

In *ibaAnalyzer*, text signals are displayed in the signal tree of the data file. You can drag the text signals into a graph like other signals, or open them by double-clicking.

Text signals are displayed in a graph at the points where they appear in the data file. They appear as a vertical line with a label indicating the value or displaying text.

The following image shows an example with the product number (Coil no.) as a text signal. Each change in value shows a new product number. For comparison, the product number is also displayed as a numerical value with the red curve above.



10.2 Functions for text signals

Using special functions in the *ibaAnalyzer* Expression builder, you can perform operations with text signals in an analysis.

- *InfofieldText, ChannelInfoFieldText, ModuleInfoFieldText*:
  Generates a text signal from an info field.
- *CompareText*:
  Compares the field content of 2 text signals lexicographically and indicates whether the texts are identical or not.
- *ToText, FromText*:
  Generates a text signal from a numerical value with the numerical value as an ASCII charac-

ter (ToText) or vice-versa (From Text).

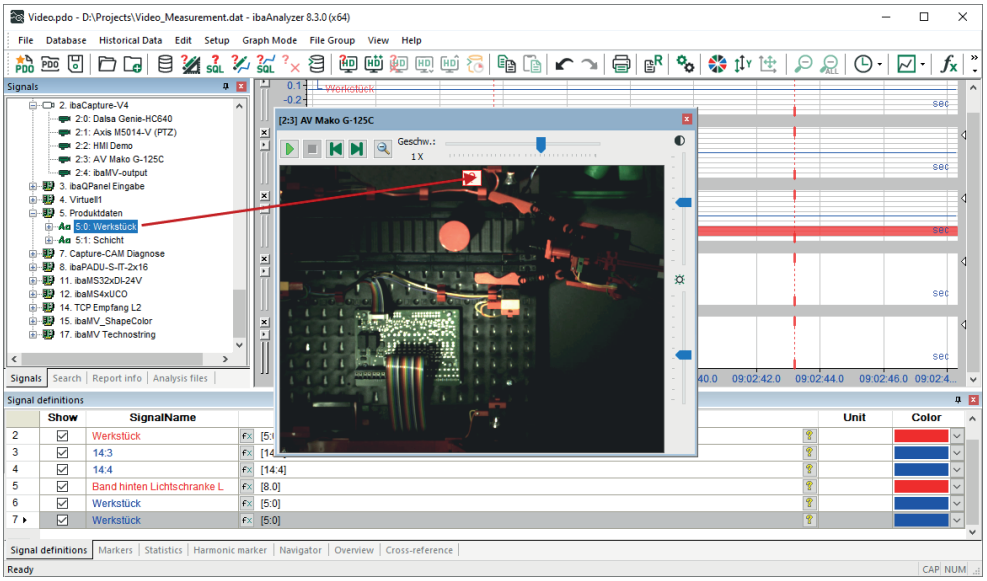
- *TrimText:*
  Removes spaces from the text of a text signal.
- *ConcatText*:
  Combines the content of multiple text signals into one long text.

For more information, see *ibaAnalyzer* manual part 3, chapter *Text functions*.

Other functions
The *Shl* and *Shr* functions (move signal or expression left or right) also support text signals.

10.3 Text signals in videos

When videos from *ibaCapture* are open in *ibaAnalyzer*, you can drag text signals from the signal tree onto the video images as overlay text. Each video window can contain one text signal only.



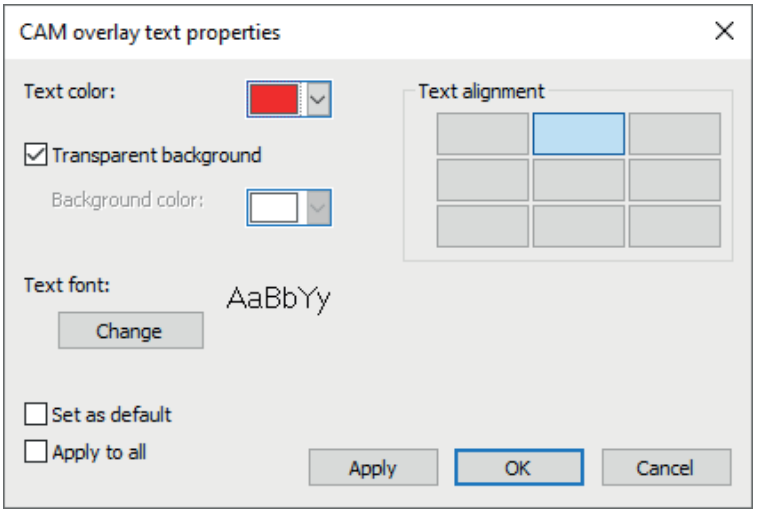
Depending on where you place the text channel above the video window, the text is aligned to one of 9 possible positions.

The text channel is updated synchronously with the video marker when the video is played, as explained in the chapter ↗ *Analysis of measurement signals based on video recording, page 263*.

Use the *Remove overlay text* command in the context menu of the video window to remove the text again.

Change of the text properties

When you have dragged the text signal onto the video, you can use the *Setup overlay text* command in the context menu of the video window to define further properties, such as position, font, background color and text color.



Set as default

If you enable this option, the current settings are saved as default values for new camera windows.

Apply to all

If you enable this option, the current settings are applied to all other visible camera views.

11 Access to HD data with ibaAnalyzer

With *ibaAnalyzer* you can access data recorded with *ibaHD-Server*. This requires a connection to the *ibaHD-Server* and the HD store. The HD query dialog is available in *ibaAnalyzer* for the configuration of HD queries. Here you can define which data is to be queried, e.g. via a time selection or signal-based query conditions. You can also use this query dialog to query time periods from the HD data. For further information see ↗ *Configuration of ibaHD-Server connection and HD queries, page 220*.

The result of an HD query is structured in a similar way to a normal measurement file. This allows you to perform basically the same operations in *ibaAnalyzer* (e.g. display signals, perform calculations, create reports, perform extractions).

Note

With *ibaAnalyzer* v8.1, the numbering of HD events has been changed. If you use analyses with queries from older versions, you may have to reconfigure the query.

Other documentation

Further information on configuring the HD stores and time periods can be found in the documentation for *ibaHD-Server*.

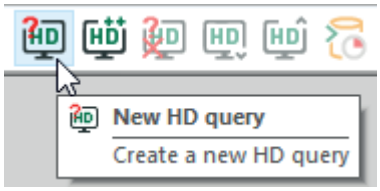
11.1 Opening the HD query dialog

Functions for accessing HD data are available in the *Historical Data* menu. Some commands can also be found in the context menu of the signal tree.

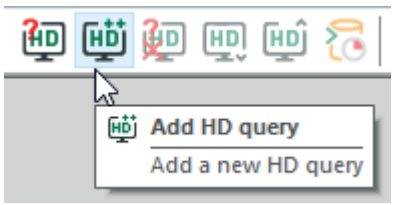
Detailed information on the *Historical Data* menu can be found in the *ibaAnalyzer* documentation, part 1, in the chapter *The Historical Data menu*.

1. To start a new HD query, click on *Historical Data – New HD query* in the menu or click on the corresponding icon in the menu bar.

→ The new query replaces existing files or HD queries in the signal tree.

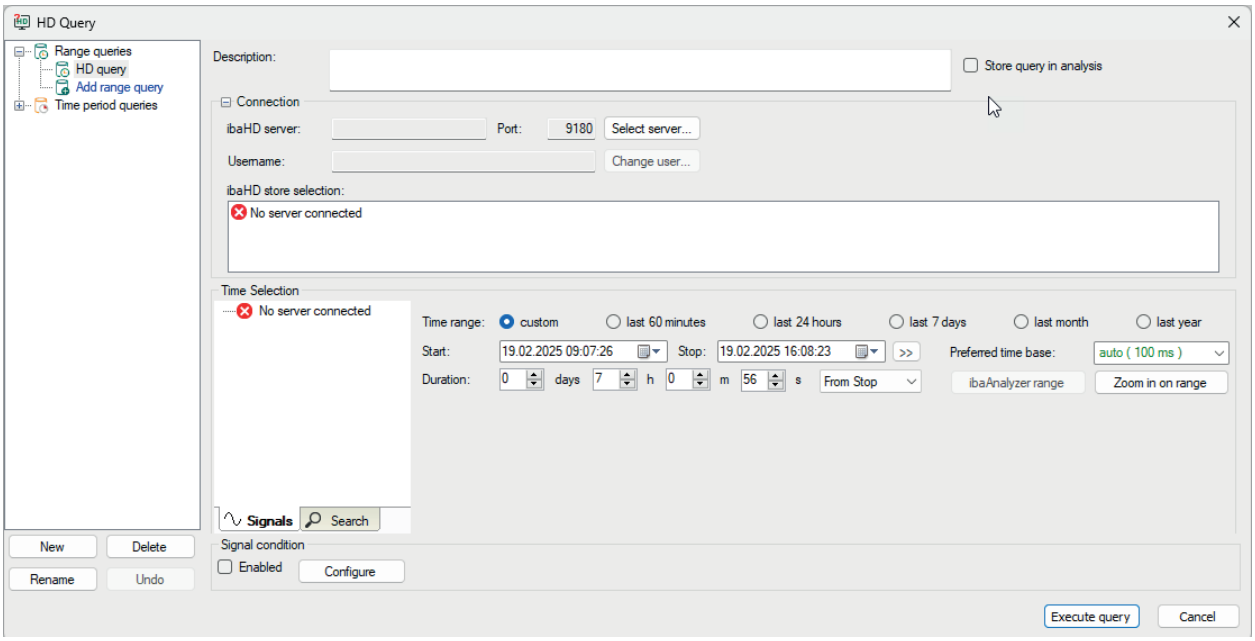


2. If there is already an HD query or a measurement file in the signal tree and you want to add an HD query, click on the menu *Historical Data – Add HD query* or click on the corresponding icon in the menu bar.



11.2 Configuration of ibaHD-Server connection and HD queries

If you add, append or replace an HD query, the HD query dialog opens.



If you are using the HD query function for the first time, you must first establish a connection to the *ibaHD-Server* and the desired HD storage. You can then configure and execute the query.

You can configure several different HD queries and time period queries and assign them a unique name. The queries are listed on the left-hand side of the dialog. You can enter a description for each HD query and specify whether the query should be saved in the analysis.

To configure a HD query, perform the following steps:

1. Set up the connection to the *ibaHD-Server* from which you want to retrieve data, see ↗ *Setting up a connection to the ibaHD server, page 221*.
2. Specify the time range for which the query is to be made.

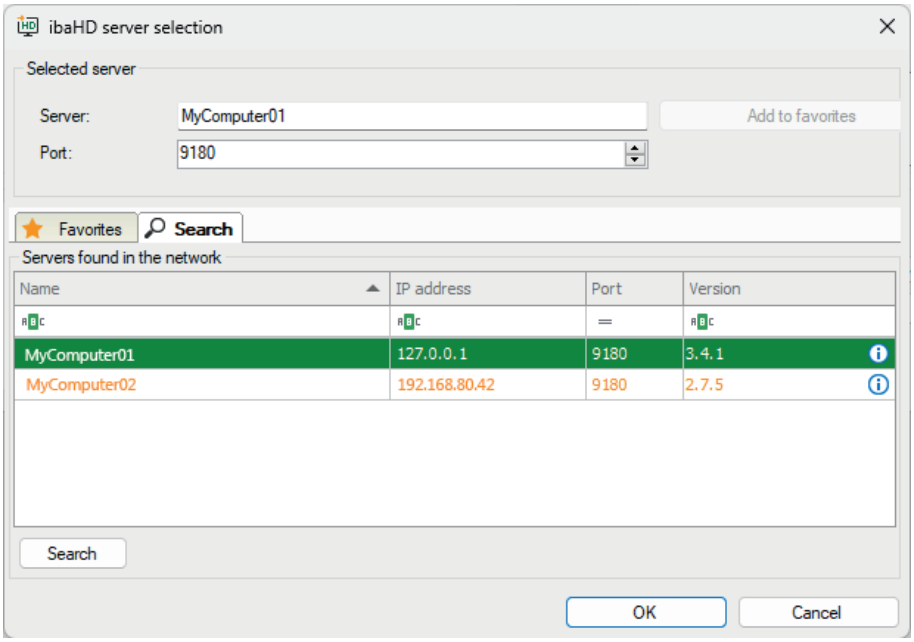
   For HD queries, see ↗ *Time selection for the HD query, page 222*.

   For time period queries, see ↗ *Time period queries, page 233*.
3. Optionally limit the query with conditions to certain events or signal states, see ↗ *Formulating signal conditions, page 228*.

11.2.1 Setting up a connection to the ibaHD server

To be able to execute HD queries, first set up the connection to the *ibaHD-Server*. You can change the connection settings at any time later if required.

1. Click on the <Select server> button in the HD query dialog.



→ The *ibaHD server selection* dialog shows a table with the computers detected in the network on which an ibaHD-Server service is running. If required, you can update the table by clicking on <Search>.

Color	Meaning
Green	*ibaHD-Server* contains stores and data
Orange	*ibaHD-Server* with non-compatible version, some functions are not available
Red	*ibaHD-Server* not compatible

Tip

If you move the mouse over the "i" icon, a tooltip opens with further information. For example, you can see whether time periods are available or which functions are not available (connection marked in orange).

2. Select the desired *ibaHD-Server* in the table.

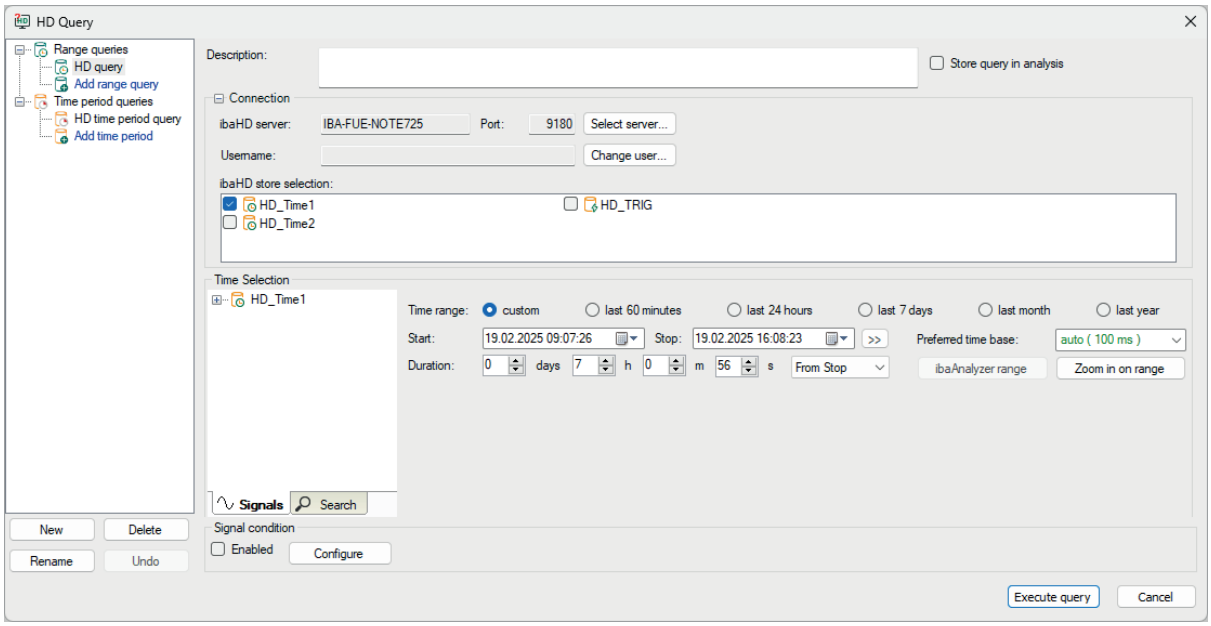
→ The name is displayed at the top of the *Address* field.

Alternatively, you can enter the computer name or IP address manually in the field.

3. Make sure that the port number matches the setting for the selected *ibaHD-Server* service.
4. Confirm the selection with <OK>.
5. If you have selected an *ibaHD-Server* with activated user administration, you still need to enter your username and password.

6. In the *ibaHD store selection* area, select one or more HD stores of the server to be included in the query.

   You can choose between time-based and event-based HD stores.

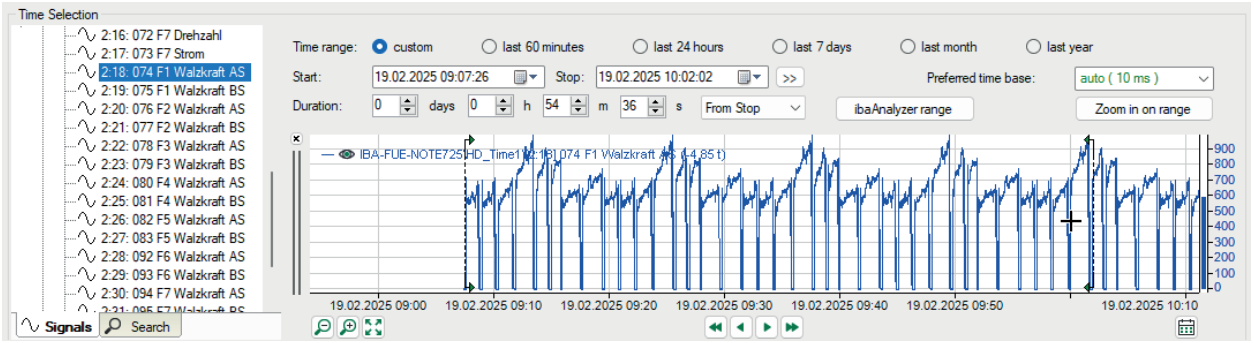


→ A separate data file is displayed in the signal tree for each of the selected stores as a query result.

7. Once you have completed these settings, select the time range, see ↗ *Time selection for the HD query, page 222*.

11.2.2 Time selection for the HD query

In the *Time selection* area of the HD query dialog, set the time range for which you want to execute the query.



Signal tree

In the signal tree, select the signals for which you want to execute the time selection.

For further information on using the signal tree, see ↗ *Time selection – signal tree, page 224*.

Option fields

Option fields make it easier to select the time range.

For all options except *Custom*, the time range is re-evaluated at the time the query is executed.

The toolbar is hidden in the signal view by default. To show the toolbar, click on *Show toolbar* in the context menu of the preview.

Custom
Select this option if you want to select the time range manually with markers in the preview or with the time control elements.

See ↗ *Time selection – preview of the HD trend graph, page 225*.

Last 60 minutes
The queried time range starts one hour before the current time and lasts until the current time.

Last 24 hours
The queried time range starts one day before the current time and lasts until the current time.

Last 7 days
The queried time range starts one week before the current time and lasts until the current time.

Last month
The queried time range starts on the same day and at the same time as one month before the current time and lasts until the current time. The length of the time range corresponds to the number of days in the month before the current month.

Last year
The queried time range starts on the same day and at the same time as one year ago from the current time and lasts until the current time. The length of the time range corresponds to the number of days in the year before the current year.

Start/Stop
Input fields or calendar function for date and time of start and stop time. The fields are only visible if you have selected the *Custom* option.

If you maintain these fields, the markers are positioned accordingly in the HD trend graph.

If you click on the [>>] button to the right of the date field of the stop marker, the stop marker is automatically positioned at the current time and the signal values currently available in the HD store are loaded.

Preferred time base
Selection of the time base with which to load the data.

For information on the time bases offered and how they are formed, see ↗ *Selection of the preferred time base, page 226*.

Duration
Duration of the time range between start and stop time

The values adjust when you move the markers in the HD trend graph or change the values in the *Start* and *Stop* fields, and vice versa fields. You can also enter the duration directly in days, hours, minutes and seconds. In this case, define how the time range is positioned:

- *From Stop*: Stop marker remains in place, start marker is moved.
- *From Start*: Start marker remains in place, stop marker is moved.
- *Centered*: The center of the selected area is stationary, both markers are moved symmetrically to it.

To change a value and set the other values to zero, hold down the <Ctrl> key. For example, you can change the minutes and set the values in the fields for days, hours and seconds to zero.

<ibaAnalyzer range>
Use the button to adopt the time range of the data currently loaded in *ibaAnalyzer* for the query. The button is only active if you have already loaded data before opening the HD query dialog.

<Zoom in on range>
The markers remain in place when zooming in or out or when moving the X-axis. On the other hand, moving the markers or changing the values for the start and stop times does not automatically adjust the zoom factor. The markers can therefore be very close together or outside the window. This button sets the zoom range so that the selected time range is displayed in the middle of the graph. Start and stop markers are positioned at ¼ and ¾ of the visible X-axis section.

HD trend graph

You can also define the time range in the HD trend graph with markers.

For further information on using the HD trend graph, see ↗ *Time selection – preview of the HD trend graph, page 225*.

11.2.2.1 Time selection – signal tree

The signal tree is located on the left-hand side of the *Time selection* dialog. There you will find the HD stores that you selected before under *Connection*. You will find the signals that *ibaHD-Server* has saved there under each HD store.

Commands for the display mode and filtering of signals are available in the context menu of the signal tree.

Display mode

Specify how the signals are displayed in the tree.

Note



The setting you select here for the display mode is applied to the normal signal tree in *ibaAnalyzer* and vice versa.

Filter settings

The selection under *Signal filter* determines which signals are displayed in the signal tree. *Active signals* are signals that are currently also written in the HD store. *Inactive signals* are signals which are recorded but not currently written.

Search

You can search for specific signals in the *Search* tab. The function is similar to the search function in the normal *ibaAnalyzer* search window.

11.2.2.2 Time selection – preview of the HD trend graph

You can display signals from the signal tree in the HD trend graph in the *Time selection* area on the right. Select the desired signals as usual using drag & drop or double-click.

The controls for navigating the HD trend graph are similar to those in *ibaPDA*:

- Zooming the time axis in and out with the mouse wheel
- Move the time axis towards the past or future with the mouse
- Zoom in and out by adjustable factors with plus/minus buttons
- Navigate towards the past or future with configurable step buttons
- Navigate to a specific date using the calendar function

A live mode is not available.

You can also limit the desired section. Once you have found the time or time range you want to query using the navigation tools, you can set the start and end of the time range in various ways.

Set start and stop markers

As soon as a signal is displayed, there are two markers on the signal trend. You can move the markers with the mouse to set the start and end of the period.

The values for the date and time of the set start and stop time are displayed in the corresponding option fields above the graph.

The cursor changes when you move over the markers at the top or bottom end of the signal trend:

	The mouse is on the start marker
	The mouse is on the stop marker.

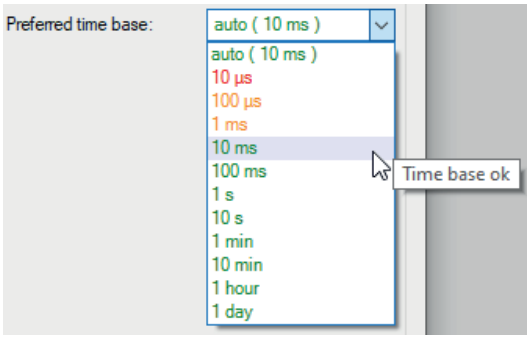
To move the markers at a constant distance, hold down the <Shift> key while moving them.

Enter start and stop time

Instead of using markers, you can also define the start and stop times manually using the corresponding fields and options and thus limit the time range more precisely, see ↗ *Time selection for the HD query, page 222*.

11.2.2.3 Selection of the preferred time base

The time base with which the loaded signal values are to be displayed later is set in the HD query dialog in the *Time Selection* area via the *Preferred time base* drop-down list.



Depending on the requirements for the time resolution of the queried data, select a small time base (high resolution) or a larger time base (lower resolution).

The values offered in the list are standard values and are only approximate. The actual time base with which the data can be loaded is determined by the store in the *ibaHD-Server*. Only the original time base (highest resolution) and the automatically determined time bases of the various aggregation levels are available.

The following cases may occur when selecting the preferred time base:

- The preferred time base is the same as an existing time base in the HD store:

  The data is loaded with this time base.

- The preferred time base is smaller than any time base in the HD store:

  The data is loaded with the smallest available time base.

- The preferred time base is between a smaller and a larger time base in the HD store:

  The decision as to which existing time base is loaded is made by *ibaHD-Server* using the following formula:

  - The larger time base is loaded when

    $$\frac{sample\ period\ of\ coarser\ level}{preferred\ time\ base} < \frac{preferred\ timebase}{sample\ period\ of\ finer\ level}$$

  - Otherwise, the smaller time base is loaded.

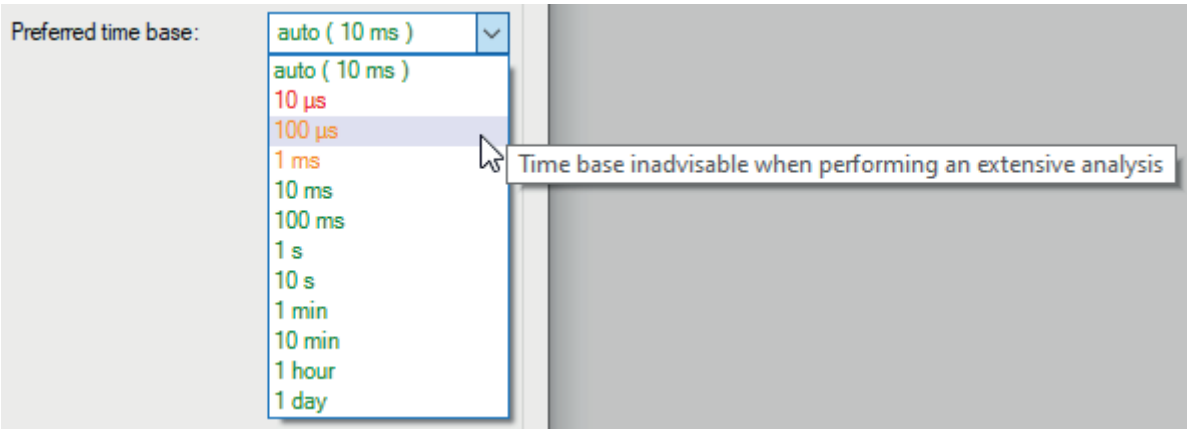
Depending on how large the set time range is, the choice of time base has a considerable effect on the amount of data.

In the HD query dialog, you can only select the time range for the query, not for specific signals. During an HD query, *ibaAnalyzer* always loads all signals contained in the HD store in question for the set time range.

If the set time base is very small and the time range is very large, the amount of data to be loaded may exceed the memory capacity. This means that further processing or analysis of the data is only possible to a limited extent or not at all. *ibaHD-Server* therefore calculates the amount of data to be expected depending on the set time range and marks the limits in color for the time

base values in the drop-down list. If you select *Auto* as the preferred time base, the optimum time base is selected automatically.

In addition, a tooltip indicates possible difficulties if you select the relevant time base anyway.

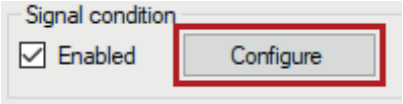


Color	Meaning
Red	The set time range and this time base require more memory than *ibaAnalyzer* can provide. If you select this time base without reducing the time range and execute the query, an error message appears. The query cannot be executed. Set a different time range or a different time base in the HD query dialog.
Orange	With the set time range and this time base, *ibaAnalyzer* can provide sufficient memory. However, only limited analysis functions are possible or only a few signals can be displayed. If you select this time base without reducing the time range and execute the query, a message appears. You can execute or cancel the query and select a different time range.
Green	With the set time range and this time base, extensive analyses are also possible without any problems. If you close the HD query dialog with <Execute query>, the query is executed.

11.2.3 Formulating signal conditions

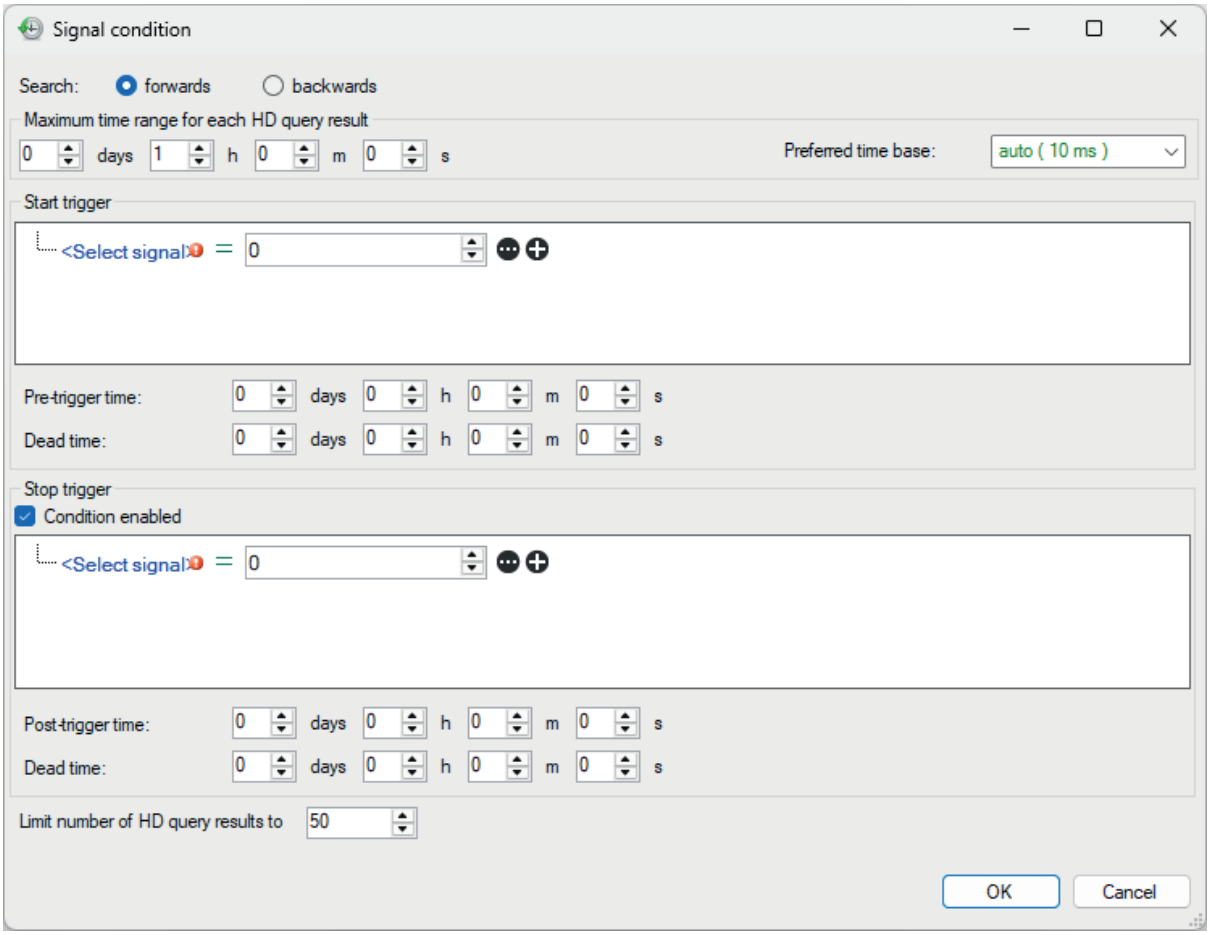
You can use signal conditions to configure conditional queries and limit the query to certain events or signal states within the set time range.

1. Activate the *Enabled* option.



2. To configure the signal conditions, click on the <Configure> button.

→ The *Signal condition* dialog opens.



Search
Select whether you want to search for conditions forwards or backwards in time. If *forwards* is selected, *ibaAnalyzer* starts searching for events of the specified condition, starting from the start time to the end time. If *backwards* is selected, *ibaAnalyzer* starts searching from the end time to the start time (i.e. backwards in time).

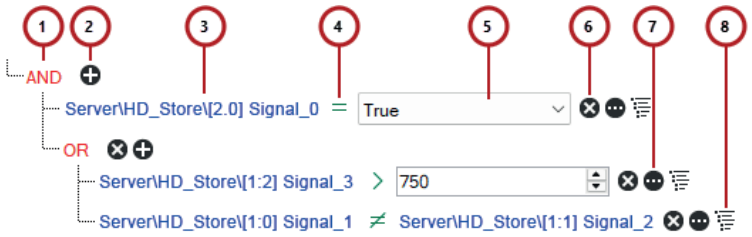
Maximum time range for each HD query result
The query with reference to a specific signal condition can return several results because a measurement file is generated for each time the condition is met. You can set the maximum length of these query results here.

If you use a stop trigger and it fails, this setting limits the length of the query result.

Start trigger
Use the *Start trigger* to configure the condition that defines the event or signal status to be searched for. The time at which the event occurs is the start of the query result or the measurement file. You determine the end of the query result either by a *Stop trigger* or the setting for the maximum time range.

Various editor functions are available in the *Start trigger* field for defining the signal condition.

The start trigger triggers when the overall result of the condition is fulfilled.



1	Logical link	By default AND Click on the link to switch between AND and OR.
2	Add expression	
3	Blue font	Signal tree for signal selection
4	Operator	Comparison operators for selection
5	Comparison value/ Comparison signal	Digital signals: True/False or signal Analog signals: Value input or signal Text signal: Text input or signal
6	Remove expression	Complete expression or group is removed.
7	Switch expression type	Switches the expression type on the right side: constant value or signal. This allows you to compare a signal directly with another signal or a fixed value.
8	Add expression group	Indented, with a separate logical link, you can add further expressions that are initially linked within the group. The group result then leads to the result with the superordinate link.

Pre-trigger time
Specify how much time should be included in the query before the start trigger.

Dead time
Specify how long the time range is after a start trigger before a new start trigger is detected.

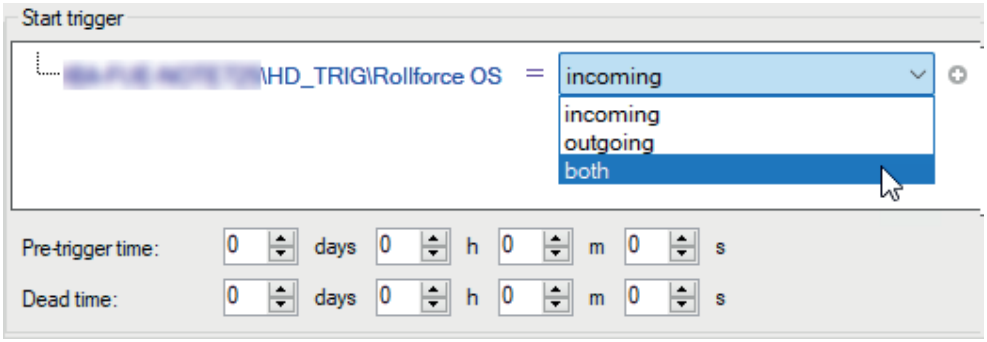
Stop trigger
Optionally define the end of a query range. The settings for the formulating the condition, the post-trigger and the dead time correspond to the start trigger.

Limit number of HD query results to …
Here you can limit the number of query results for a file group.

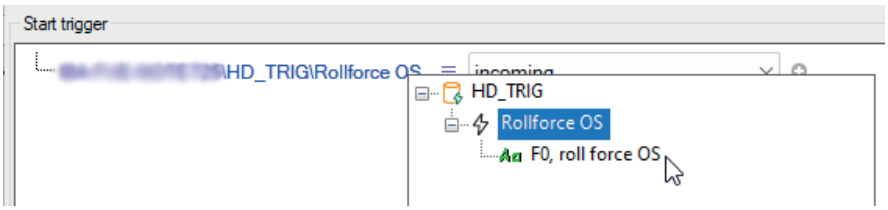
11.2.3.1 Formulating conditions for events

Queries based on a condition can also contain events. The following options are available:

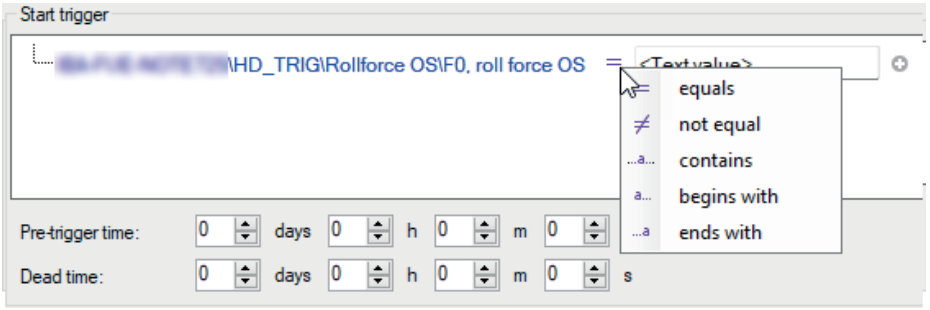
- Specify the event and also whether the condition is fulfilled when the trigger is received, goes out or both.



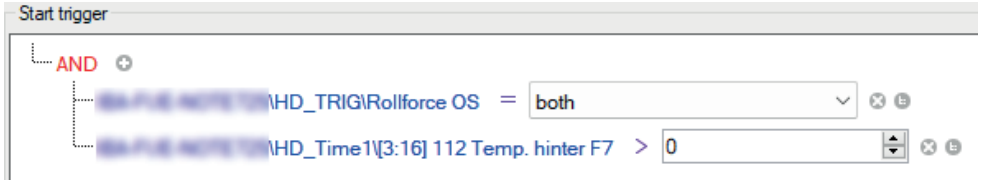
- Use a numeric field of the event and check it for a limit value. This is comparable to applying a condition to an analog signal.



- Check whether the text fields of an event are identical or partially identical to a specific text.



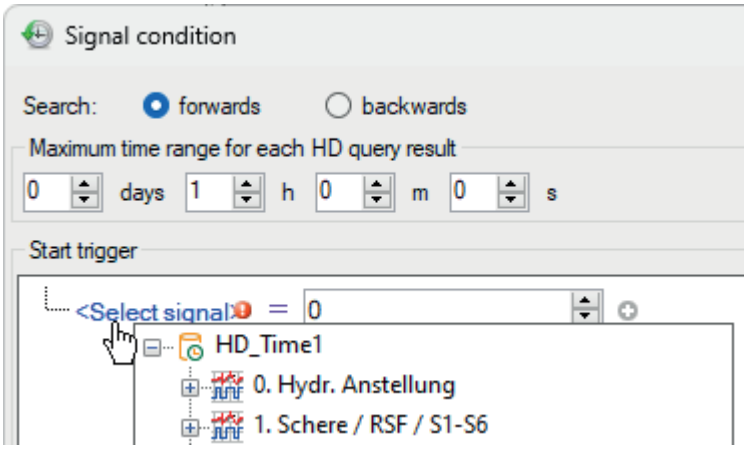
- Link the conditions for events with other event conditions or with conditions for normal HD signals using the Boolean operators AND or OR.



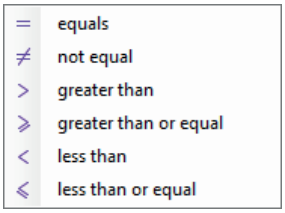
11.2.3.2 Example: Formulating signal condition

The following example shows the procedure for creating a condition.

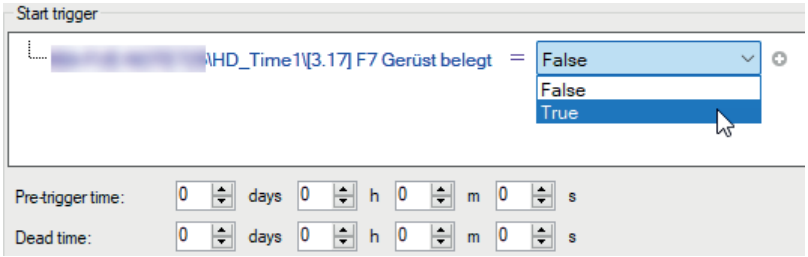
1. Click on the blue text *Select signal* and select the desired signal in the signal tree, e.g. a digital signal.



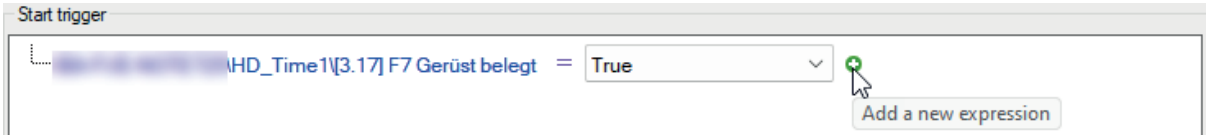
2. Click on the operator symbol and select the operation, e.g. equal or unequal for a digital signal.



3. Select a comparison value, e.g. "True" for a digital signal.

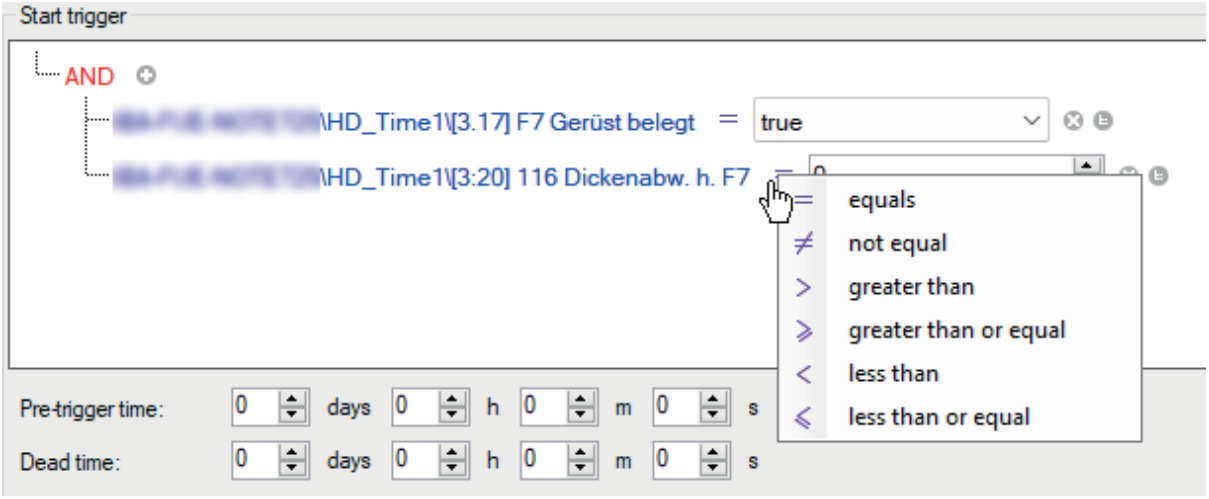


4. If you want to add another condition, click on the  icon.



→ A new expression is created and logically linked to the previous expression with AND.

5. Formulate the second expression, e.g. with an analog signal that you compare with a limit value.



6. Enter the desired limit value above which the trigger should be activated by the selected analog signal.

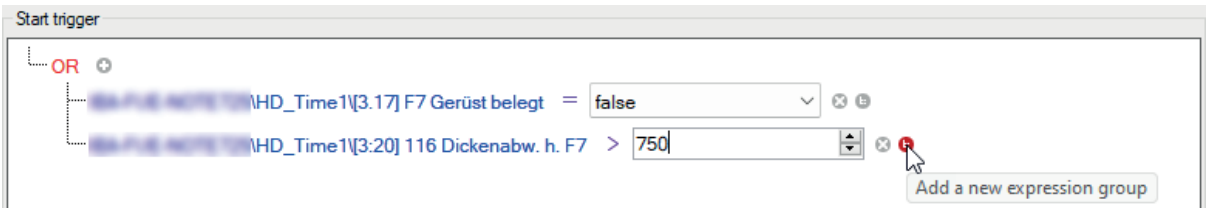
   Here in the example 750 µm.



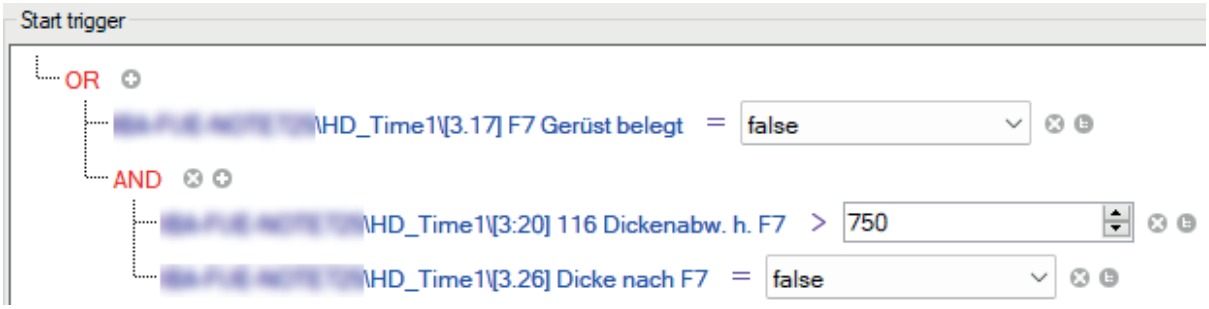
7. With the AND link, both expressions must be "True" for the trigger to fire. If you want the trigger to fire when only one condition is met, click on the red AND to switch to the OR link.



8. You can also cascade conditions by combining expressions into groups. To create a group, click on the icon for the expression that is to be the first member of the group.



9. The expressions in the group are combined with their own logical link (AND by default).



To add further expressions to the group, use the icon at group level.

11.2.4 Time period queries

To query time periods, *ibaAnalyzer* must be connected to a time-based HD store in which time period stores are configured. As with an range query, the result of the query is displayed in the signal tree of *ibaAnalyzer*. You can treat the query result like a data file.

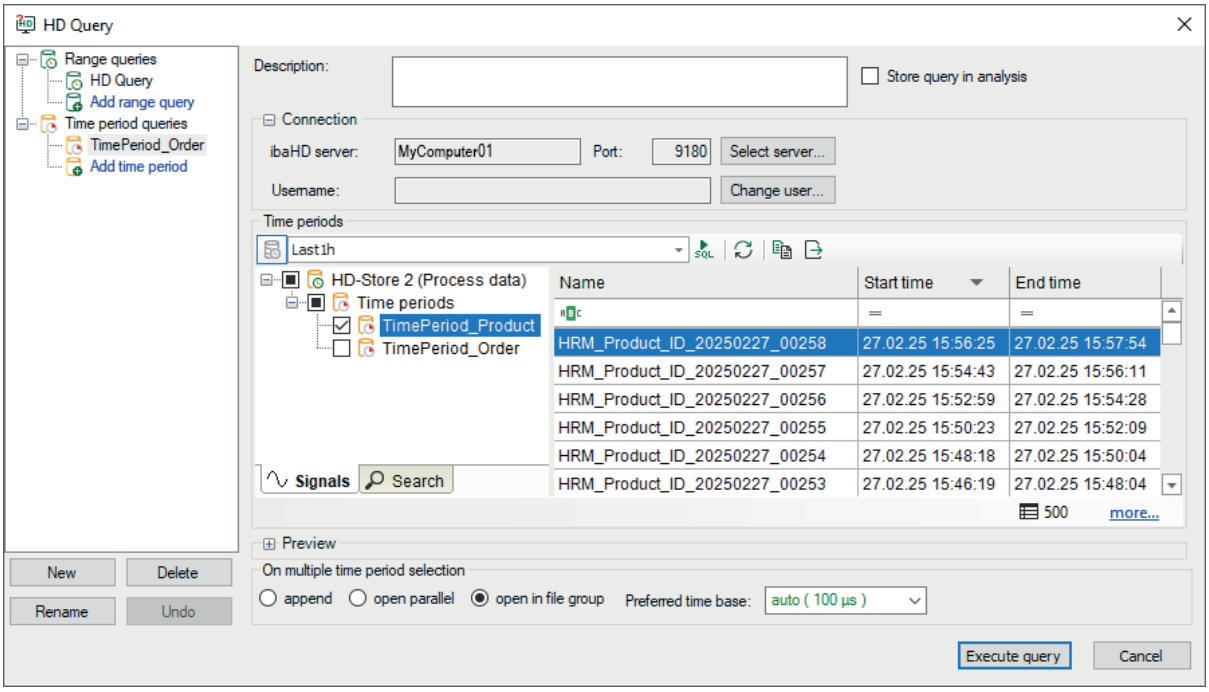
The time period function is supported by *ibaHD-Server* v3.1 or higher.

Create time period queries

On the left-hand side of the dialog you will find the time period queries under the range queries.

1. Click on *Add time period* in the left-hand tree or select an existing query.
2. In the *Time periods* section of the store tree, select the time period stores that you want to display.

→ The table shows the time periods of the store. The functions of the table are similar to the time period table in *ibaPDA*, see ↗ *Operation of the time period table, page 234*.



3. Select the relevant time periods from the table.

   If required, you can also filter the time periods in the table using the columns or with conditions, see ↗ *Configuring time period queries, page 236*.

4. If you have selected multiple time periods, use *On multiple time period selection* to determine how *ibaAnalyzer* should handle the query result of the time periods.

   For further information see ↗ *Query results of the time periods, page 244*.

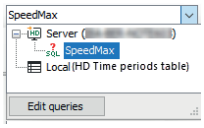
5. Click on <Execute query>.

→ The query result is displayed in the *ibaAnalyzer* signal tree.

11.2.4.1 Operation of the time period table

The time period table shows the time periods from the selected time period file.

The toolbar for the time periods table contains the following controls:

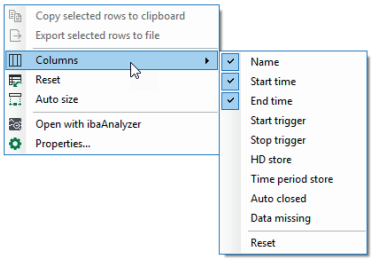
	Switches back to the display of the latest available time periods after a condition query.
	Drop-down list for selecting the time period query. If you expand the list field, you can see all existing time period queries and select the one you want. The <Edit queries> button opens the configuration dialog for conditional queries within the table. You can then edit existing queries or create new ones.
	Executes the condition query selected in the drop-down list. If the query takes longer, a progress bar is displayed in the footer of the table. A link will also appear that you can use to cancel the query if it takes too long.
	Updates the display according to the query executed.
	Copies the time periods marked in the table to the clipboard.
	Exports the time periods marked in the table to an Excel or text file.

Status bar

The status bar is at the lower edge of the table. It provides information about SQL queries carried out (e.g. status of the running query, number of results, error messages) and whether a filter is currently active.

Show/hide columns

You can show and hide columns in the table's context menu. The upper columns are the standard columns. The lower columns refer to the info fields. You can configure the info field columns in the properties of the time period table.



Filter line

A filter line is located directly below the column headings. This allows you to filter using text input or other options. Press <Enter> after the entry and the table will be sorted. To return to the unfiltered view, delete the entries from the filter line. The filter options are determined by the data type of the info fields.

A detailed description of the filter functions can be found in the documentation for *ibaPDA*.

Open time periods ibaAnalyzer

You can use the context menu to open the selected time period in a new *ibaAnalyzer* window. If you have selected an *ibaHD-Server* with activated user administration, you still need to enter your user name and password.

Sorting and filtering

Sorting

You can sort the table by each column in ascending or descending order. Click on the header of the column which you want to use as sorting criterion. The arrow shown (down or up) indicates the sort order (descending or ascending). By default, the table is sorted by the *Start time* column so that the most recent time period is at the top.

Filter line

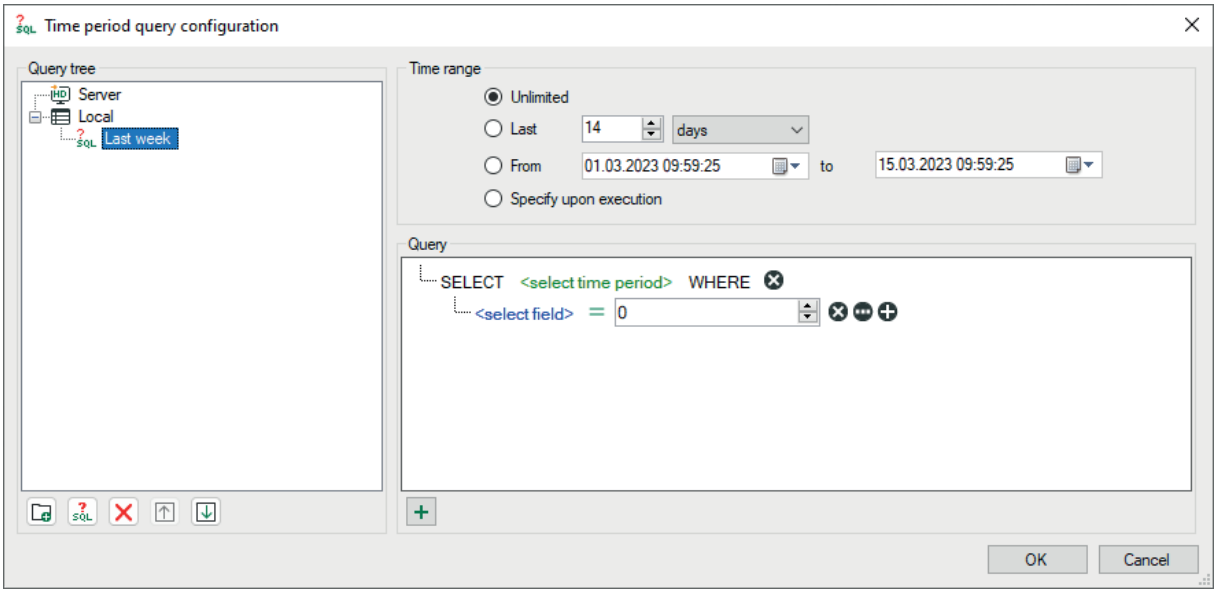
A filter line is located directly below the column headings. This allows you to filter using text input or other options. Press <Enter> after the entry and the table will be sorted. To return to the unfiltered view, delete the entries from the filter line. The filter options are determined by the data type of the info fields.

11.2.4.2 Configuring time period queries

You can use time-period queries to filter the stored time periods, for example, for a specific time range. In this way, you can easily find the specific time periods you want to display in the HD time period table or load into *ibaAnalyzer*.

In the time period table toolbar, you can filter the rows displayed by particular criteria. Enter a value or configure a query by clicking on the icon. If you remove a value, the filter is deleted.

To configure time period queries, open the query selection list and click on the <Edit queries> button.



Note

Depending on the configuration, time period queries are stored either on the *ibaHD-Server* or locally.

Configuring queries

All existing queries are shown in the query tree. You can arrange queries by using a directory structure.

	Adds a new folder at the selected position
SQL	Adds a new query at the selected position
	Deletes the selected node and any secondary nodes
	Moves the selected query within the folder or moves the entire folder up
	Moves the selected query within the folder or moves the entire folder down

You can also move queries or folders using drag & drop or the context menu for queries and folders to change their order.

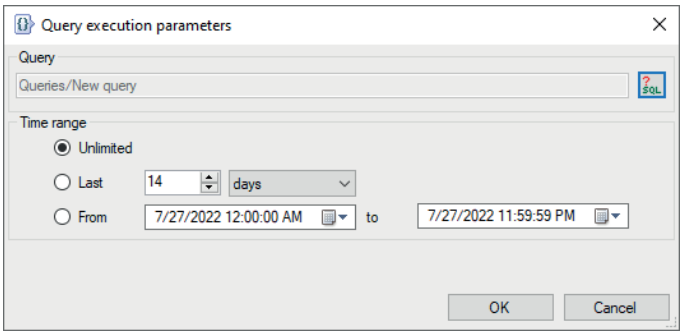
There are two types of queries:

- *Server queries* are saved on the *ibaHD-Server*, which means that all connected clients can use the same server queries.
  Note: Only users who have the right *Edit ibaHD queries* can save or edit server queries.
- *Local queries* are saved in the query. Other connected clients cannot see these.

Query range

Every query is executed over a defined time range. There are several options for restricting the query range.

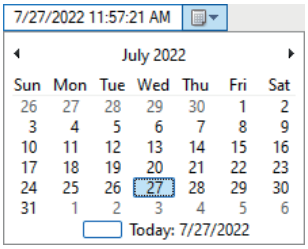
- Unrestricted: The query covers the entire HD recording
- Specify upon execution: After clicking on the start button for the query, a dialog appears in which you have to specify the query range



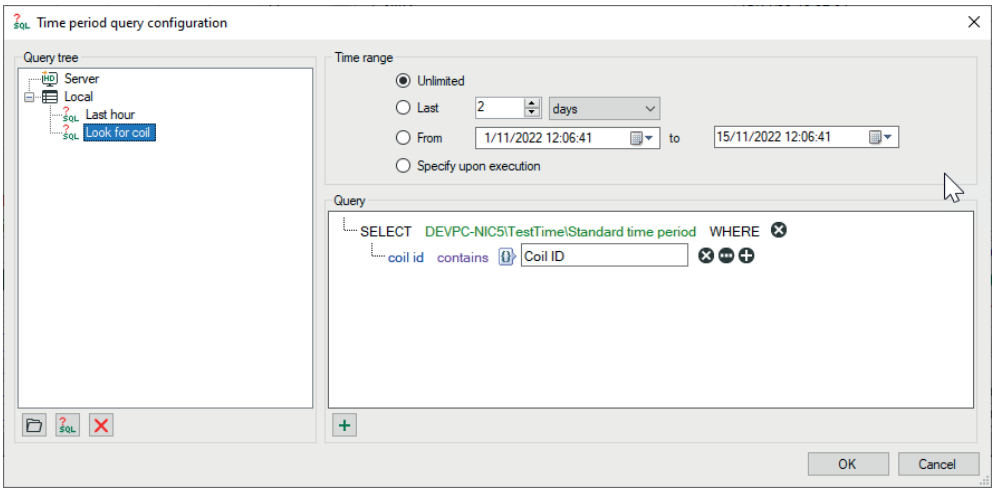
  Clicking on <OK> executes the query.
- The last xx minutes/hours/days/weeks/months: The query only covers a fixed period of time from the start of the query, e.g. the last 8 hours for a shift overview.
- From *Date* to *Date*: The query covers the time range between the set days (inclusive).

Tip

To set the current date, open the Date & Time Picker by clicking on the down arrow. Here, you can select the entry under *Today…*.

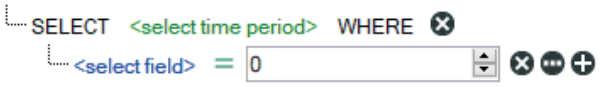


Query details

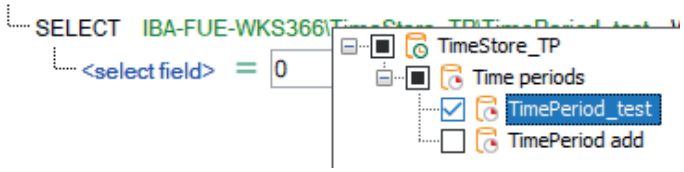


Here, you create the actual query using notation similar to SQL.

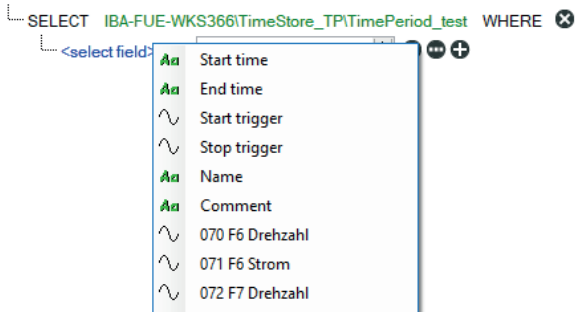
The query begins with a SELECT statement.



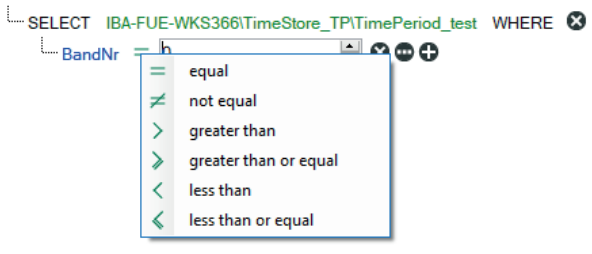
Clicking on the green text opens the tree structure for the configured time periods. Select the time period you want to search for.

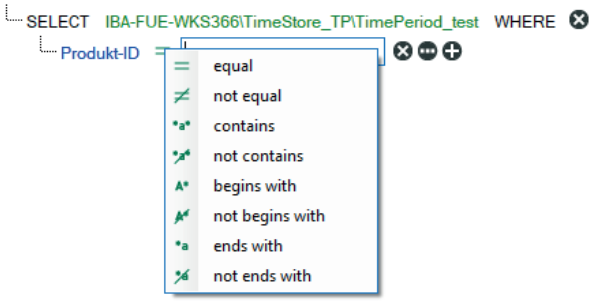


Click on the blue text to select a column in the table or any existing information fields.



Depending on the selected field (numeric or text field), clicking on the green equals symbol displays a list of operators. Select the required operator.





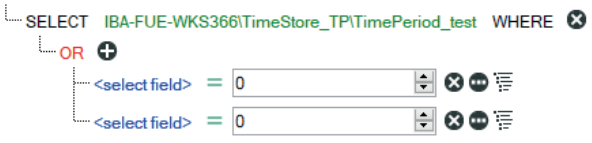
Behind the elements in the query, there are different buttons which you can use to extend or modify the query:

 Deletes the expression

 Switches the right side of an expression between a fixed value and a field

 Adds a new expression to the instruction

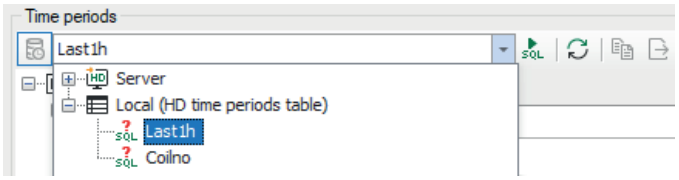
You can change the logical operator that is linked with a group of expressions by clicking on the operator.



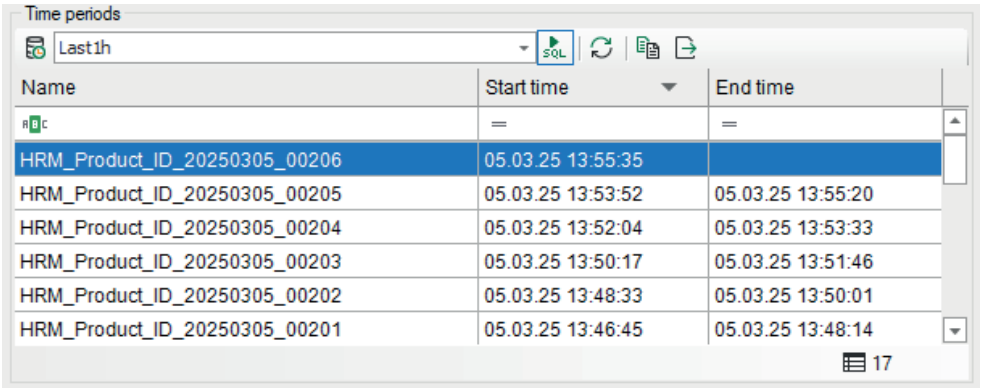
Clicking on <OK> closes the editor and saves the created queries.

Executing the query

First select the relevant query from the list in the toolbar and then click on the button to the right.



When the query has been completed, the number of results found is displayed in the status bar on the right.



A query can return more results than the number of rows configured in the table. To display any other results, click on the blue “more...” link.

With the  button you can return to the display of the latest available time periods.

11.2.5 Query of non-equidistant data from ibaHD-Server

In connection with the storage of non-equidistant data in *ibaHD-Server*, please note the following points when querying.

HD query

Non-equidistant data is recorded differently in *ibaHD-Server* than the usual time based or length based data.

Non-equidistant recording only takes place at the raw data level, i.e. the level where the uncompressed data is stored. There, the non-equidistant data is available as time-value pairs. At the aggregation levels, on the other hand, the data is available in equidistant form.

If you execute an HD query and the query result originates from an aggregation level, you will not recognize any difference at first. Only after a drill down to the raw data level, the non-equidistant character would be recognizable.

Use the *IsNE* function from the Expression builder to create an indicator that shows whether a signal or expression is equidistant (IsNE = FALSE) or non-equidistant (IsNE=TRUE).

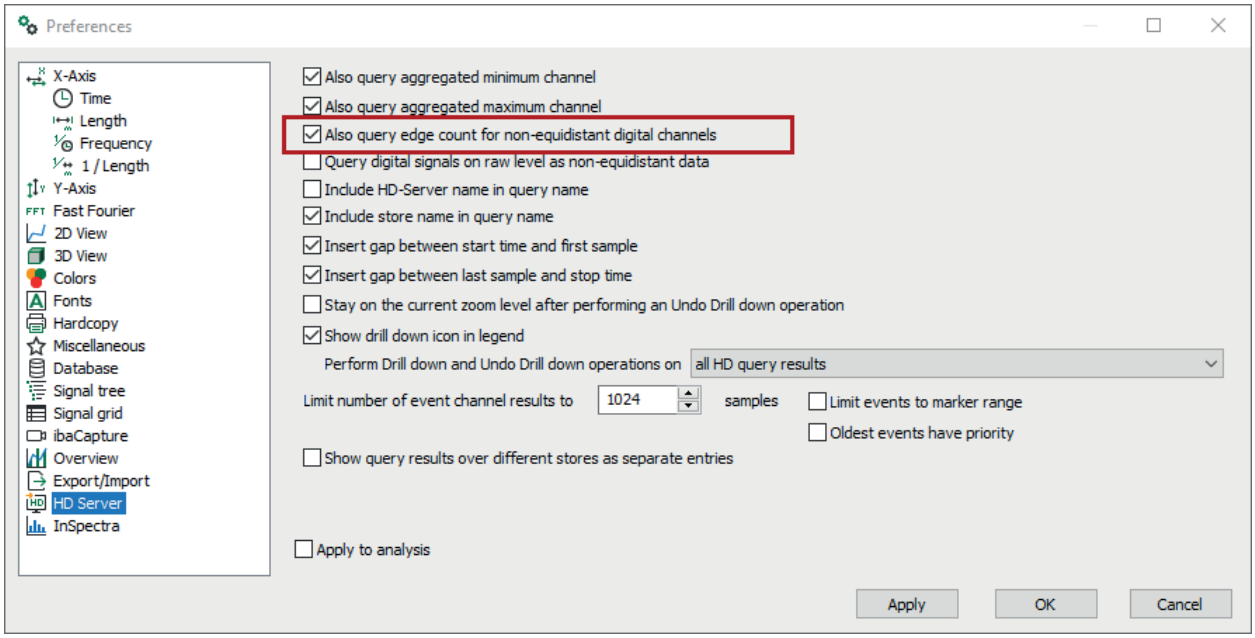
Drill down

If you drill down to the raw data level, note the following.

1. The time base displayed for the signals (including sub-signals) is the smallest time interval between two data points.
2. The raw data corresponds to the displayed min and max channels.
3. Non-equidistant signals are automatically displayed as a staircase curve (*Staircase* mode).
4. The result of the *IsNE(…)* function is TRUE.

"edgeCount" sub-signal for digital signals

Digital signals that are recorded with a non-equidistant profile can receive an additional "edgeCount" sub-signal. This sub-signal provides the number of edges within the aggregation time for the aggregation level. You can specify whether this sub-signal is visible or not in the preferences or graph setup in the *HD server* node.



Note

Please note that no subsignals of the digital signals are visible if you activate the option *Query digital signals at recording level as non-equidistant data*.

11.3 HD query results

The *ibaAnalyzer* signal tree displays the result of an HD query for a period of time like a data file.



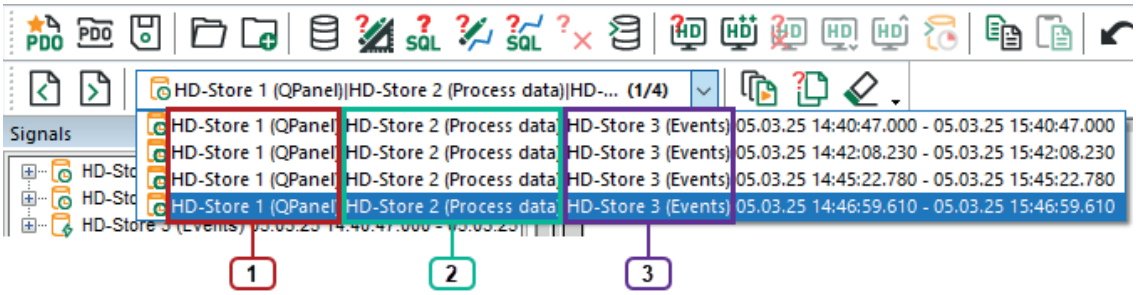
The signals in the query result are treated in the same way as the signals from a normal data file.

You can also apply analyses that were created using a data file to HD query results and vice versa. You also have the option of exporting query results as a DAT file.

See ↗ *Opening the data files*, page 16.

Depending on the options selected under *HD Server* in the graph setup or preferences, you can query the maxima and minima of the aggregated values of a signal as subchannels, see ↗ *Subchannels min/max, page 249*. This can be particularly interesting when selecting a large timebase or data from a higher aggregation level. So-called outliers are then easier to recognize.

Results of a query with a signal condition are displayed as a file group, as there are usually several results. By default, one line is displayed for each search result in the drop-down list, even if several HD stores are involved.



1	Time-based store 1
2	Time-based store 2
3	Event-based HD store

If you want the results being listed separated by HD stores in the file group field, you have to enable the option *Show query results over different stores as separate entries* in the preferences, *HD Server* tab.

11.4 HD query results of an event-based HD store

In *ibaAnalyzer*, events are available as text signals. If numerical fields are defined for the event, these events are available as analog, non-equidistant subchannels of the event text signal. All text fields of the events are also available as subchannels of the event text signal.

Three additional signals are available for each event:

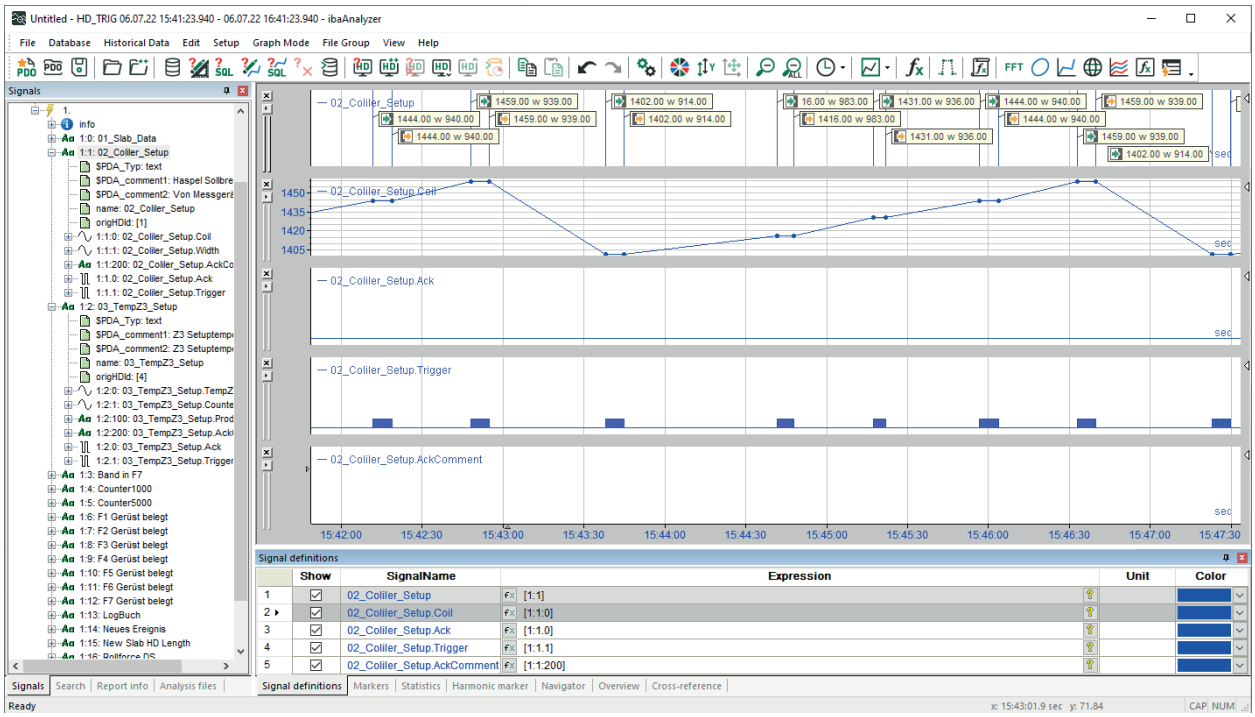
- **.Ack*: Non-equidistant digital signal

  The signal is true for every confirmed event and false for every unconfirmed event.

- **.Trigger*: Non-equidistant digital signal

  For an event that can be both incoming and outgoing, this signal is true if the event is incoming and false if the event is outgoing. This signal is always true for a signal that is never outgoing.

- **:AckComment*: A text signal containing the confirmation comments.



The events are marked by icons as incoming or outgoing events in the graph.

Similarly to *ibaPDA*, you can also use the event table for HD data in *ibaAnalyzer*, see ↗ *Event table*, page 166.

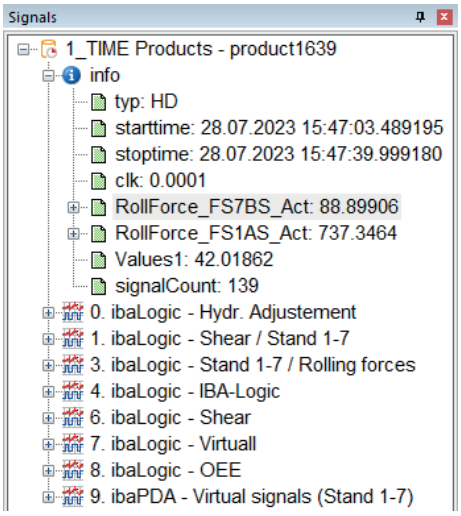
Display options in the signal tree

If you select the display option *Show groups per file* or *Show groups across all* files in the context menu of the signal tree, the displayed groups correspond to the folders in which the events are organized on the *ibaHD-Server*.

11.5 Query results of the time periods

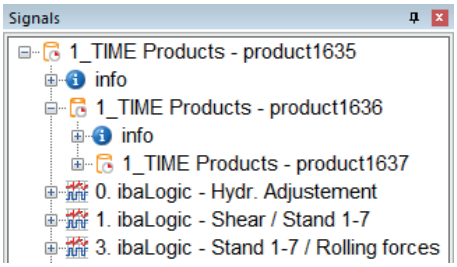
ibaAnalyzer displays the result of a time period query like a data file in the signal tree. The options you have for HD query results also apply to time periods, see ↗ *HD query results, page 242*.

The signals and info fields come from the HD store.



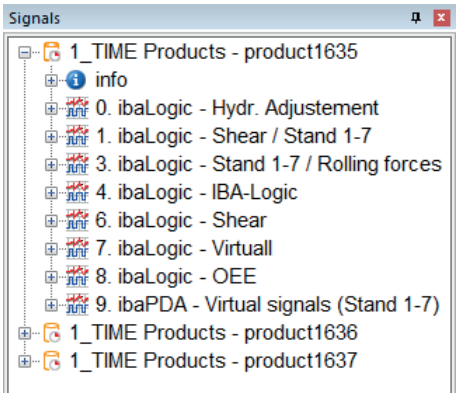
If you have queried several time periods, *ibaAnalyzer* can handle the query result of the time periods in different ways. You determine this in the query under *On multiple time period selection*.

- *Appended*: The time periods are displayed one after the other as if the data files were appended to each other, see ↗ *Appending data files*, page 23,.

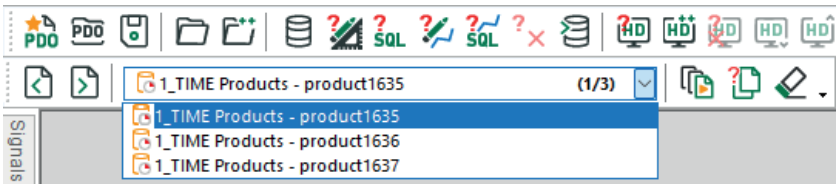
  

  You can also display the info fields as a graph by double-clicking on them. For further information, see *ibaAnalyzer* manual part 3, *InfoField*, *InfoFieldText* and *InfoFieldTime* functions.

- *Parallel*: The time periods are opened side by side.

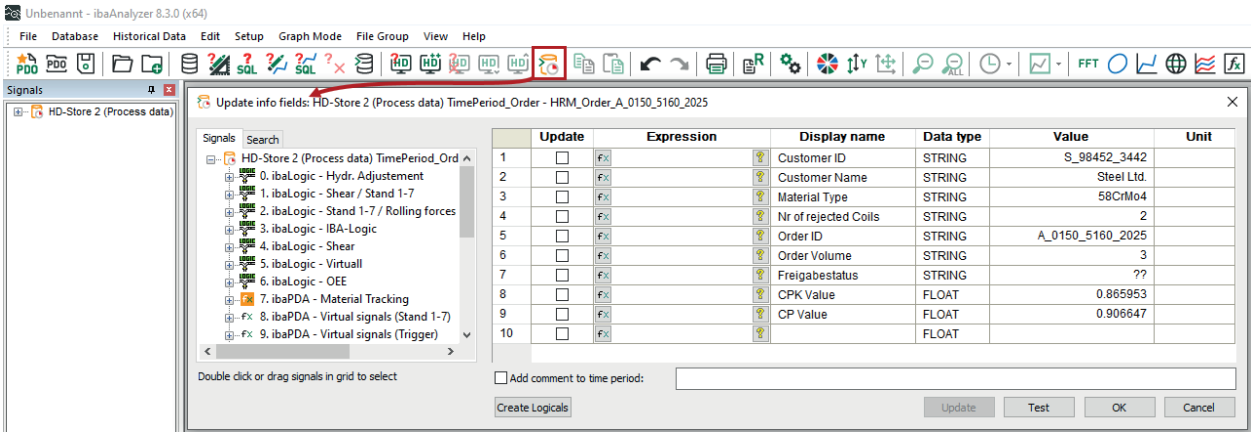
- *File group*: The time periods are available in the file group field and are treated like a data file group., see ↗ *Defining groups of data files*, page 21. You can also perform a file group query, see ↗ *Trend query from file groups or time periods*, page 32.



11.6 Updating the info fields of the time periods

First, *ibaPDA* writes the info fields of the time periods. In *ibaAnalyzer*, however, you can update the existing info fields and create new info fields.

You can open the *Update info fields* dialog from the toolbar or via the *View* menu if you have queried at least one time period.



The dialog shows all available info fields in the time period store. The values correspond to the values of the respective time period. Question marks "??" indicate fields without values.

You can show and hide some columns via the context menu in the table.

The *Add comment to time period* option allows you to add any text as a comment to any time period. Note that the input field always shows the last entered text. This may not match the comment for the selected time period. You can find the actual comment in the info fields. Click <Update> to set the comment.

Use the <Create logicals> button to transfer expressions from the *Expression* column to the group *Time Period values* in the Logical expressions. Then these expressions appear also in the signal tree. In the *Update info fields* dialog, the expression is then replaced with its display name and group name. For example, instead of the entered expression "Max([Signal])" with the display name "Max", "Time period values\Max" appears in the Expression column after the transfer.

Editing existing info fields

You can change the values of the info fields by entering a suitable expression. You cannot edit other columns.

1. Activate the *Update* column in the desired rows.
2. Enter suitable logical expressions in the *Expression* column to change the value.
3. Optionally: Check the new values with the <Test> button.

→ Changed values are highlighted in orange and unchanged values in green.

4. Apply the changes using the <Update> button.

→ All selected info fields are updated and existing values are overwritten.

Creating new info fields

You can also create new info fields and add them to the time period store.

1. Drag a signal from the signal tree of the dialog into the bottom empty row of the table. Optionally, you can also enter the data manually.

→ The data is added to the corresponding columns and a new empty row appears.

→ The rows of new info fields that are not yet saved in the store are highlighted in yellow.

2. Specify a display name and the data type.
3. Click <Create> to add the new info fields to the time period store.

→ The info field is created and the configured value is written for the time period.

→ The info field is available for all time periods in the time period store.

→ You can edit the info field in the same way as other existing info fields.

11.7 Drill-down function

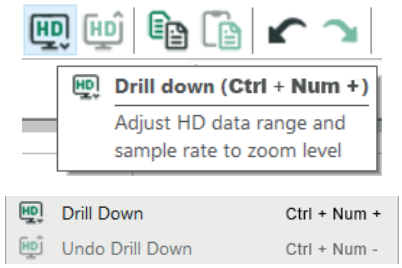
If you use the zoom function in *ibaAnalyzer*, the data is not reloaded, unlike with the HD trend graph in *ibaPDA*. The number of samples and therefore the resolution of the signal curve remains unchanged.

If you have executed an HD query with a timebase of 1 min, for example, because the time range was correspondingly large, then the distance of 1 min is also kept when zooming in. Zooming in therefore does not create any new information.

For a better data resolution, you can therefore perform a so-called drill-down in the zoomed-in display.

During a drill-down, the time range and timebase are recalculated according to the set zoom level and the data is requested from the HD server accordingly.

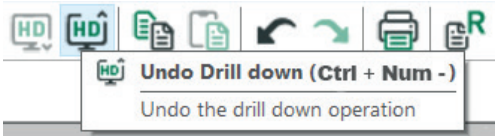
To perform a drill-down, click on the corresponding icon in the menu bar or the command in the *View* menu:



The command is only available in the menu bar and menu bar if you have previously zoomed in. If you want to perform another drill-down, you must zoom in further.

Undo drill-down

You can undo the drill-down operations step by step. The icon in the menu bar and the *Undo drill-down* command are only available if you have performed at least one drill-down.



You can only apply drill-down operations to HD queries whose signals are displayed in the currently selected graph or are used in an expression that is displayed in the currently selected graph.

Accordingly, you can only undo a drill-down for HD queries to which the displayed signals or expressions are related.

Behavior in the signal tree

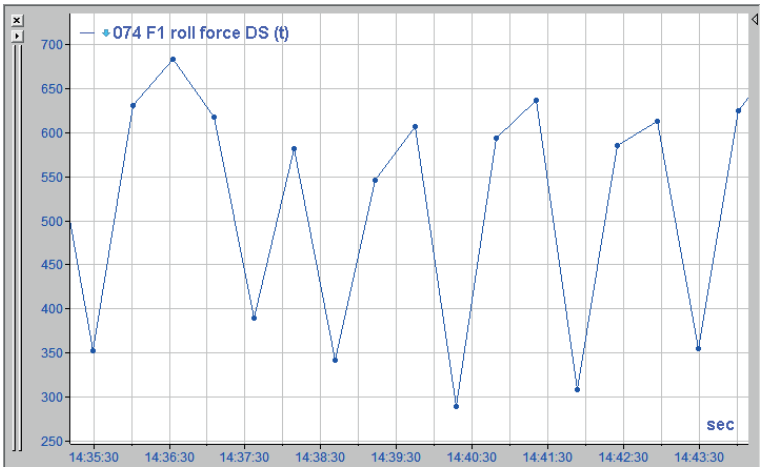
If you perform a drill-down, a modified HD query is executed and the original HD query in the signal tree is overwritten. The HD query generated by a drill-down has the following properties:

- Same HD server and same HD store
- Time range corresponding to the zoomed X-axis range
- A preferred timebase, calculated according to the following formula:

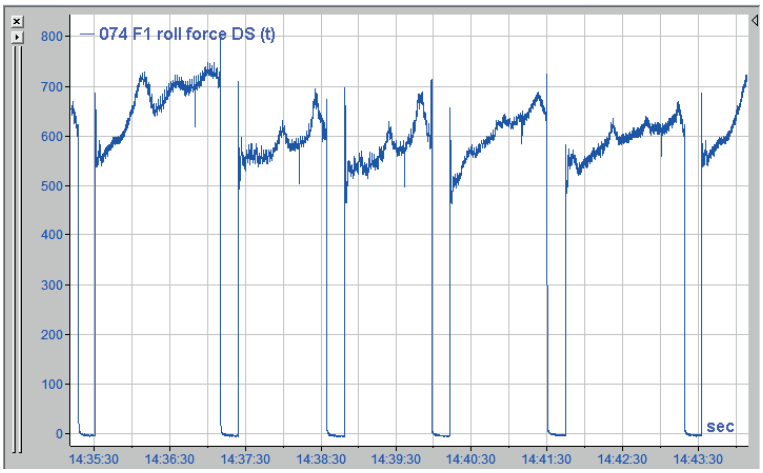
$$new\ preferred\ time\ base = \frac{zoomed\ time\ range\ length}{old\ queried\ time\ range\ length} \times old\ preferred\ time\ base$$

Example

An HD query over a time range of 2 days and 8 hours with a timebase of 30 s provides the following image after zooming in to a time range of only 8 min:



Only the execution of a drill-down provides sufficient information for a meaningful curve. The zoom factor remains unchanged.



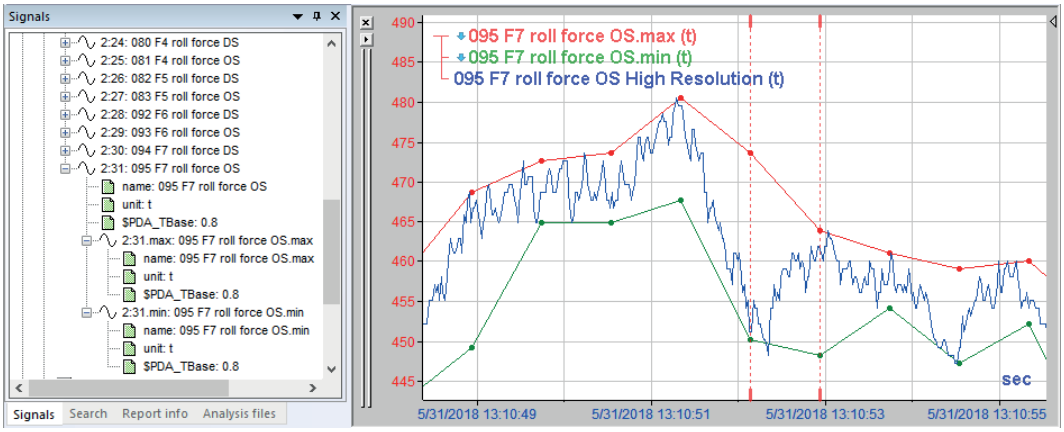
11.8 Subchannels min/max

For each HD query, the *min* and *max* subchannels are automatically loaded for each analog signal if the following options are activated in the *ibaAnalyzer* default settings under *HD Server*:

- *Also query aggregated minimum channel*
- *Also query aggregated maximum channel*

For more information on the settings for HD servers, see ↗ *HD server*, page 94.

In the following image, you can see the measured value and the subsignals for *max* and *min* in the signal tree. The graph shows the corresponding curves in red and green. The blue curve shows the high resolution data for comparison.

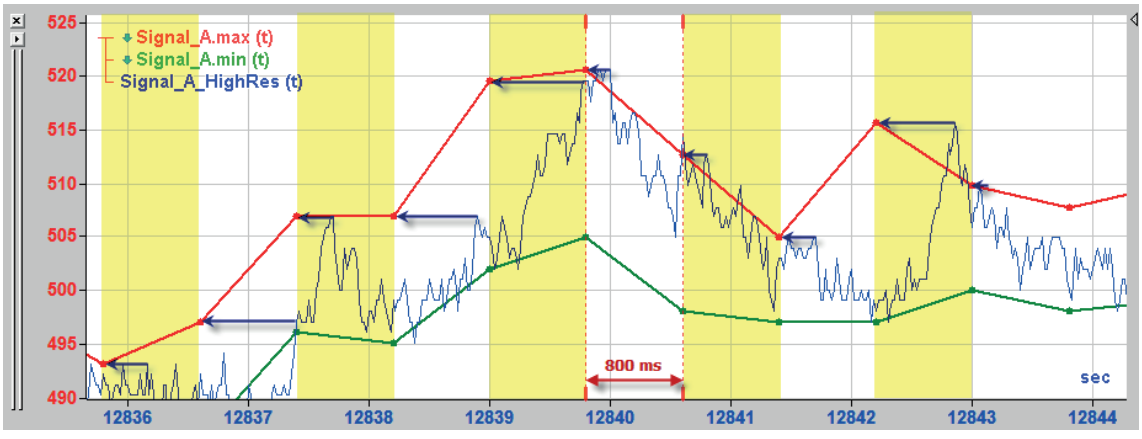


The calculation of the maximum and minimum values always relates to the determined time basis of the query.

This provides you with information about the maximum and minimum values that occurred in the time base intervals for the aggregation. Measured value outliers are therefore not lost.

Determination of minimum and maximum values

The following figure illustrates how the maximum values are determined, for example.



In this example, the preferred time base for the HD query is 800 ms. The aggregated values for the measured value (not shown here) as well as the maxima and minima are entered in this time grid.

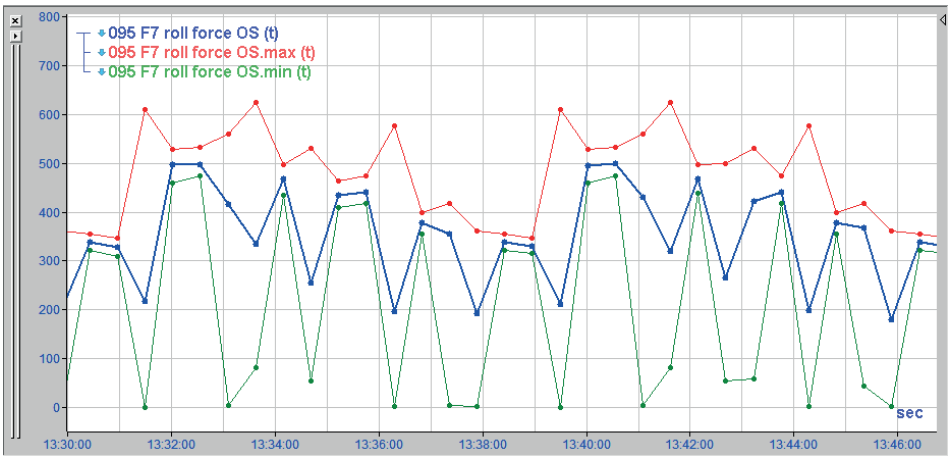
The blue curve in the background shows the measured values in the highest resolution as they would look after a drill-down. The maximum value of the blue curve from each 800 ms interval is drawn on the red curve.

Minimum and maximum in drill-down

If you perform a drill-down in a display with maximum and minimum signals, the curves get closer to each other until they are congruent at the smallest time base offered by HD recording.

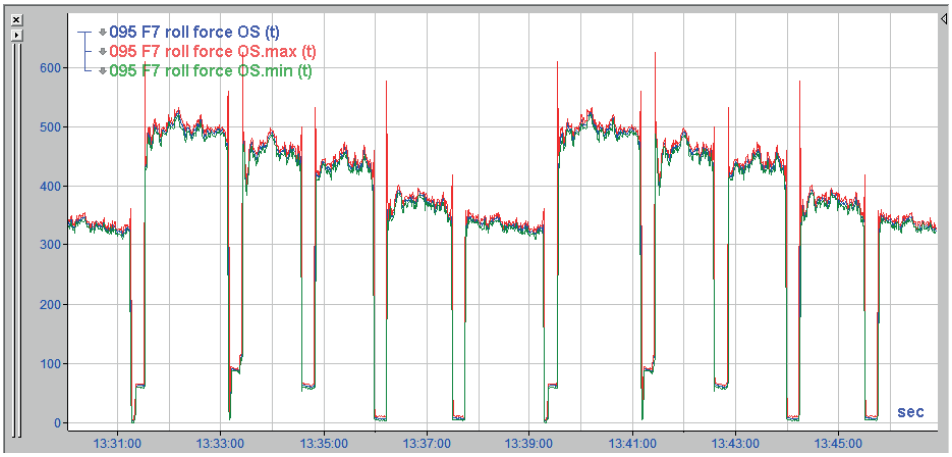
Example 1

Curves after zooming into an HD query with a timebase of about 30 s:



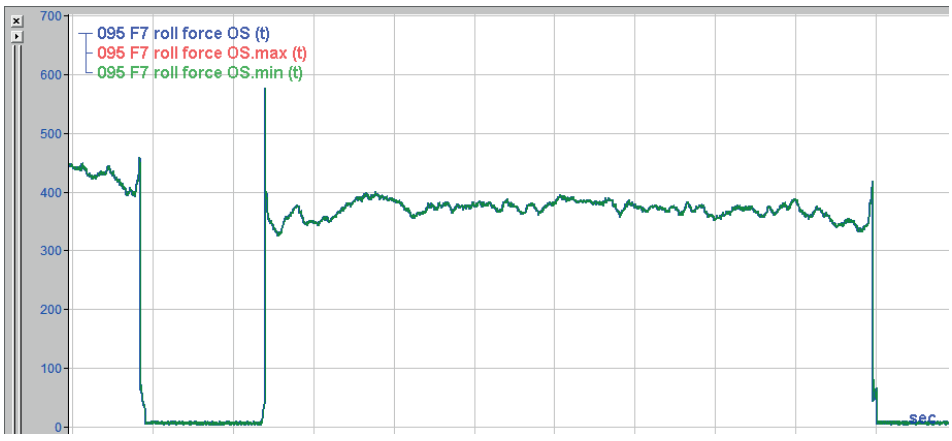
Example 2

Curves for the same time range after an initial drill-down.



Example 3

Curves after zooming in and drilling down again (highest resolution); In this example, only the green curve is visible because it is in the foreground and covers the others:

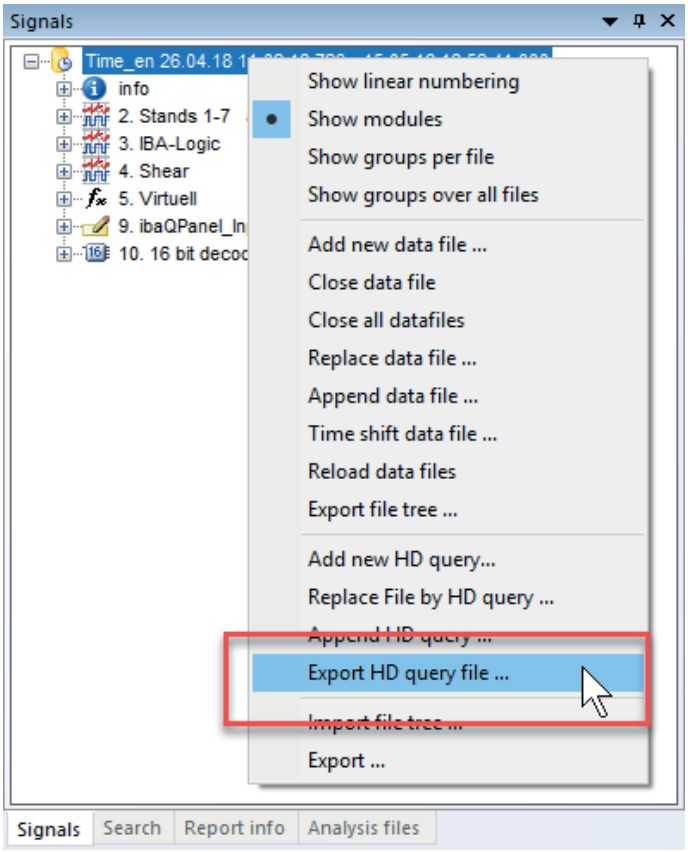


11.9 Export and import of HD queries and time periods

You can export and import the result of an HD query or time period query in file form. The procedure is identical for both types of queries.

Exporting an HD query or time period query

1. Once you have executed a query, click on *Export HD query file* in the context menu of the signal tree.



→ The *Save datafile* dialog opens.

2. Select the destination path.
3. Enter a file name.
4. Save the file with <OK>.

   The file is given the extension .hdq. You can open it like a data file.

   See ↗ *Opening the data files*, page 16.

Opening an HDQ file

Because the file only contains the query parameters and no measured values, a connection to the corresponding HD server needs to exist in order to open the HDQ file.

You can edit the exported query (HDQ file) with a text editor.

Example of an exported HD query:

Content	Description
`[HDQ file]`	Identification HD query
`port number=9180`	Port number
`server=HD-Computer`	HD server name
`starttime=31.10.2013 10:15:50.336000`	Start of the time period that is to be read out
`stoptime=31.10.2013 10:19:25.758000`	End of the time period that is to be read out
`store=HD store_Time`	HD store where the data is stored
`timebase=0.001`	Time base of the measured data
`type=time`	Time-based or length-based data

Instead of `starttime` or `stoptime`, you can also specify a `duration` in seconds. You can also specify the value "now" as the `stoptime`.

In this way, you can formulate queries for a consistent time range, e.g. for the last 7 days, always from the time range when the query is executed or the HDQ file is opened. Example of HD query for the last 7 days from "now":

HD-Abfrage letzte KW.hdq

```
1 [HDQ file]
2 portnumber=9180
3 server= HD-Computer
4 duration=604800
5 stoptime=now
6 store=HD-Ablage 1
7 timebase=60
8 type=time
```

Example of an exported time period query:

Content	Description
`[HDQ file]`	Identification time period query
`port number=9180`	Port number
`server=HD-Computer`	HD server name
`store=HD-Store 2 (Process data)`	HD store
`timebase=0.01`	Set timebase
`timePeriodId=116268`	Identification of the time period
`timePeriodName=HRM_Product_ID_20250303_00231`	Time period name
`timePeriodStoreName=TimePeriod_Product`	Name of the time period store
`type=time`	Time-based data

Exporting HD query to a standard data

If an HD query is loaded in *ibaAnalyzer*, you can use the usual export function (*File* menu – *Export…*) to generate a normal iba data with the extension .dat. This allows you to make an HD query available to users who do not have a connection to the HD server.

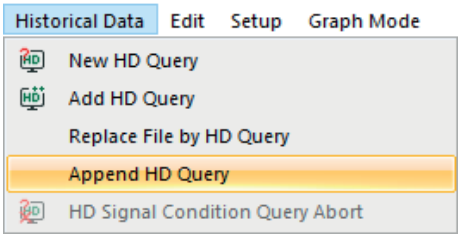
See ↗ *Data export and extraction, page 276*.

11.10 Appending an HD Query

Just like normal DAT files, you can also append HD queries or time period queries to HD queries, time period queries or DAT files that are already open in order to display longer periods of time.

It only makes sense to append to a DAT file if the signal structure of the DAT file and the HD record is the same.

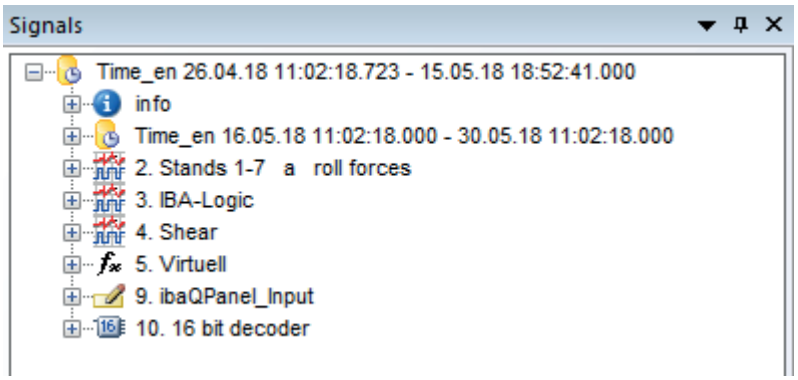
1. To append a query, select *Append HD Query* in the *Historical Data* menu or in the context menu of the signal tree.



→ The configuration dialog for the HD query opens.

2. Optionally, set a different query period and execute the query.

→ The new query result is nested in the signal tree.

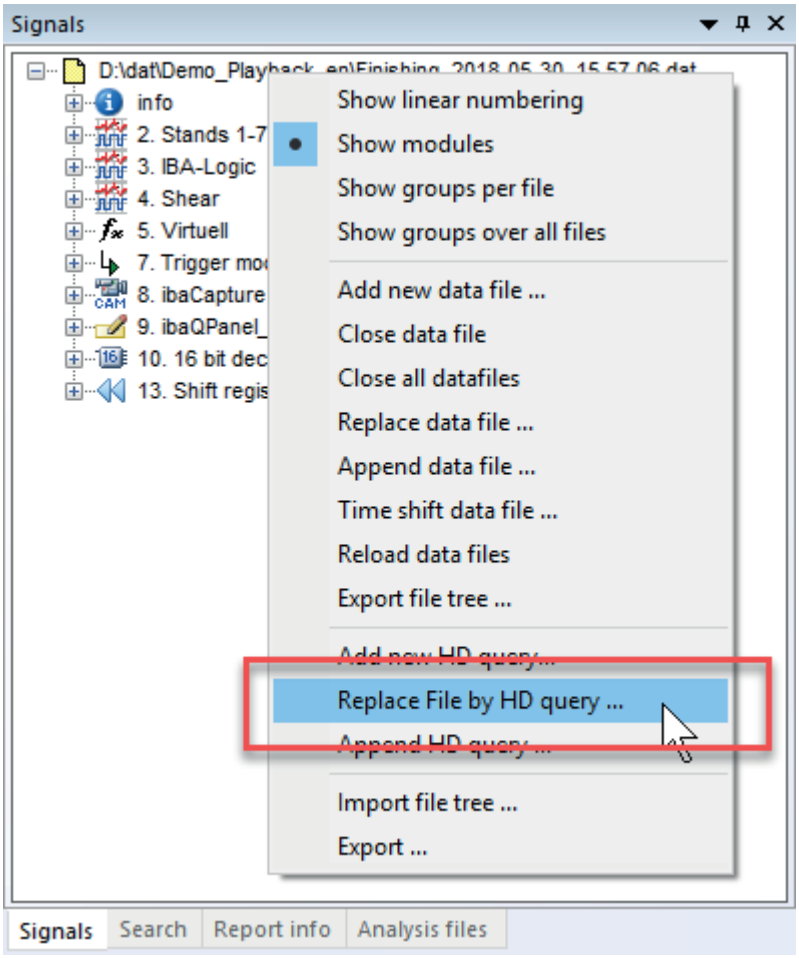


Note

To ensure a timely appearance in the graphs, make sure that the option *Synchronize files on recording time* is enabled in the X-axis settings.

11.11 Replacing a file by HD query

The *Replace File by HD query* command can be found both in the *Historical Data* menu and in the context menu of the signal tree.



When you execute this command, the configuration dialog for the HD query opens, where you can set a new time range and a new condition. After executing the query, the existing data file or HD query is overwritten by the new HD query result.

If there are several data files or HD queries in the signal tree (next to each other or appended), the replace command affects the selected file or query.

11.12 Automation of HD analyses

For regularly recurring analyses of HD data, e.g. to create day or shift reports, an automation of the HD query using *ibaDatCoordinator* is suitable.

In contrast to *ibaPDA* data recordings, where the execution of the *ibaDatCoordinator* jobs is triggered with the creation of new data files, this option is not available for the "endless" HD recordings. However, you have the option of defining scheduled jobs and tasks. This makes it possible to execute analyses independently of the creation of new data files.

You can use *ibaDatCoordinator* to define a scheduled job, i.e. a time-controlled job that executes an HD query cyclically. With the appropriate configuration of HD query and analysis (*.pdo), any evaluations and further steps such as report creation or database extraction can be automated.

Other documentation



Further information on configuring scheduled jobs can be found in the documentation for *ibaDatCoordinator*.

12 Analysis with ibaCapture videos

ibaAnalyzer provides access to video data saved with *ibaCapture*. *ibaCapture* records visual information synchronized with measured values based on *ibaPDA*. In *ibaAnalyzer*, you can view the video recordings together with the measurement data.

The image information is saved in separate video files. The storage location of the video data is configured in the *ibaCapture* module in *ibaPDA* and is included in the data file as a UNC path. When you open an *ibaCapture* module in *ibaAnalyzer*, the corresponding video file is loaded based on the path information in the data file. You can customize the path or server however.

There are also settings available for downward compatibility with the remote *ibaCapture-HMI* software. HMI recording is now integrated into *ibaCapture* as a virtual camera. The *ibaCapture-CAM* software is also part of *ibaCapture*.

Note

The *ibaCapture* Server, where the videos are stored, must be accessible for the computer with *ibaAnalyzer* via the network.

12.1 Showing video recordings

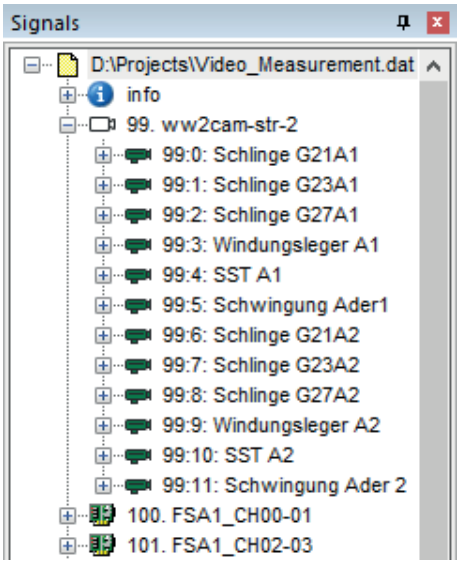
Once the data has been successfully recorded, *ibaAnalyzer* can perform an offline analysis.

Note

The *ibaCapture* Server, where the videos are stored, must be accessible for the computer with *ibaAnalyzer* via the network.

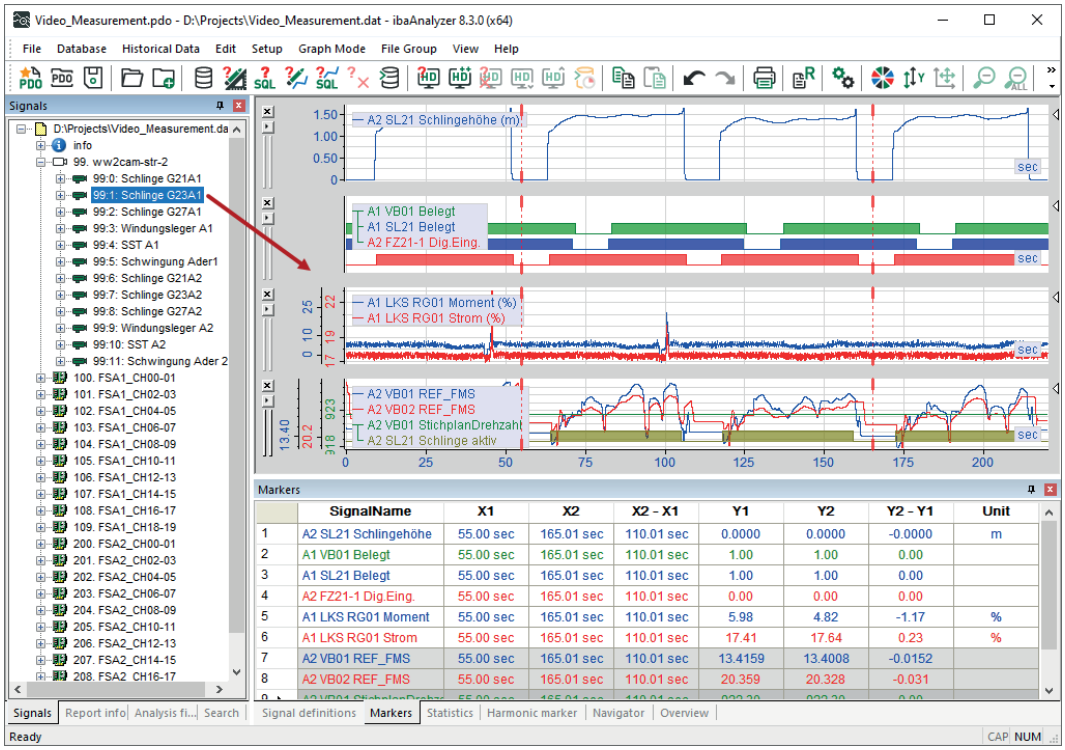
In the following, it is assumed that you have created an analysis and added signals to one or more graphs.

If a data file also contains *ibaCapture* video data, the signal tree also includes the *ibaCapture* modules in addition to the measured values.



Displaying a video

► To display a video, drag & drop the video signal (green camera icon) into the recorder window or double-click on a video signal.



→ A video player opens up playing the selected video.



Each video signal is opened in a separate window.

For more information on use of the video window, see ↗ *Functions in the video window, page 260*.

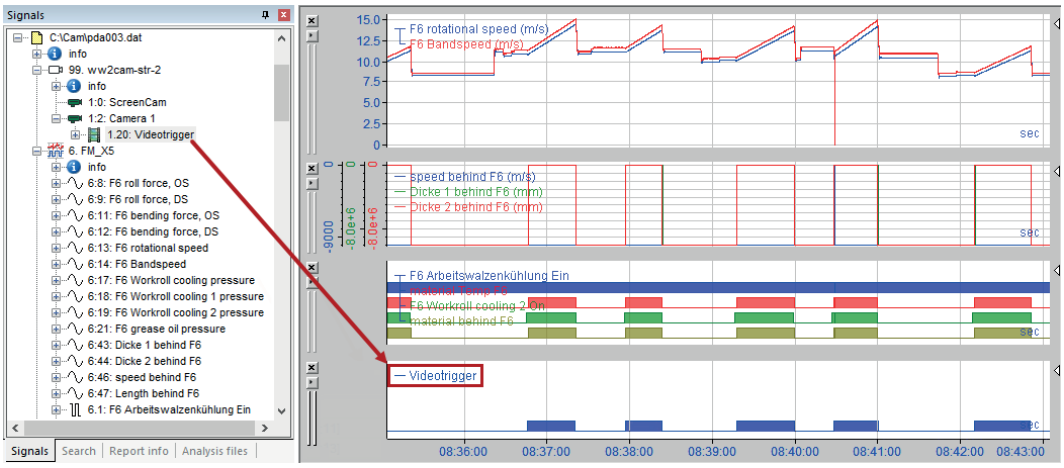
For more information on analyzing measurement data and video data, see ↗ *Analysis of measurement signals based on video recording, page 263*.

Displaying videos with trigger signal

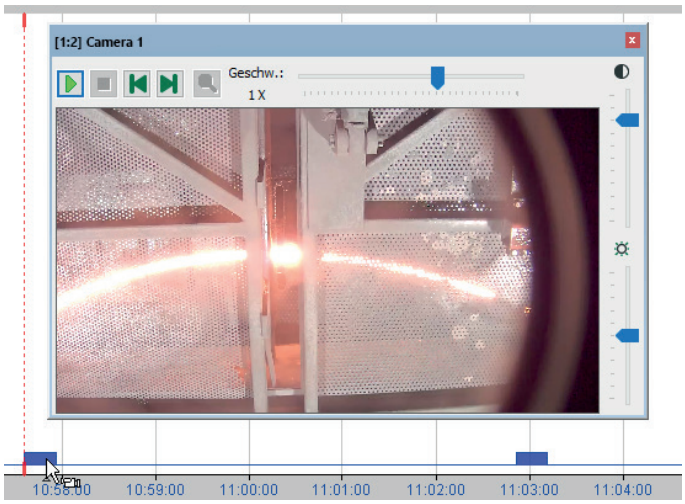
If videos have been recorded based on a trigger signal, you can display the signal and the video sequences.

1. To display triggered video sequences, drag & drop the video trigger into the recorder window or double-click on the trigger signal.

→ The trigger signal appears in a new graph.



2. To open the corresponding video sequence, double-click on the trigger signal.

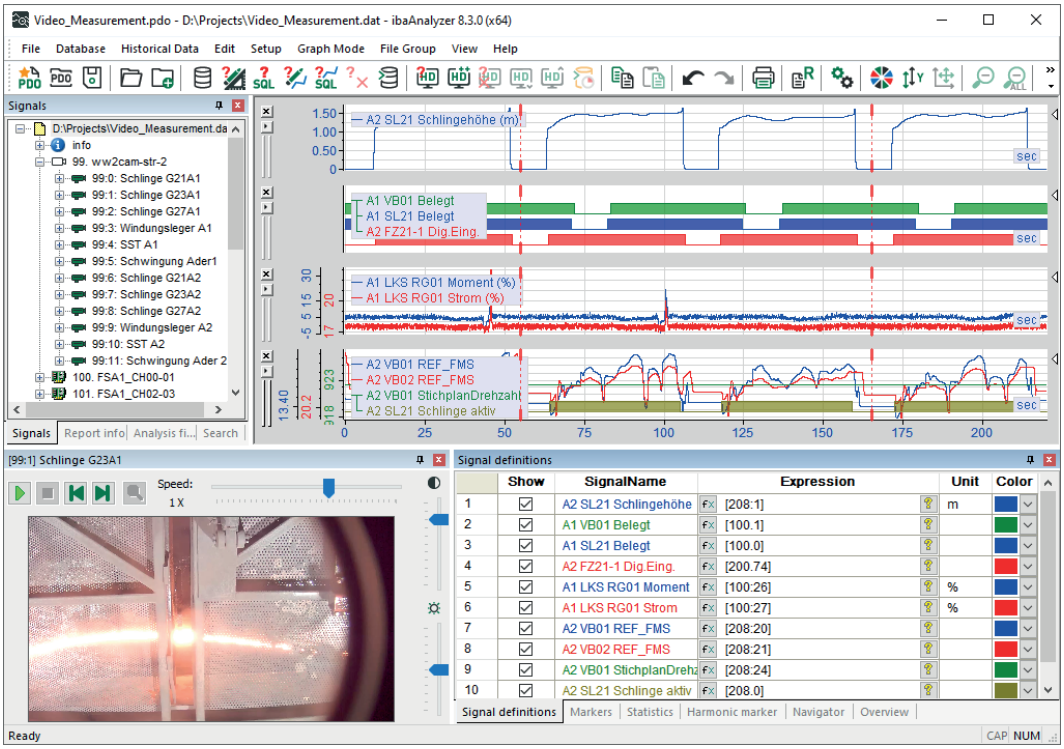


3. If video signals are open in addition to the trigger signal, you can change the video image by moving the X1 marker.

Positioning the video window

You can move the video window with the mouse and dock it at various points in the *ibaAnalyzer* window.

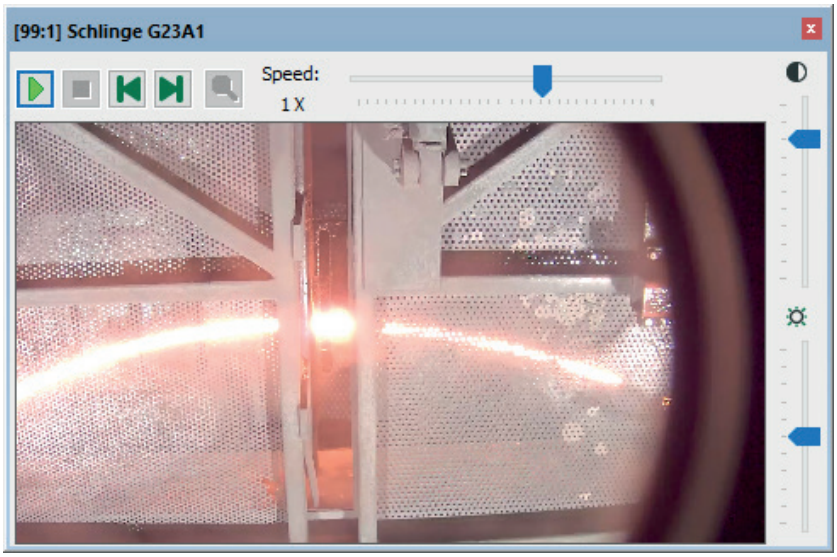
If you change the size of the video window, the image proportions remain the same.



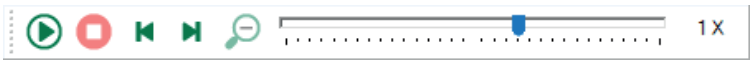
To undock the window, double-click on the window header or drag the header to a different position.

12.2 Functions in the video window

Each video signal is opened in a separate window. The video window features a toolbar and a context menu offering additional functions.



If the video controls are not visible in camera views, you can find them at the top of the *ibaAnalyzer* toolbar with the same functions.



You can show the toolbar in the video window via the context menu with the *Show video controls* command.

Video window toolbar

You can use the video window toolbar to control playback.

	Start/Play	Start playback
	Pause	Interrupt playback
	Stop	Stop playback
	Forward	One frame forward
	Back	One frame back
	Zoom out	Zoom out to the previous zoom level

Click on the Play button to start playback. If multiple video windows are open, the videos are played synchronously with the active video. If associated signal trends and the marker view are also open, the X1 marker shows the current position of the video in the signal trend.

To pause playback, click the Pause button or press the space bar.

Click on the <Stop> button to stop playback. The X1 marker jumps back to its original position.

If you hold down one of the buttons for longer than 1 second, the image is advanced at about 5 frames per second. This allows you to fast forward or rewind frame-by-frame.

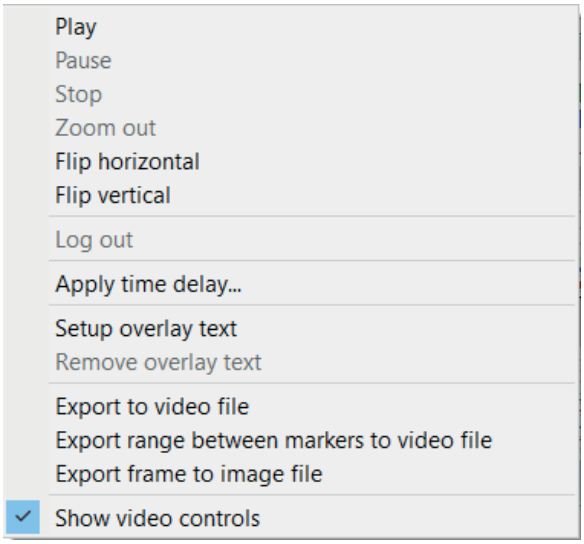
Sliders
You can use the slide controls on the right-hand side of the video window to adjust the brightness and contrast.

You can adjust the playback speed using the slider above the video. You can set the speed in the positive and negative directions. If the playback speed is negative, the video is played backwards.

You can also operate the sliders during playback.

Video window context menu

Right-click on the video window to open the following context menu:



The *Play*, *Pause*, *Stop* and *Zoom* menu items correspond to the toolbar functions described above.

Reverse horizontally/vertically
These commands mirror the video image in the respective direction. Click on the command again to cancel the mirroring.

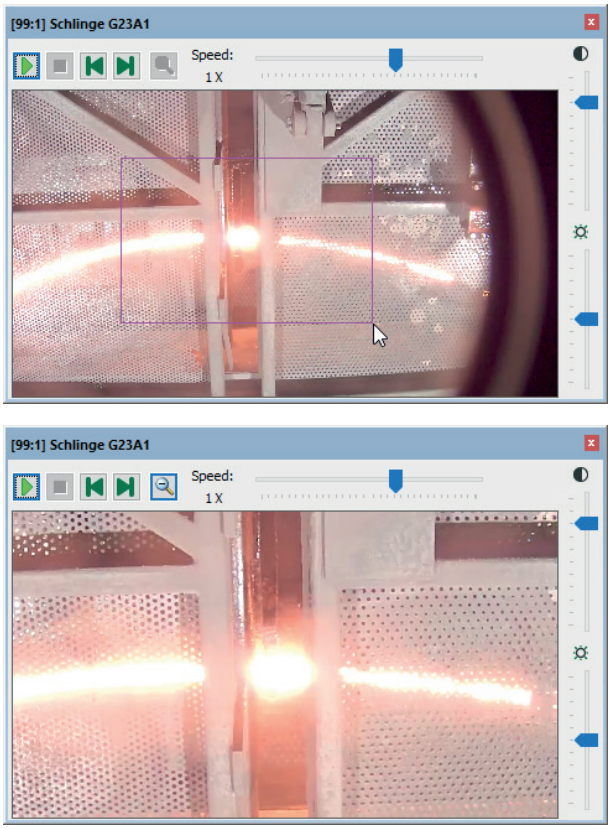
Apply time delay
Set the time in seconds by which you want to delay the video. The image then no longer matches the marker position. Instead, the video shows the image corresponding to seconds before.

Overlay text settings
For more information see ↗ *Text signals in videos*, page 217.

Export options
For more information see ↗ *Export of video data, page 266*.

Zoom

You can enlarge image content in the video window. To do this, either draw a frame around the detail in question with the mouse, or simply turn the mouse wheel while the cursor is positioned on the image.



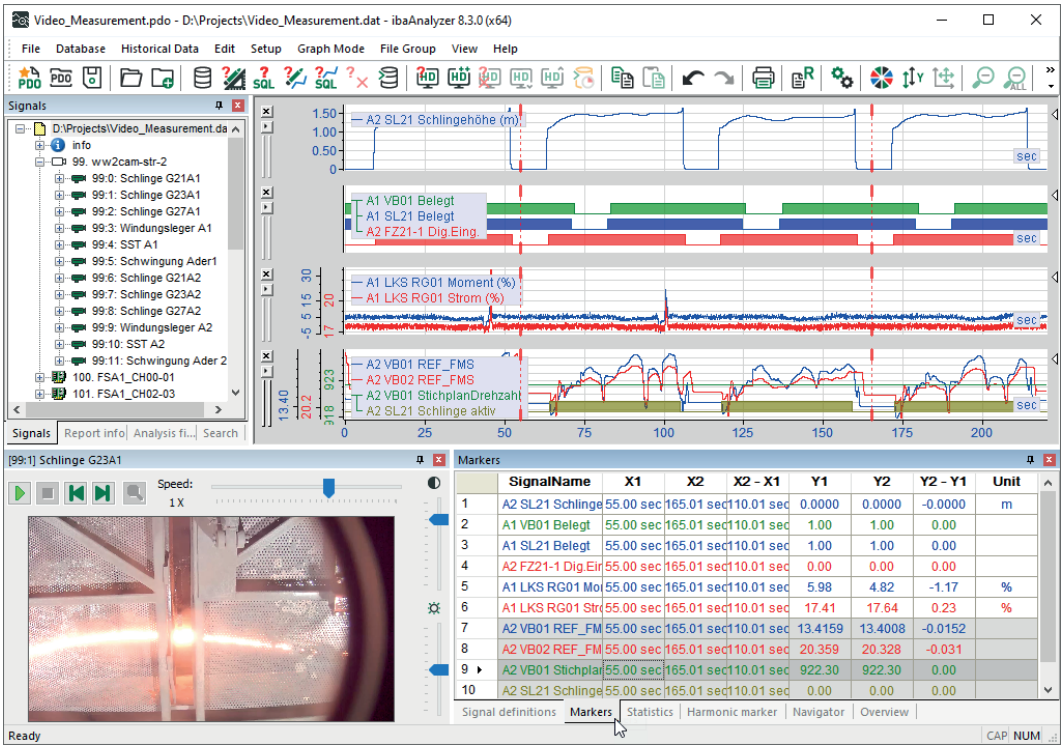
When zoomed in, you can pan the image detail by moving the mouse while keeping the <Alt> key pressed. The cursor changes to the index finger icon.

You can zoom out either with the mouse wheel or by clicking on the button .

12.3 Analysis of measurement signals based on video recording

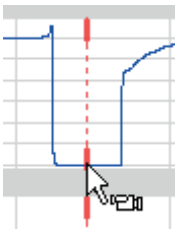
Once you have configured *ibaAnalyzer* with all the required measurement signals and video recordings, you can analyze the measurement data and view the measured values in relation to the video sequences.

Select the *Markers* tab in the signal table window to display the markers in the graphs.



Place the marker at any point in the graph. The marker shows the current measured values recorded in the respective video sequences. The video sequences are linked to the measured values by a common time base.

When you move the X1 marker, the video window displays the corresponding video sequences. Text channels that are visible in the camera view are also updated. The camera icon on the marker indicates the link between the measured values and the open video.



When you play the video sequence, the marker also runs through the measured values. You can use the <Pause> and <Stop> buttons to interrupt the film sequence, and thus also the sequence of measured values.

12.4 Playing protected videos

You can protect videos with *ibaCapture*. Only certain users can then play this protected video. *ibaAnalyzer* supports user management for *ibaCapture*. If *ibaCapture* user management is enabled, you must authenticate yourself as a user in order to view videos in *ibaAnalyzer*. This ensures that viewing rights are not infringed.

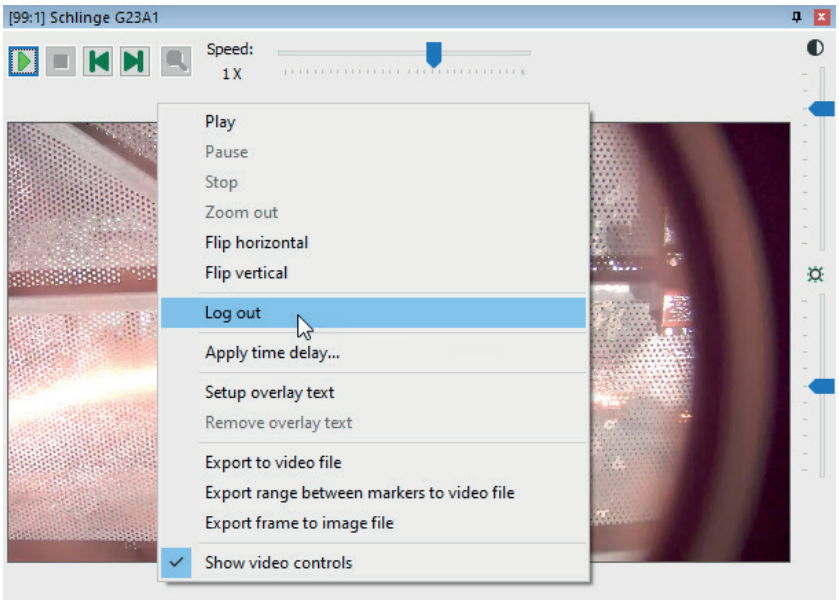
Analyses with protected camera views

In a protected camera view, an initial message informs you that access to the video is denied.

Click on <Log in as> and enter your user data.

If multiple camera views of an *ibaCapture* server are open, you only need to log in once. After successful authentication, all camera views are enabled according to the assigned rights.

You do not need to close all camera windows in order to restore protection. You can log out via the context menu.



Tip

You can also store your login credentials in encrypted form in the analysis. To do this, select the *Save password* option in the login dialog. The next time you save, your access data will also be saved so that video access rights are immediately available when you reopen the analysis file.

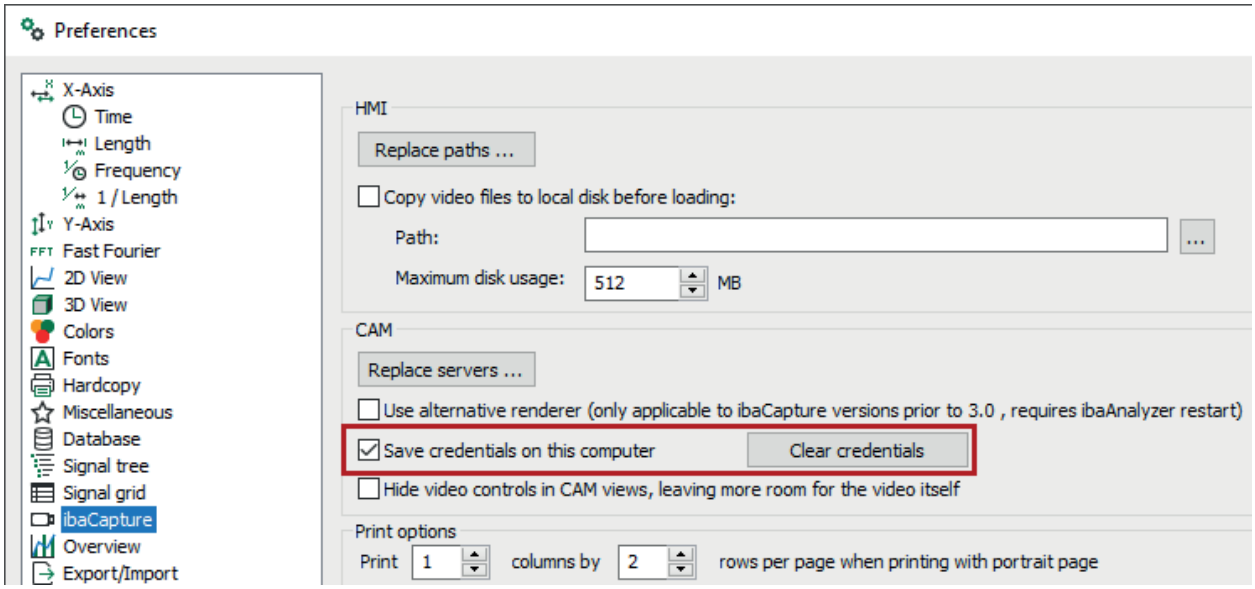
For security reasons, this save action only works on the computer on which the analysis file was created.

This function is helpful if you want to create automatic reports with still images from videos or video exports when user management is enabled.

Saving credentials for all analysis files

You can also save your *ibaCapture* credentials on the computer.

Open the Preferences, and under *ibaCapture* select the *Save credentials on this computer* option.



This assures access to protected videos on the computer regardless of the loaded analysis file.

You can use the <Clear credentials> button to remove the login data from the computer.

12.5 Export of video data

You have various options for exporting video data. You can use the *File – Export* menu and create a new iba data file (extension .dat). You can also export the video file via the context menu in the video window.

Export of an iba data file with video data

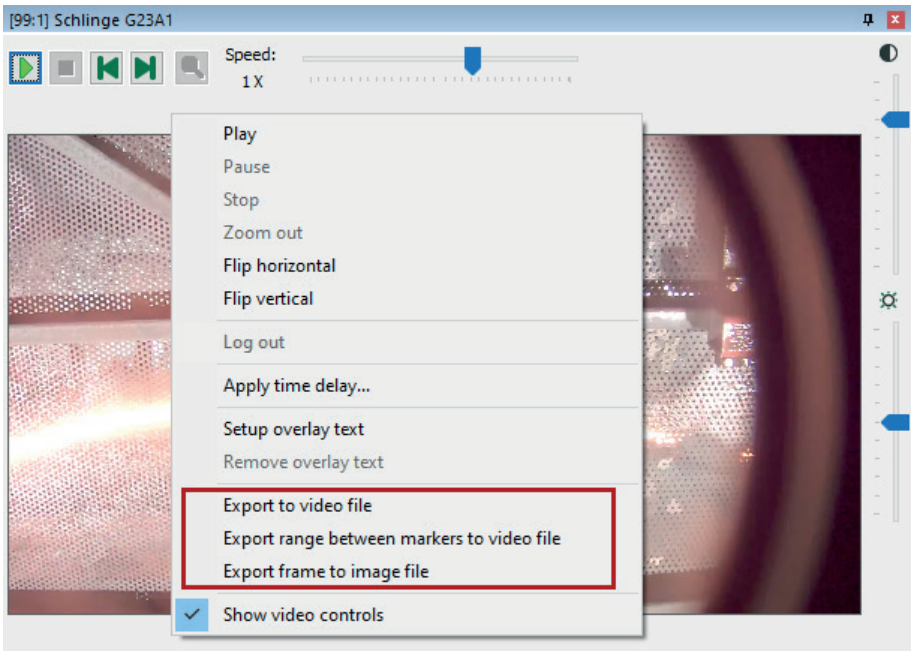
When exporting a DAT file, you can export the videos as part of the data file or as a separate video file. The relevant video sequences for the part of the data file you are exporting are compressed and saved in the new data file.

Continuous and triggered video recordings can be exported together in one data file.

For more information, see ↗ *Data export and extraction, page 276*.

Export of video files or still images

If you want to export a video sequence or single images without the measurement data, you can use the export commands in the context menu of the video window.



Export to video file
This command exports a video file corresponding to the complete time range of the data file. Enter the path and file name. The output format is MPEG-4 (.mp4).

Export range between markers to video file
To export only certain parts of the video rather than all of it, first set the markers in the recorder window to the relevant time range. Then use this command. This also reduces the size of the exported video file.

Export frame to image file
This command creates a snapshot of the current image – including during playback.

For a more precise selection, you are recommended to navigate to the desired position with the X1 marker and adjust the exact position with the buttons ⏮ ⏭ .

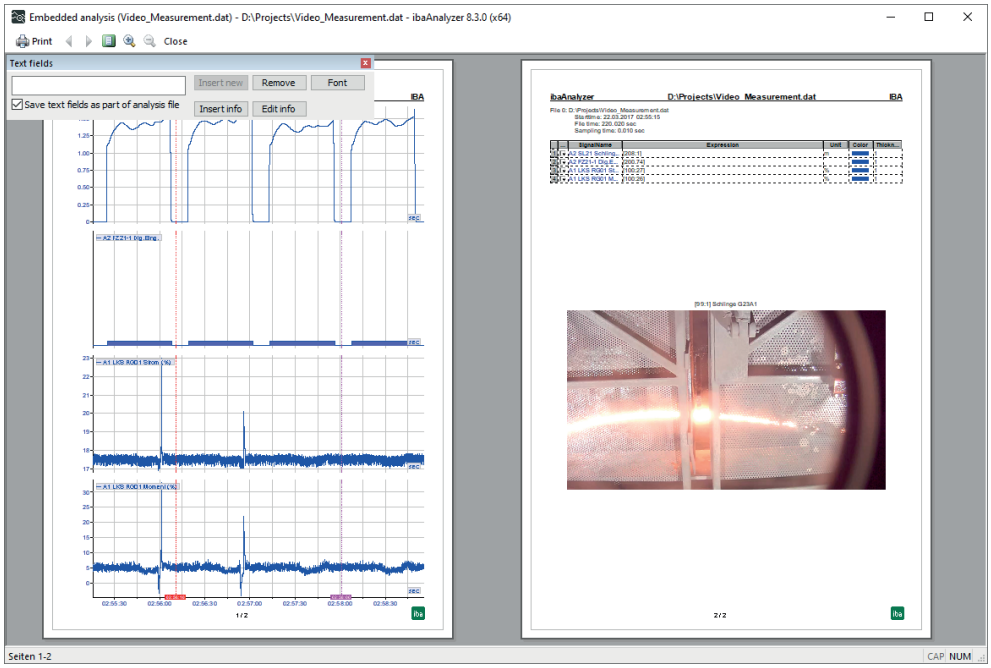
Enter the path and file name. Select the output format (BMP, JPEG, GIF, TIFF or PNG).

12.6 Printing and reporting of video images

You can print still images from the videos with the print function of *ibaAnalyzer* or add them to a report using the report generator.

12.6.1 Print with video images

You can print out all currently displayed video images using standard print function of *ibaAnalyzer*. The still images from the cameras are printed out after the signal table, as can be seen in the print preview below.



You can customize the layout in the *Preferences* dialog under *ibaCapture*, see ↗ *ibaCapture*, page 89.

12.6.2 Report with video images

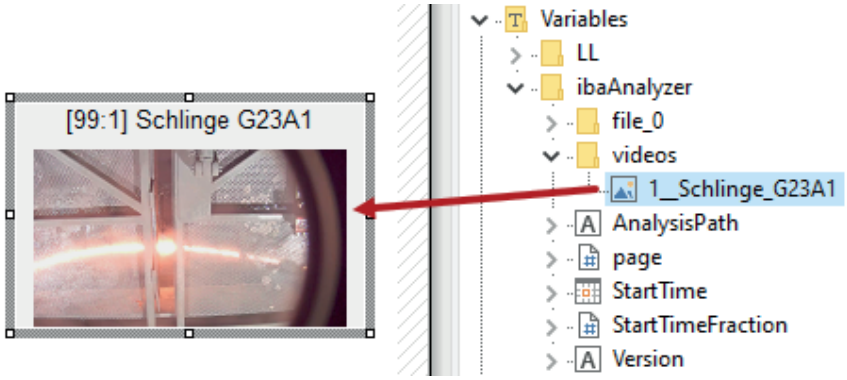
You can also use the report generator to present still images in a report.

Note

iba AG offers standard training courses for the *ibaAnalyzer-Reportgenerator*. You can find information on the training schedule our website, or you can contact your local iba representative.

ibaAnalyzer exports each visible camera view to the variables list of the report generator into the "videos" folder.

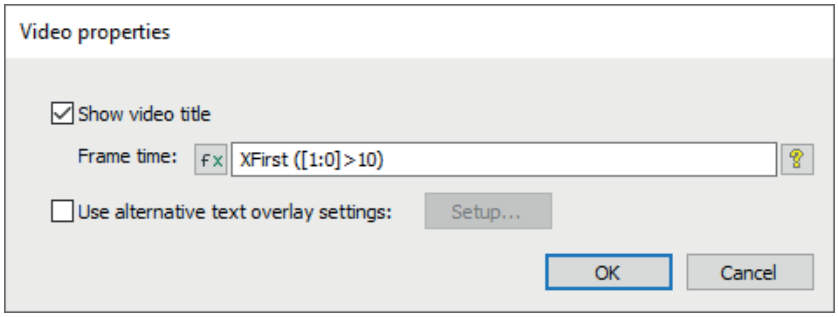
1. In the *Report Designer*, drag the desired camera view from the tree into the layout window.



2. Position and scale it in the same way as the normal curve views.

Changing the video properties

If you want to change the properties of the camera image, double-click on the image.



Show video title
If you enable this option, the name of the source signal is displayed as the video title.

Frame
Enter an expression or a format from the *ibaAnalyzer* expression builder to use the video image relevant to this time. The time is relative to the start time of the oldest data open file in *ibaAnalyzer*.

Use alternative text overlay settings
If the camera view has an overlaid text, you can change the text settings here.

13 Print function (Hardcopy)

In addition to the report generator, you can also print analyses directly.

The print function always prints the current displayed trend views and signal table. You can also print the *Markers* and *Statistics* tables.

ibaAnalyzer uses the standard Windows print function. This allows you to use the standard printers or PDF generators provided they are configured and available as printers under Windows.

13.1 Requirements and setup

Requirements

To use the print function, a printer must be installed and connected to the computer or network. At least one valid printer driver must be installed on the analysis computer, otherwise you will not be able to use the print preview.

Setup

You can access the printer settings via the *File – Print setup* menu. The dialog includes the standard Windows settings for printer setup.

You configure the printout itself in the Preferences under *Hardcopy*, see ↗ *Hardcopy*, page 81.

13.2 Print preview settings

You can use the print preview to check the image before printing. Using the print preview, you can also insert and format additional information as objects in the print image, such as comments or file information.

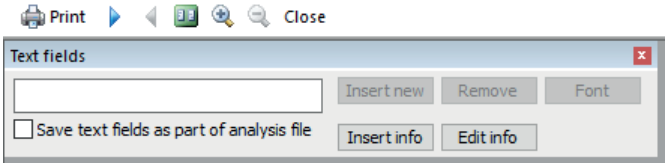
Note

The print preview only works if at least one printer driver is installed locally or in the network.

You can access the print preview via the *File – Printer preview* menu.

Print preview dialog

The print preview offers the following functions.



<Print>
Opens the printer dialog and starts printing.

Next page/Previous page
These buttons are only enabled when more pages exist than are shown in the preview.

One page/Two pages
Switch between one-page view and two-page view in the print preview.

Zoom in
Slightly enlarge the displayed page (one zoom level)

Zoom out
Reduce the size of the displayed page until one or two pages fit within the window.

<Close>
Exit the print preview

Text field
Enter additional comments in this field. You can position the text freely on the page. Click <Insert new> to include the text and enter further comments.

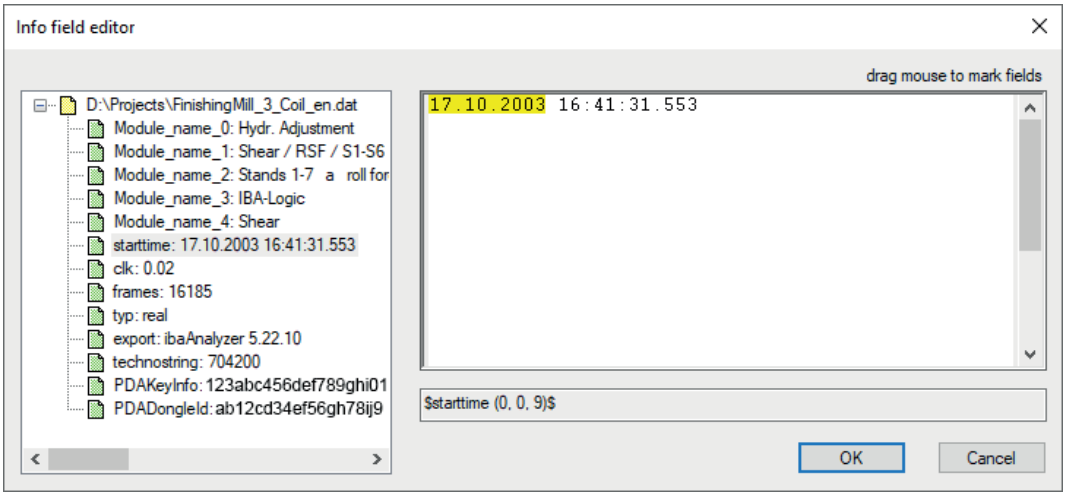
<Insert new>
Enter the line of text on the page.
You can position the text freely on the page.

<Remove>
Delete a selected object on the page (text and info only).

<Font>
Set the font for a selected object (text and info only).

<Insert info>
You can insert information from the info fields as text in the data file.

Select the info field and mark the desired character range with the mouse – in the example only the date from the start time. Confirm your selection with <OK>. You can position the text freely on the page.



<Edit info>
Edit information that has already been added. Select the info field and mark the desired character range with the mouse.

Save text fields as part of analysis file
If you want to regularly apply the inserted texts, comments and information to any data files, enable this option.

This option is particularly important for automatic logging. All additions are saved in the analysis file and are available again when the analysis is reused. To do this, save the analysis when all settings have been made.

14 Using views in other programs

You can use the tables and graphs from *ibaAnalyzer* as objects in other programs. This gives you an easy way to use the graphs and tables in process or fault analyses.

You can copy the content to the clipboard, see ↗ *Exchange of tables and graphs via the clipboard, page 272*. When you do so, tables remain editable as HTML objects, and graphs are copied as graphic objects. However, the copied tables and graphs are not embedded OLE objects that change when changes are made to the *ibaAnalyzer* source.

You can also save individual graphs as an image file or copy them to the clipboard, see ↗ *Exchange of individual graphs as image files, page 275*.

14.1 Exchange of tables and graphs via the clipboard

You can copy graphs and visible tables via the Windows clipboard and paste them into other programs to use them there. You have several options for doing this:

- Use the *Edit – Copy* menu to copy all graphs and visible tables to the clipboard.
- Select the *Copy* command from the context menu in a graph to copy all graphs and visible tables to the clipboard.

You can then paste the objects as usual into other programs, such as Microsoft Word. When you do so, tables remain editable as HTML objects, and graphs are copied as graphic objects. You can edit the tables and graphic objects further, such as by adjusting their formatting or scaling.

Note that all signals and expressions in the signal table are copied, including those that are hidden in *ibaAnalyzer*.

Copyable tables

You can copy all tables from the tabs.

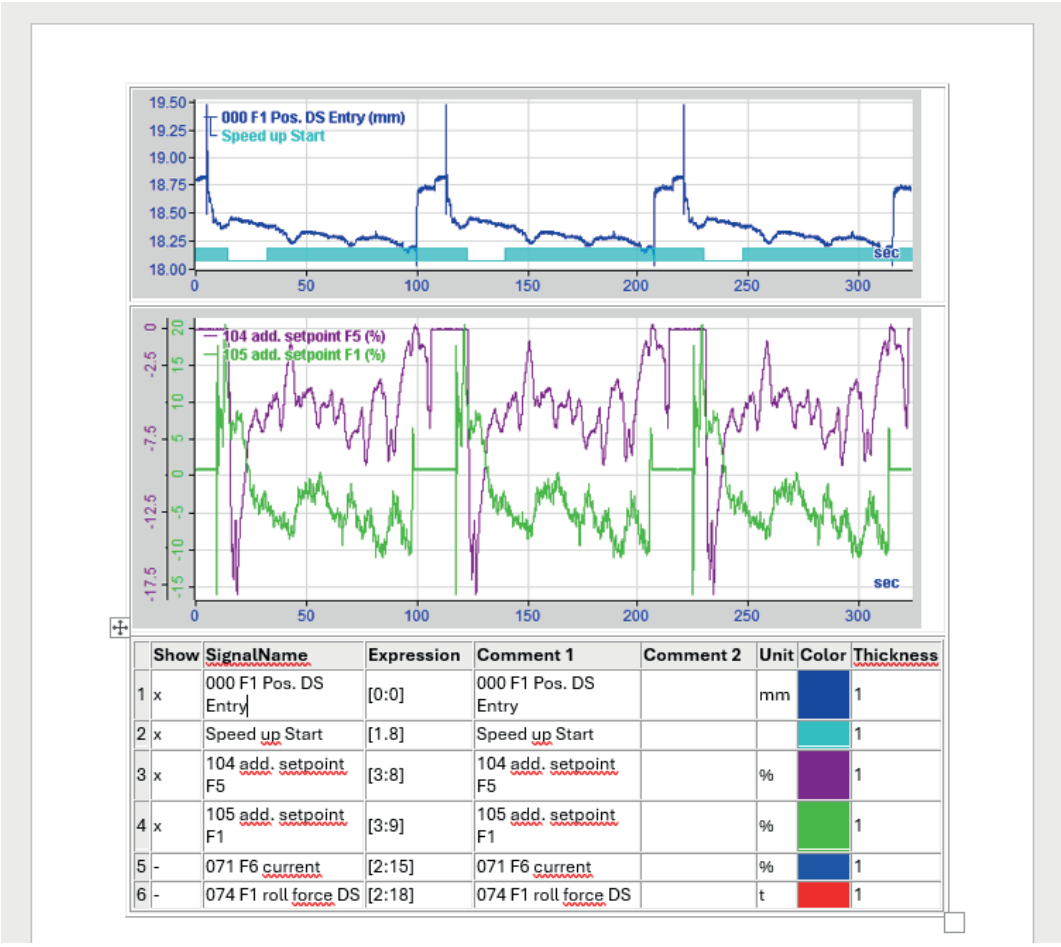
- Signal tables including the signal definitions
- Statistics
- Markers
- Harmonic markers

If you arrange these tables as separate windows in the user interface, they are also copied to the clipboard as separate objects.

The navigator view and the trend overview is not copied to the clipboard.

Example from Microsoft Word

The following example shows the inserted objects in a Word document.



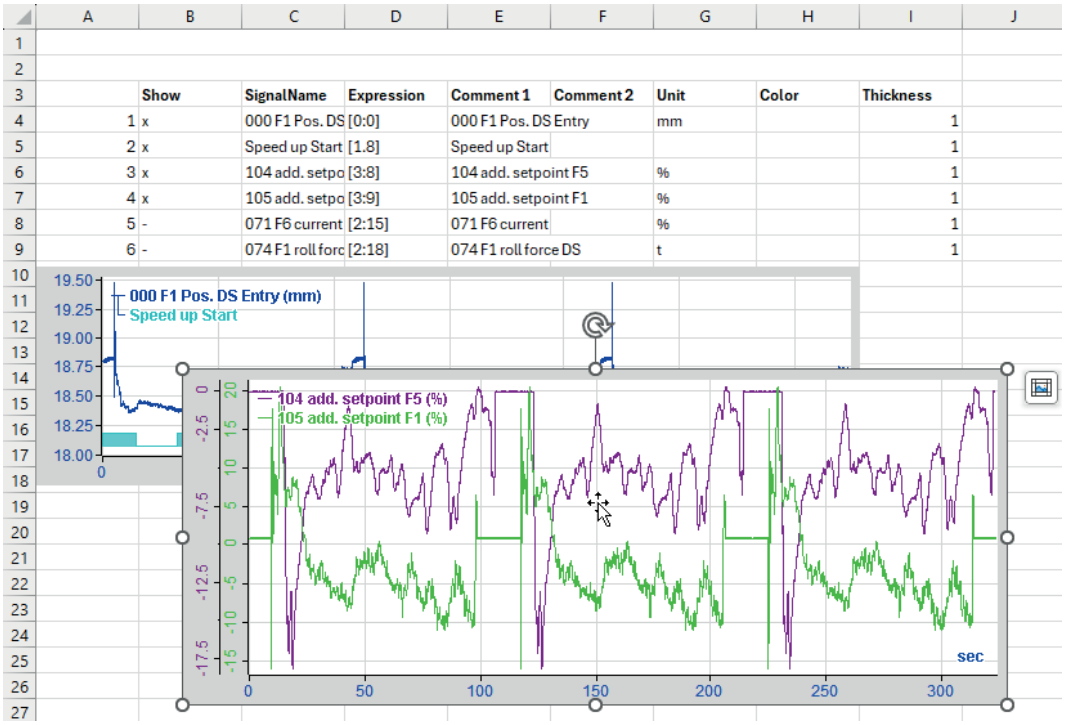
	Show	SignalName	Expression	Comment 1	Comment 2	Unit	Color	Thickness
1	x	000 F1 Pos. DS Entry	[0:0]	000 F1 Pos. DS Entry		mm		1
2	x	Speed up Start	[1.8]	Speed up Start				1
3	x	104 add. setpoint F5	[3:8]	104 add. setpoint F5		%		1
4	x	105 add. setpoint F1	[3:9]	105 add. setpoint F1		%		1
5	-	071 F6 current	[2:15]	071 F6 current		%		1
6	-	074 F1 roll force DS	[2:18]	074 F1 roll force DS		t		1

The graphs are inserted as graphic objects. You can select and move the graphic objects, rescale them, or change their properties in the same way as with other graphics.

You can customize and format the signal table using the standard Microsoft Word table functions (borders, shading, orientation, etc.).

Example from Microsoft Excel

The following example shows the inserted objects in an Excel worksheet.



The cells in the signal table are copied directly as table cells. The graphs are inserted as graphic objects.

14.2 Exchange of individual graphs as image files

You can save individual graphs in *ibaAnalyzer* as an image file or copy them to the clipboard.

The context menu in a graph offers you two commands for this: *Export graph image to file* and *Export graph image to clipboard*.

Note that these commands only apply to the graph on which you have opened the context menu. To export multiple graphs, repeat the process for each graph, or use the copy function for all graphs and visible tables, see ↗ *Exchange of tables and graphs via the clipboard, page 272*.

Exporting individual graphs as graphic files

1. Select the *Export graph image to file* command in the context menu of the desired graph.

→ A dialog for exporting to an image file opens up.

2. Use the browser button <…> to select a path.
   a) Enter a file name in the browser dialog.
   b) Select the file type for the graphic. You can choose from the following file formats: BMP, JPEG, GIF, TIFF and PNG.
   c) Confirm your entries with <Save>.
3. Optionally: Adjust the image size by altering the number of pixels. Only the graph size is changed when you do this; the labeling is not distorted.

   Enable the *Maintain aspect ratio* option to prevent the image from being distorted.
4. Click <OK> to export the graph as an image.

Copying individual graphs to the clipboard

1. Select the *Export graph image to clipboard* command in the context menu of the desired graph.

→ A dialog with settings for exporting to the clipboard opens up.

2. Optionally: Adjust the image size by altering the number of pixels. Only the graph size is changed when you do this; the labeling is not distorted.

   Enable the *Maintain aspect ratio* option to prevent the image from being distorted.
3. Click <OK> to copy the graph to the clipboard.

→ You can insert the graph as an image in other programs.

15 Data export and extraction

The measurement data that you view in *ibaAnalyzer* can also be output in various file formats. You can use the export function to document the analysis or pass it on to third parties who do not work with *ibaAnalyzer*.

The file output can include all visible original signals, logical expressions, and virtual signals from *ibaAnalyzer*. If the original data file contains video data from *ibaCapture*, you can also export video sequences to the new file or as a separate video file.

You can either export the files or extract them. The export function differs from the extraction function primarily in terms of the customization options.

Export function	Extraction function
■ only export to files possible ■ only for time-related data ■ limited data selection (e.g. for info fields) ■ simple selection of the time range of the measurement data ■ cannot be automated with *ibaDatCoordinator*	■ extraction in files and databases possible ■ for time and length-related data ■ detailed selection and customization of data (info fields, calculated columns, etc.) ■ selection of the time range of the measurement data using logical expressions ■ can be automated with *ibaDatCoordinator*
see ↗ *Exporting data, page 286*	see ↗ *Extracting files, page 291*

Use and advantages of the supported file formats

If further analysis is also executed with *ibaAnalyzer*, you can output the data to an iba DAT file. You can embed the analysis in the new DAT file to show the desired display when opening it. *ibaAnalyzer* treats exported or extracted data files like regular data files.

A generally readable format in other programs is the text file. Other programs can import the output ASCII file from *ibaAnalyzer*, e.g. for table processing in Microsoft Excel, for analysis in MatLab, for word processing in Microsoft Word or for use in a database such as MS Access.

ibaAnalyzer also supports output to column-oriented formats such as Apache Parquet or Matlab format (.mat).

For special evaluations, e.g. in energy technology, output to COMTRADE format is also possible.

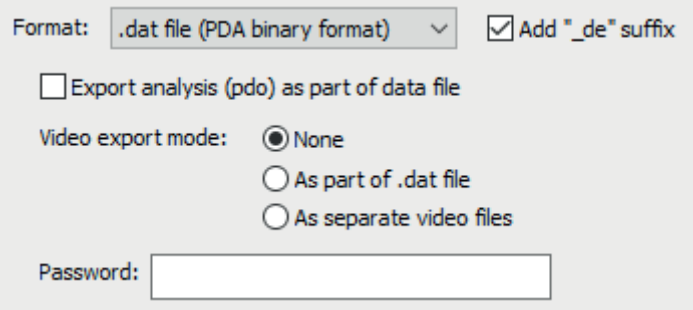
15.1 Supported file formats

You can export and extract measurement data into different file formats.

Some settings for the formats are only available for file extraction, such as the "de" suffix, see ↗ *Extracting files, page 291*.

15.1.1 iba format (DAT file)

If you select the format *.dat file (PDA binary format)*, you can create new and modified data files.



If the option *Export analysis (pdo) as part of data file* is selected, the exported data file contains the analysis. When you open this new file in *ibaAnalyzer*, you can decide whether to import the embedded analysis.

Note

All extracted data is converted to Real format. This can cause a loss of precision for some data types.

You can also create password-protected files. The password specified in this dialog is used for all exported files.

Video export mode
If necessary, you can additionally embed available video data in the exported data files. You have the following options for this:

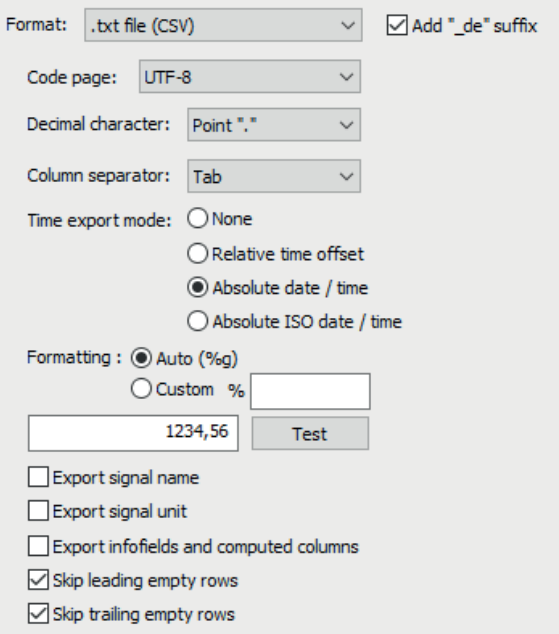
- *None*
  No video data is exported.
- *As part of .dat file*
  ibaCapture videos can be exported as part of a new data file. All videos that are selected in the signal tree are exported. You can then play back the exported data file without a connection to the *ibaCapture* server.
- *As separate video files*
  ibaCapture videos are exported as separate MP4 files to the same directory as the data files. Although 2 separate files are created, opening the resulting data file automatically calls up the associated video file. The filename suffix "_nn" indicates the individual pairs.

Note

The *ibaCapture* Server, where the videos are stored, must be accessible for the computer with *ibaAnalyzer* via the network.

15.1.2 Text file (CSV)

If you select the *.txt file (CSV)* format, you can export plain-text files.



ibaAnalyzer automatically generates delimiter-based text files. This means that the signal data in the columns and the columns themselves are separated by a specified separator character in the extracted file. You can configure the output files by various settings.

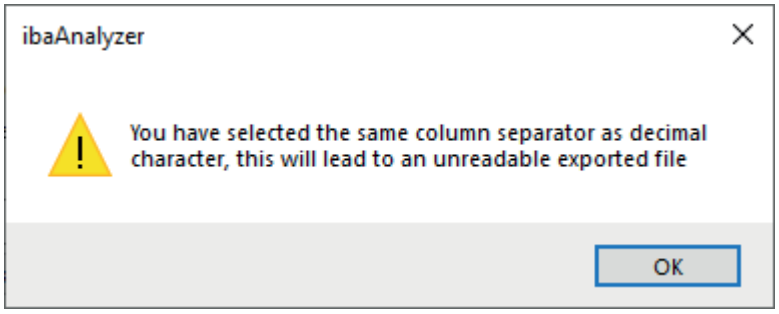
Code page
You can select different coding variants for the output file. You are recommended to use UTF-8 by default.

Decimal character
Select whether you want to use a comma or a point as the decimal separator for the exported values. The *System* option corresponds to the current standard of your operating system.

Column separator
By default, tabs separate the columns. You can also set other separators, for example to create CSV files (with comma-separated values).

Note

If you set *Comma* as the decimal separator and column separator, the output files will not be usable. *ibaAnalyzer* displays a warning if you have set that combination.



Time export mode
Select whether you want time data to be exported. If you do, select the format. The time stamp then appears in a separate column in the exported file.

- *None*
  No time stamp is exported.
- *Relative time offset*
  Begins with "0" and counts the seconds from the file start.
- *Absolute date/time*
  Displays the absolute date/time.

Time	[18:1]	[18:3]	[18:5]
30.01.2017 00:11:55.120000	0	217.272	190.276
30.01.2017 00:11:55.220000	0	238.018	196.484
30.01.2017 00:11:55.320000	0	219.696	182.417
30.01.2017 00:11:55.420000	0	226.913	182.002

- *Absolute ISO date/time*
  The date and time are extracted in ISO 8601 format.
  `YYYY-MM-DDThh:mm:ss.ffffff±hh:mm`
  The letter "T" separates the date and time, the fractions of a second are given in six digits and for the time zone +00:00 (UTC), "±00:00" is replaced by the letter "Z". The time zone information is taken from the info field $PDA_UtcOffset. If that info field is not included in the source file, no time zone information is exported.

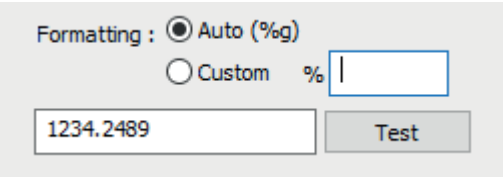
Time	[18:1]	[18:3]	[18:5]
2017-01-30T00:11:55.120000+01:00	0	225,451	186,9
2017-01-30T00:11:56.120000+01:00	0	232,631	184,234
2017-01-30T00:11:57.120000+01:00	0	238,449	187,122
2017-01-30T00:11:58.120000+01:00	0	235,515	182,911

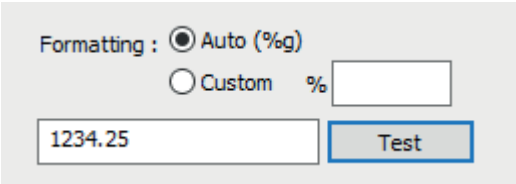
Formatting
You can also specify the number of digits in a number when writing floating point values. The syntax is based on the printf statement in C++.

The *Auto* option uses the *%g* format. Under *Custom* you can use different variants. Use a suitable reference for the printf statement in C++.

To test the behavior, there is an additional test field in which the input is formatted appropriately.

Enter a value and click on <Test> (image on left). The formatted value is then displayed (image on right).





Export signal name, Export signal unit
If you enable these options, additional lines are inserted in the header of the output file. The lines contain the name and unit of the measured values.

Export infofields (export only)
If you enable this option, text signals created in *ibaPDA* as an info field in the data file are also exported (via the data recording configuration, *Data storage – Files – Info fields*).

Other info fields from the data file, such as internal, automatically generated info fields, are not exported.

You can also write other exportable info fields to the data file with other applications, such as the data extract in *ibaAnalyzer*, *ibaFiles*, or *ibaDatCoordinator*.

Export infofields and computed columns (extraction only)
This option enables unpacking of all signals that were created in the *Infofields* and *Computed columns* tabs of the data extractor window. The data is written to a separate area of the file with a key-value structure.

Note

Text signals in the source file are treated like all other source signals.

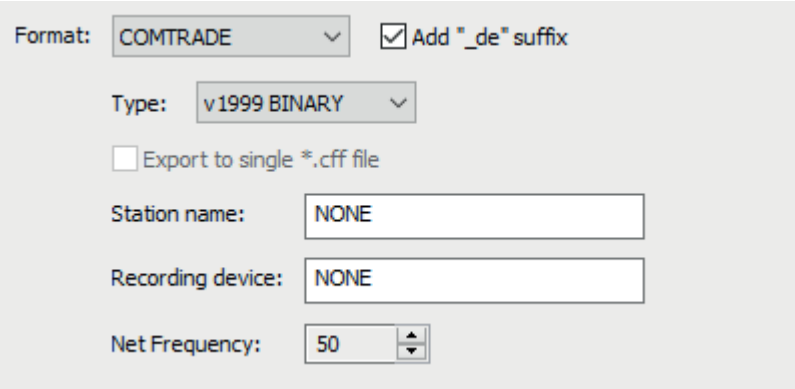
Skip leading empty rows, Skip trailing empty rows (extraction only)
Normally, the exported data begins with the first line at the start time and ends with the last data corresponding to the end time of the file.

If not all data was extracted (such as because XMarkValid was used), this can lead to the result containing complete lines with "NaN".

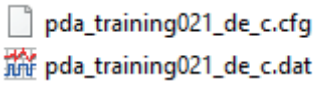
If you enable these options, such empty lines will be automatically removed from the resulting text file.

15.1.3 COMTRADE

The *IEEE Standard Common Format for Transient Data Exchange for Power Systems* (short COMTRADE) defines a particular format for the exchange of data files as documented in the IEEE Std C37.111-1999 standard. The standardization applies to both the format of the files and the type of media to be used for exchanging fault signal, test data or simulation data of energy supply systems.



During the COMTRADE export, *ibaAnalyzer* generates a DAT file with the measured values and a CFG file with the configuration data such as channel information (signal number, signal name, info columns), start time, end time, etc.



Type
ibaAnalyzer supports different types of this format, which you can select via the drop-down list.

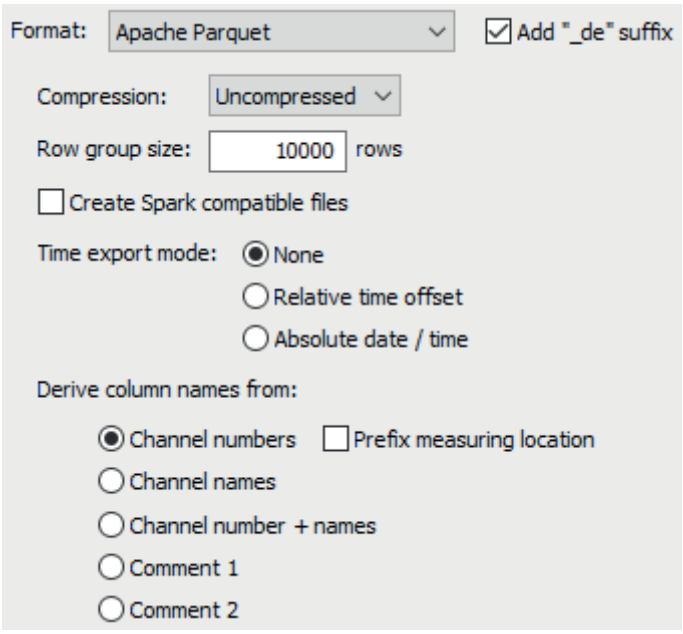
Export to single *.cff file
For all file types after v1999, you can also export to a single CFF file.

Station name and Recording device
According to the COMTRADE convention, the information regarding Station name and Recording device must be added to the file. This is done by entering the information in the corresponding fields. This information is stored in a CFG file that *ibaAnalyzer* creates during the export process in addition to the DAT file.

Net Frequency
Set the appropriate main frequency (50 Hz or 60 Hz) here.

15.1.4 Apache Parquet

Apache Parquet is a column-oriented, binary data format, which provides efficient data compression and different encodings. Due to its columnar structure and the possibility to add meta data to the file, it resembles the iba data file format. Also due to the comparable storage size, it is recommended to use these files as an interchange format to external systems.



The data in the output file is structured as follows:

- A channel (or expression) corresponds to a Parquet column
- The module structure available in iba data files has no direct pendant in Parquet and is therefore mapped using meta data (see below).
- All info fields are stored as Parquet meta data.

Note



All extracted (numerical) data is converted to the FLOAT Parquet data type. This can cause a loss of precision for some data types. STRING and BOOL receive the same corresponding Parquet data type.

Settings

Compression
The Apache Parquet format offers different compression methods. *ibaAnalyzer* supports plain encoding (Uncompressed), Snappy, Gzip, Brotli, LZ4, and ZStandard.

Row group size
The Apache Parquet format has an additional row-wise structure mechanism called "column chunks" or "row group". In *ibaAnalyzer*, you have the possibility to choose the row group size according to your needs. The input corresponds to the number of rows per row group.

Create Spark compatible files
You can use the Apache Spark framework to work with Parquet files. Because several characters are not allowed as column name within this framework, this option replaces all such characters by underscores.

Time export mode
Select whether to export time data. If so, select the format. The time stamp appears as a separate column in the exported file.

- *None*
  No time stamp data is exported, however the start time and sampling rate are still available as meta data.
- *Relative time offset*
  Begins with "0" and counts the seconds from the file start. The additional column contains FLOAT values.
- *Absolute Date/Time*
  The additional column contains the absolute date and time. The Parquet data type TIMESTAMP is used.

Derive column names from
Here, you can select the column names for the Parquet file. The channel numbers, channel names or one of the comments are available for selection. Note that when using a comment, this information needs to be available.

When selecting the channel number, the names are formatted as follows:

- [M:C] for analog channels
- [M_C] for digital channels (dots are not allowed for channel names in the Parquet format)
- [M:C:S] for subchannels

With "M" as the module number, the signal (or channel) number is "C" and the subchannel number is "S".

The Parquet format only allows unique column names. If the column names are not unique in the original data, *ibaAnalyzer* automatically adds a corresponding suffix like "_1", "_2", etc.

Prefix measuring location
For the extraction of *ibaQDR* data files, you can optionally add the measurement location number as prefix before the column name of the channel number.

Meta data

When you export data from iba data files, a range of meta data (or info fields) are available. This data is also written to the Parquet format, where only one level of meta data exists. Therefore, this data is structured as follows:

- File level information (the standard info fields) are stored as normal key value pairs with the info field name as key.
- For computed columns and info columns, the specified name is used as key.
- Module level information use a key of the form "M[x]y" where "x" is the module number and "y" the field name.
- Channel level information use a key of the form "[x]y" with "x" being the channel name and "y" the field name.

Using this data structure, *ibaAnalyzer* can restore the complete file structure when opening the extracted Parquet file.

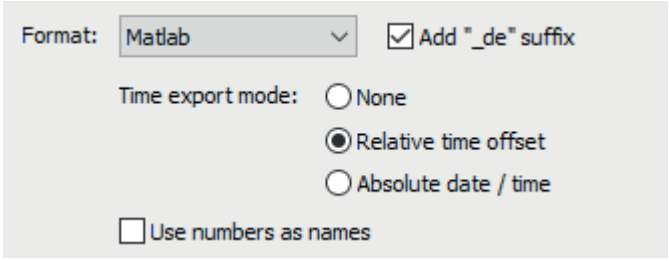
Length-based data and ibaQDR data (extraction only)

You can also extract length-based data or *ibaQDR* files to Parquet format.
If the correct length-based profile is used for the extract, *ibaAnalyzer* writes additional meta data to the file to indicate this. The following fields are used:

- "Lengthbased"
  This field indicates that the column contains length-based data.
- "LengthBase"
  In case of length-based data, this field shows the sample rate in meter.

15.1.5 Matlab

The software Matlab distributed by MathWorks provides its own binary data format with file extension .mat. In order to better support the Matlab integration, *ibaAnalyzer* can create MAT files. You can open these files directly in the Matlab software.



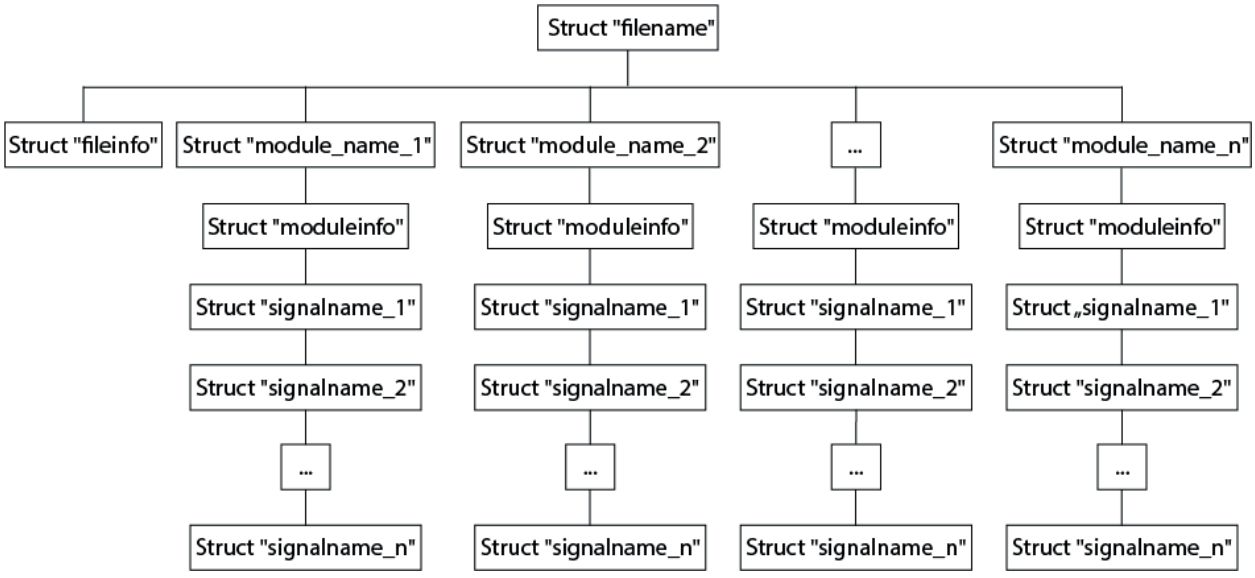
Time export mode
Similar as for other formats, it is possible to export an array containing the timestamps either as relative time offset or as formatted string. The time array is added to the "fileinfo" structure, which is described below.

Use numbers as names
With this option it is possible to not use the module and signal names for the names of the struct objects (see below). In this case, the module or signal number is used with some prefix "M" for modules and "S" for signals.

In order to display the correct name when opening the file again with *ibaAnalyzer*, the original names are stored in an additional field "name" within the structure.

Data structure

The software Matlab supports different data types and structures. *ibaAnalyzer* supports MAT files, which contain so-called struct objects. *ibaAnalyzer* creates nested struct objects in the following form:



This structure is used to resemble the internal structure of the iba data files.

The struct "filename" is the root structure. It contains only other structures containing info fields or modules. The name of the structure is derived from the filename displayed in the *ibaAnalyzer* signal tree.

The info fields on file level are stored as key-value pairs in a fixed structure "fileinfo". The field "clk" contains the sampling rate of the signal data. Further, a field "starttime" contains time stamp of the first data point. Several other info fields will automatically be added to this structure.

The individual modules are present in form of structures having the name of the module. In case the *Use numbers as names* option is set, the structures are named as "Mx" where x is the module number.

Every module has a fixed structure "moduleinfo", which contains module-level info fields as well as a field "name" containing the module name, and a field "ModuleID" containing the module number.

Like for the modules, the individual signals within one module are present as structures using the signal name or a string "Sx" with "x" being the signal number.

Every signal structure contains an array "data", which holds the signal values. Other key-value pairs are written representing the info fields on signal level. The most important fields are:

- "SignalID"
  the signal number within the module
- "Name"
  the signal name (displayed in *ibaAnalyzer*)
- "Unit"
  the unit (if present) as displayed in the signal grid
- "PDA_Comment1"
  the comment 1 displayed in the signal grid. Similar for comment 2
- "PDA_TBase"
  This field is present in case the sampling rate is different from the global "clk" value.
- "Lengthbased"
  This field indicates that the column contains length-based data.
- "LengthBase"
  In case of length-based data, this field shows the sample rate in meter.

Naming conventions

When extracting to MAT files, the nested structs receive names automatically. If you enable the *Use numbers as names* option, the signal ID is used.

15.2 Exporting data

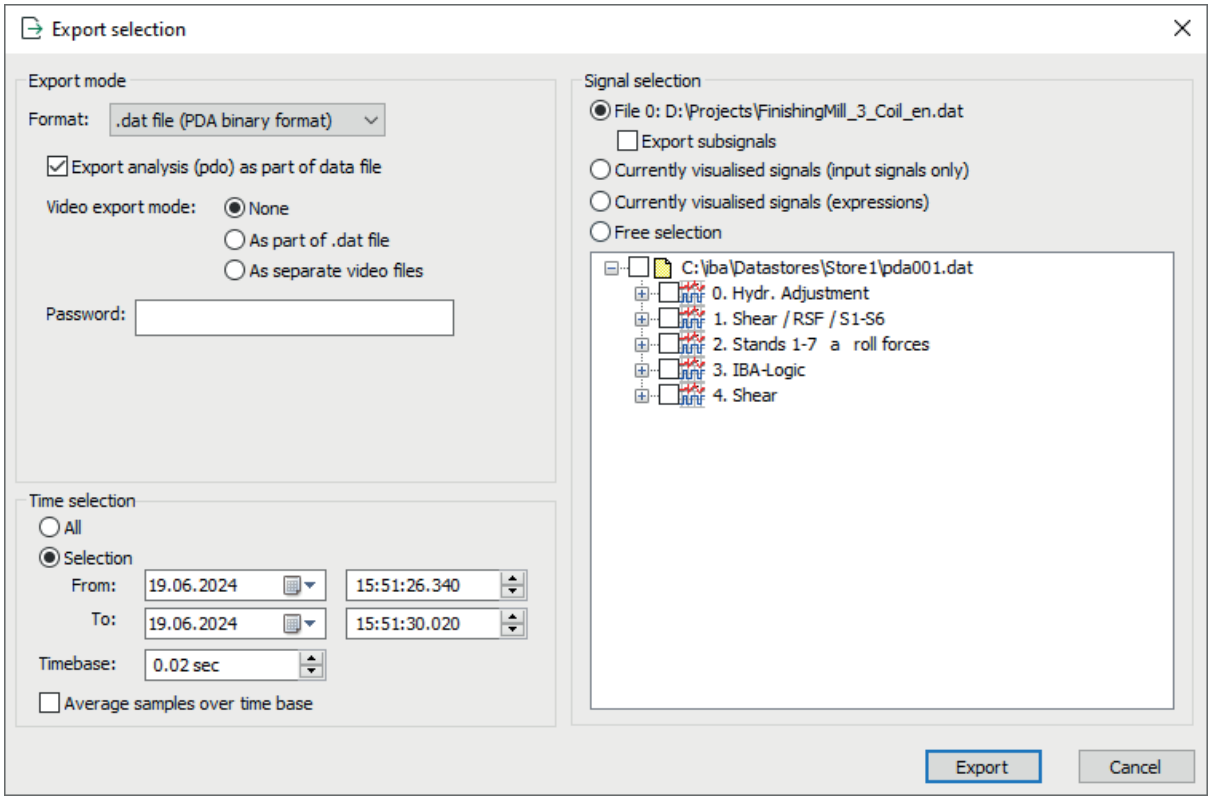
One advantage of the export function is the ability to limit the time range for the export file. For example, if you are investigating a malfunction in a system and have found the its cause and conditions, you can narrow down that specific time range from the measurement data and export it to a new file. This significantly reduces the file size and makes it easier to share the data related to the malfunction.

You can only use the export function manually, meaning you must configure and initiate each export manually.

For automated exports, you can use data extraction with *ibaDatCoordinator*. This requires the *ibaDatCoordinator-File-Extract* license, see also ↗ *Automatic extraction using ibaDatCoordinator, page 309*.

Procedure

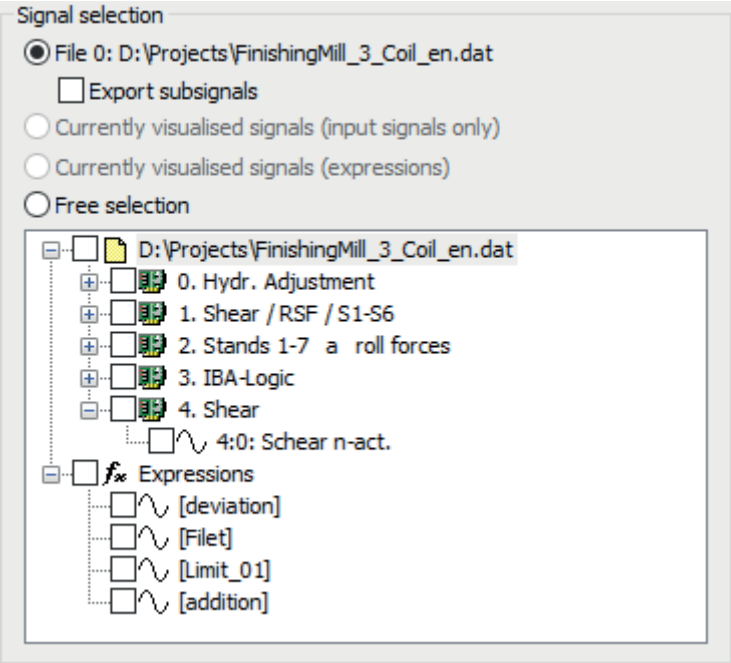
1. To export the open measurement data, access the export dialog via the *File – Export* menu.



2. Select the format for the export file and its settings, see ↗ *Supported file formats, page 276*.
3. Select the signals to export, see ↗ *Signal selection, page 287*.
4. Select the time range from which to export, see ↗ *Selection of time criteria, page 288*.
5. Once you have made the desired settings, click <Export>.
6. Enter a path and file name for the export file and confirm with <OK>.

15.2.1 Signal selection

Select the signals that you want to export to the new file.



The following options will help you make your selection.

File [number]
All signals in the data file are exported.

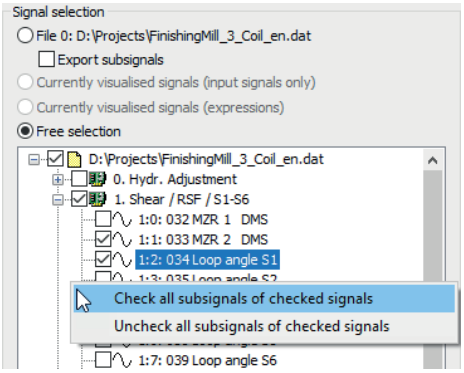
The number specifies the data file if multiple data files are open. The file selected in the signal tree window when the export dialog is opened is referenced automatically.

Export subsignals
If you enable this option, all existing subsignals of a signal are exported.
Subsignals are additional characteristic values such as maximum, minimum, average, or standard deviation. They can occur in the following cases:

- Database query: If the additional channel information was enabled when the data was extracted, the corresponding values are created as subsignals below the parent signals in the signal tree. By enabling this option, these subsignals are exported in both binary and ASCII files. Each subsignal has its own labeled column.
- HD query: As this query is based on a time base, the minimum and maximum signals are provided as subsignals in the signal tree by default.

Tip

If you do not want to export all the signal content of the file, but select the signals to export individually, you can add the subsignals via the context menu in the signal tree.



Currently visualised signals (input signals only)
All input signals that are visible in a graph in the current analysis are exported.

Currently visualised signals (expressions)
All currently displayed input signals and expressions are exported.

Free selection
You can select the signals and expressions to export in the signal tree below using the checkboxes.

If you select a module, all signals of the module are also selected.

To select individual signals, open the module with the <+> button and select the desired signals. When you select a signal from a module, the module is also highlighted. In this way, you can see at a glance the modules from which signals are being exported.

15.2.2 Selection of time criteria

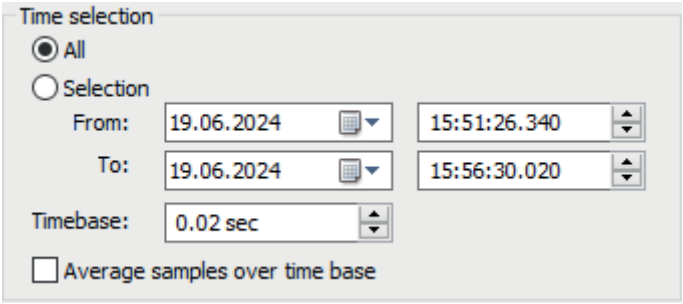
You can set various time criteria for the export in the dialog. You can only export data from a specific period, not the entire recording time, see ↗ *Time range, page 289*.

You can also change the time base to obtain a lower data resolution, see ↗ *Time base, page 290*.

Setting the time criteria to the required range can significantly reduce the file size.

15.2.2.1 Time range

When exporting, you can choose whether you want to export the data for the entire recording time (*All* option) or only a specific time range (*Time* option).



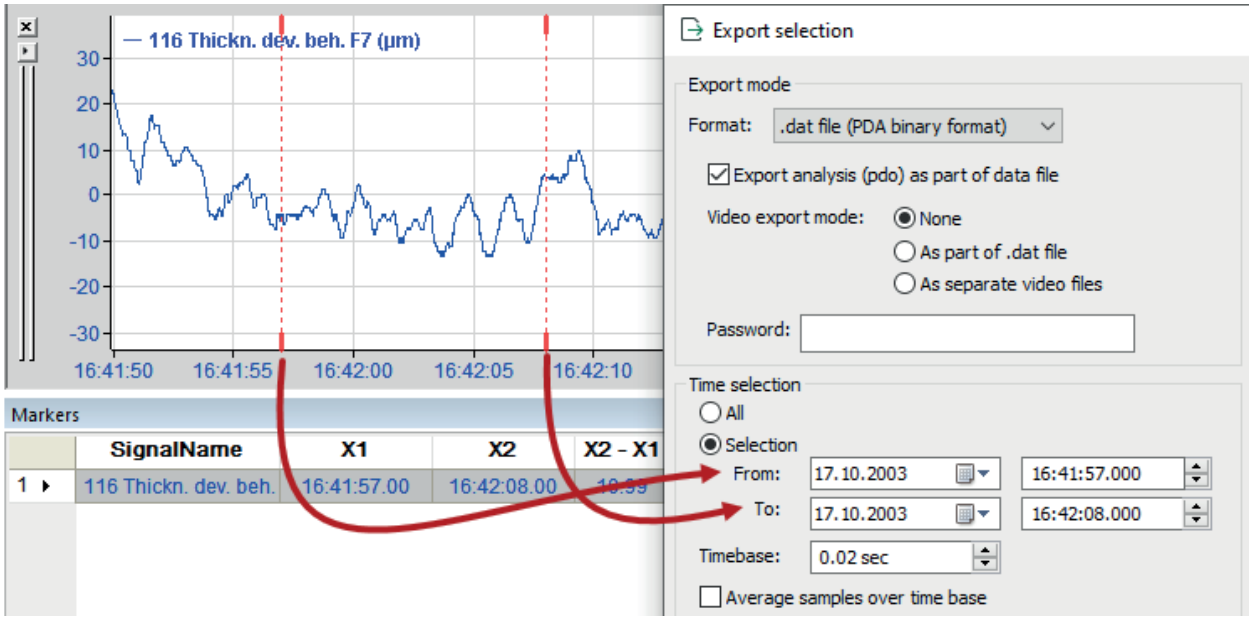
The default dates (*from*, *to*) are taken from the info fields of the data file or HD query. You can customize the entries by way of the fields.

Applying time range from markers

If you have used markers in the graph to limit a time range, you can also apply that marker range for the export.

1. Open the *Markers* table and place the markers in the recorder window.
2. Open the export dialog via *File – Export*.

→ The *Time* option shows the time range that the markers cover.



15.2.2.2 Time base

The time base dictates the time resolution of the values in the export file. The default is the time base from the data file.

If the precision from the data file is not required, you can increase the time base to export the values at wider intervals. This also means that the number of exported values will be smaller.

Use the arrow buttons to increase the time base by multiples of the original time base. The export time base can never be smaller than the original time base.

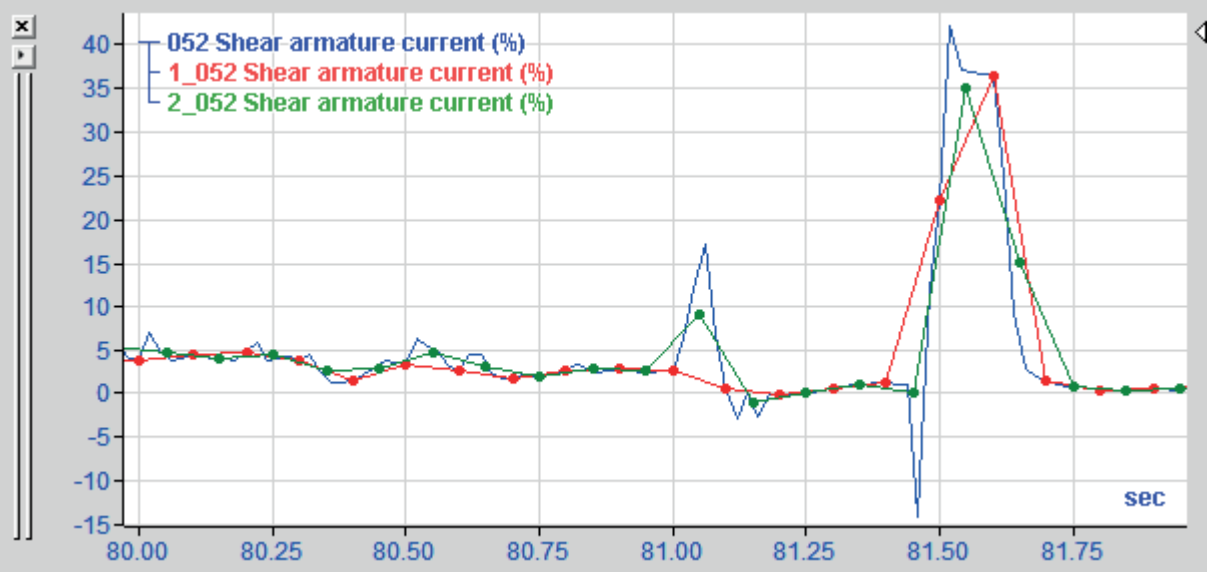
Average samples over time base
If you enable this option, the average values are calculated on the basis of the changed time base.

By default, this option is disabled, and the values skipped during export are not averaged.

Effects of time base adjustment

The actual values are exported, and linear interpolation is inserted between the values.

The graphic shows a comparison between the different options.



Signal	Color	Setting
052	Blue	Original signal with time base 0.02 s
1_052	Red	Exported signal with time base 0.1 s without averaging
2_052	Green	Exported signal with time base 0.1 s with averaging

15.3 Extracting files

The file extraction function in *ibaAnalyzer* is a standard function that you can use interactively without an additional license.

You can extract, transform and load all data in iba data file format (*.dat) originating from *ibaPDA*, *ibaQDR*, *ibaLogic*, *ibaScope* or third-party applications using the *ibaFiles* library.

Supported formats are the iba format (*.dat), text files (CSV or ASCII), Apache Parquet, Matlab (*.mat), COMTRADE and TDMS. For further details, see chapter ↗ *Supported file formats, page 276*.

Summary of the major functions

- Loading measurements from iba data file
- Transformation of data or generation of new virtual signals
- Extracting data to various formats
- Automated processing with *ibaDatCoordinator*

Other documentation



For the usage of *ibaDatCoordinator*, see the corresponding manual or ↗ *Automatic extraction using ibaDatCoordinator, page 309*.

Notes for users of already existing solutions with ibaAnalyzer-File-Extract and USB MARX dongle

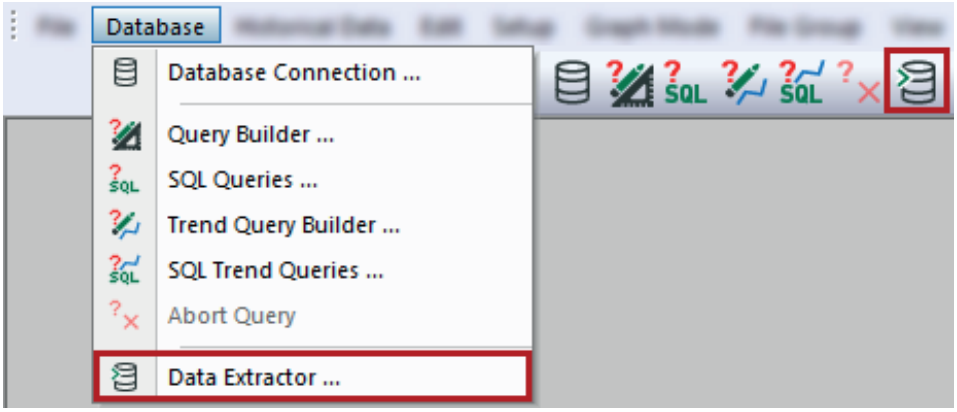
Even after updating *ibaAnalyzer* to v8.x and *ibaDatCoordinator* v3.x, you can continue to use USB MARX dongles with the licenses *ibaAnalyzer-DB* or *ibaAnalyzer-File-Extract* as usual.

Procedure

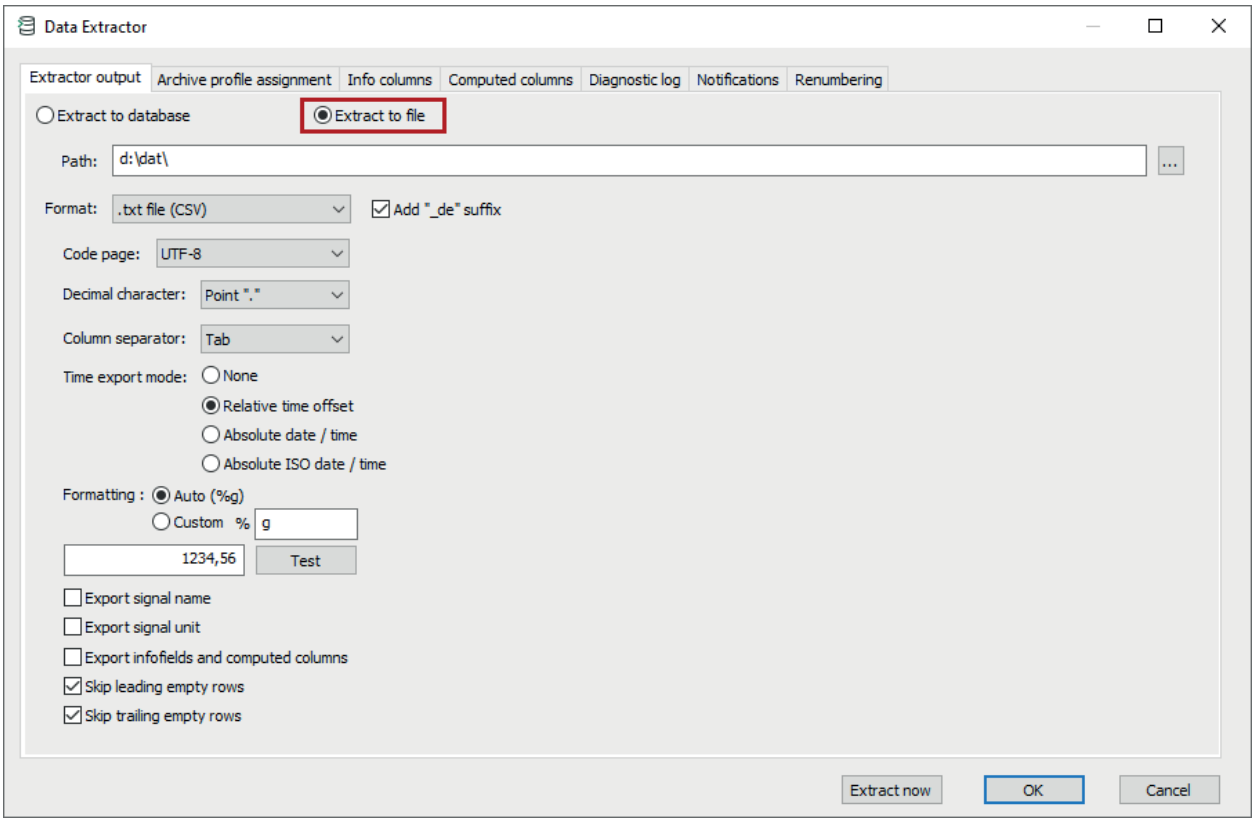
Open the Data Extractor dialog and configure the extraction file using the various tabs, see ↗ *Data Extractor, page 292*.

15.3.1 Data Extractor

Use the dialog *Data Extractor* to configure the file extract. You can open the Data extractor either via the menu *Database – Data Extractor* or via the corresponding button on the toolbar.



The *Extractor Output* tab appears automatically, for further details, see ↗ *Extractor output, page 294*.



Settings for data extraction to file

On the *Extractor Output* tab, select the option *Extract to file*.

→ Thereby all registers adjust the respective settings to the data extraction into a file.

The following chapters describe the individual tabs of the data extractor with the *Extract to file* option selected.

Buttons on each tab

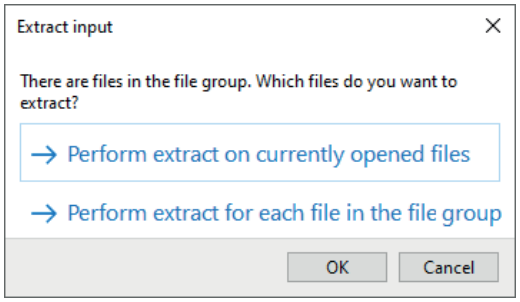
All tabs have the following buttons:

- <Extract now>
  ibaAnalyzer creates a file based on the current settings for file extraction. By default, all extracted filenames consist of the source filename augmented by "_de". If extraction from the same source file is repeated, the extracted files are indexed starting with "_00".
- <OK>
  ibaAnalyzer saves all current settings within the dialog and closes the window.
- <Cancel>
  ibaAnalyzer discards all changed settings and closes the window.

Tip



If *ibaAnalyzer* opens multiple files in the file group (e.g. from a conditional search using *ibaHD-Server*), you can apply the extraction to all files contained in the file group. When you click the <Extract Now> button, a corresponding dialog appears.

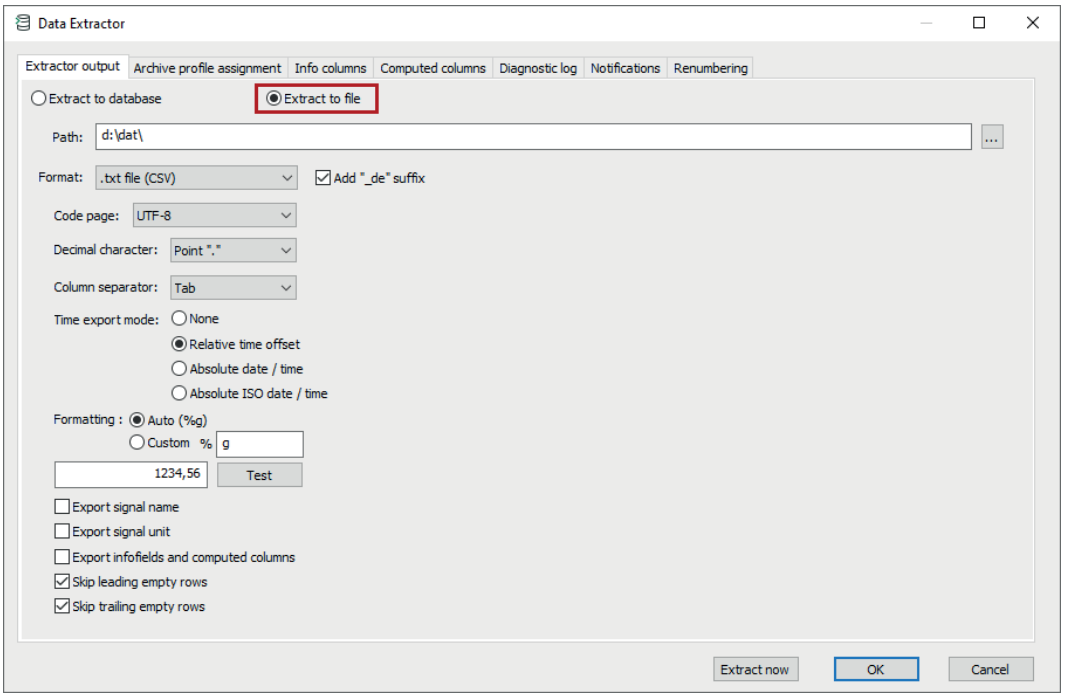


To perform the extraction for multiple files in the file group (2nd option), you need the *ibaDatCoordinator-File-Extract* license.

15.3.1.1 Extractor output

The *Extractor Parameters* tab automatically opens first when you open the data extractor.

Select the *Extract to file* option. Here you can specify the location and format of the extracted file.



The Extract to database option extracts data into a database, see also part 4 *Database Interface*.

ibaAnalyzer automatically saves the extracted files in the specified path. If the directory does not exist, it is created automatically. Use the <...> button to browse the file system if necessary.

Filename
File extraction creates output files with the same name as the input file and an additional "_de" suffix. If you disable the *Add "_de" suffix* option, the original filename is used. The filename extension depends on the desired output format.

Use this tab to specify the output file format. For further information, see chapter ↗ *Supported file formats, page 276*.

Note

If you call the analysis with files for an extraction from *ibaDatCoordinator* , you have to define the location of the extracted files there. This overwrites the location defined in *ibaAnalyzer*. In case the directory is password protected, you have to enter the username and password in *ibaDatCoordinator*.

15.3.1.2 Archive profile assignment

In the *Archive profile assignment* tab, you can configure archiving profiles that define the sampling rate (time-based or length-based). You then assign the profile to the signals for extraction. You can configure more than one profile. You can assign only 1 profile to a signal.

To create an archiving profile, at least one data file must be open for extraction.

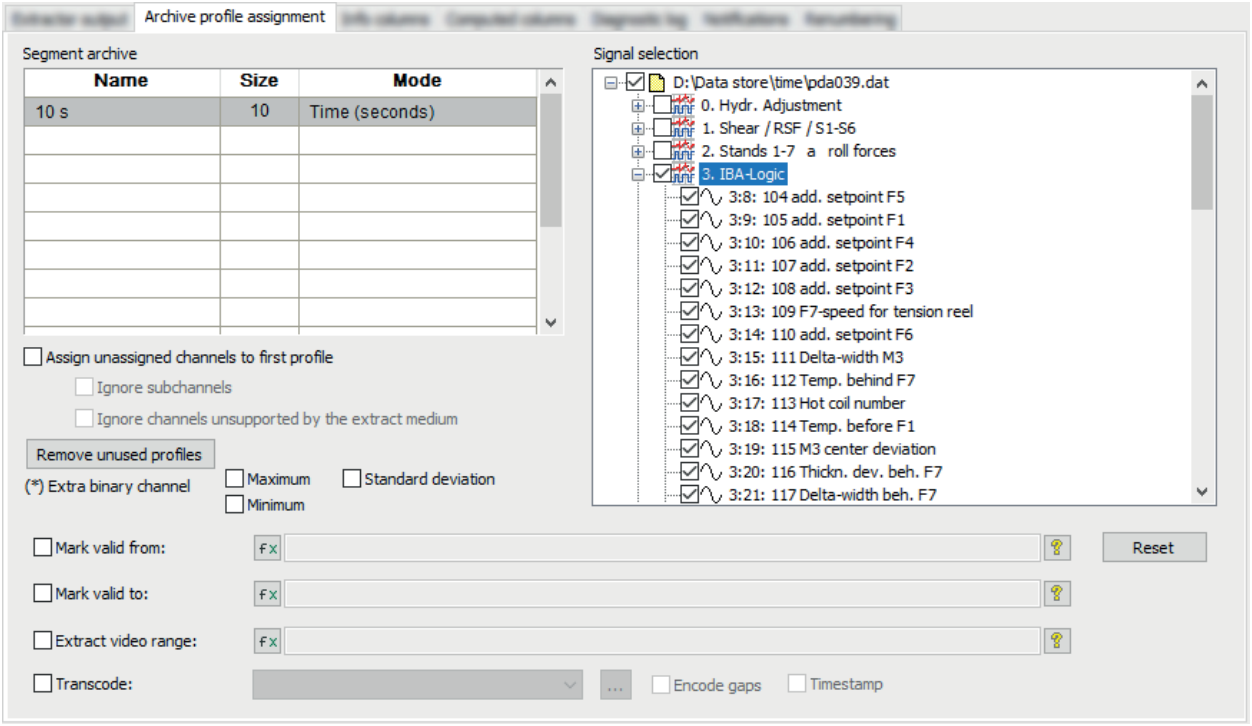


Table columns

Name column

Select a cell in the *Name* column and enter a profile name.

It is recommended to use a name that reflects the selected sampling rate for the extracted file, e.g. "10 s" = Size (10) × Mode (Time (seconds)).

Size and Mode column

In the *Size* column, enter a multiplier to reflect the sampling rate for the extracted data, e.g., 10 s = "Size (10)" × "Mode (Time (seconds))".

Click in the *Mode* column and select an extraction mode:

- For *Time* and *Length* modes, you can enter a size < 1 to obtain fractions of the unit under *Mode*.
  When entering the fraction factor for values < 1, be sure to enter a period instead of a comma as the decimal separator (e.g., "0.5" instead of "0,5").
- The **signal default base* mode uses the original time base of the signal or a multiple of it. Therefore, fractions are not useful here. For example, a temperature measured every minute does not need to be extracted every 100 ms.

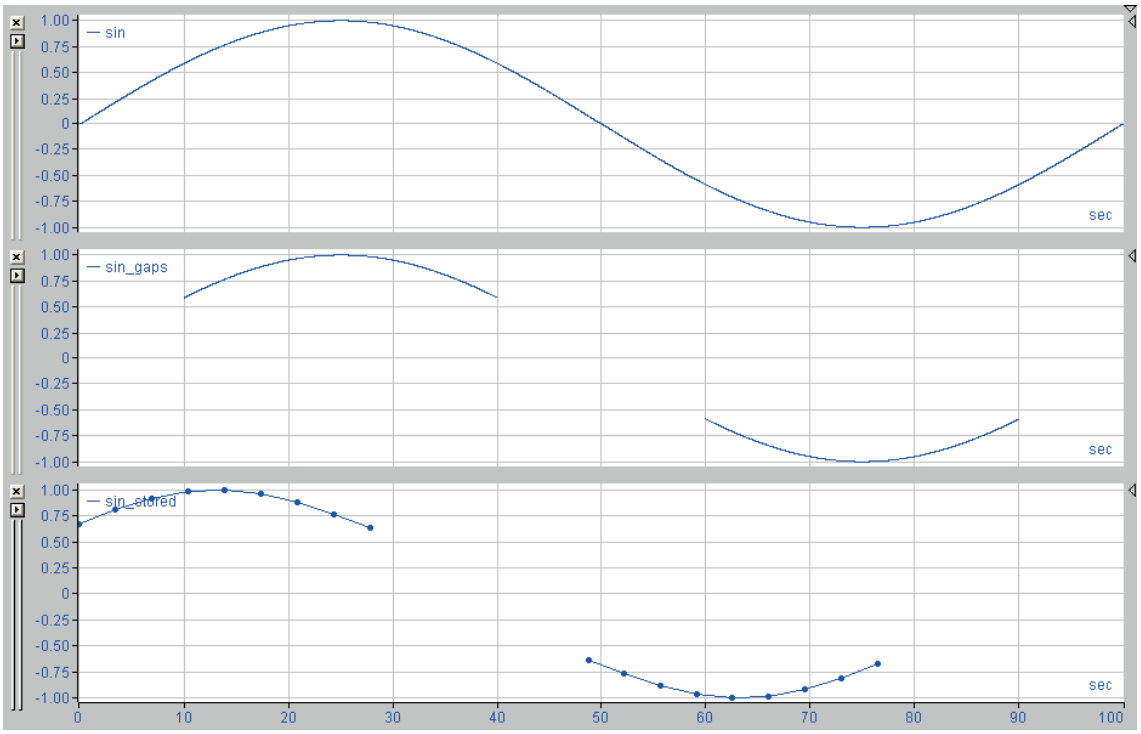
  If the time base of the recording signal is 1 ms, the 100 ms profile could also be defined like this: *Size* = 100 and *Mode* = **signal default base*.

The **signal timebase* and **signal lengthbase* modes have the same function, but only for time-based or length-based signals.

In the same way, the **signal time- and lengthbase (QDR)* mode supports the extraction of QDR files with time-based and length-based signals. The resulting data file also contains time- and length-based data like the original file.

- With the modes *Time segments* or *Length segments*, you can define a fixed number of equidistant time segments or length segments. For example, *Size* = 1 and *Mode = Time segments* means that only 1 segment value is stored for the entire data channel.
  If parts of the source signal are invalid, the extracted segments containing the invalid parts are empty. If the empty segments are at the beginning or end of the signal, they are not included in the extracted data. If the empty segments are located between valid segments, the extracted data contains corresponding gaps with invalid values.

  The following figure shows as example a signal with gaps over time (central trend graph). In the gaps there is no valid data. The lower trend graph shows the result after extracting the signal with *Time segments* mode. The first and last 10 seconds have been discarded while the gap in the middle remains within the trend.



- *Time, non-equidistant* mode is supported only for extraction in text files and extracts non-equidistant data.

Assign unassigned channels to first profile

If you enable this option, all signals that have not been manually assigned to a profile are automatically extracted with the first profile. However, they are not marked in the signal assignment. This happens e.g. by the extension of the data files in *ibaPDA*. This option can be useful if the file structure changes over different files.

Ignore subchannels
If you enable this option, subchannels from an HD query are not extracted, e.g. the channels min/max.

Ignore channels unsupported by the extract medium
If you enable this option, channels that the destination file format does not support are not extracted, e.g. *ibaCapture* synchronization channels for non-iba formats.

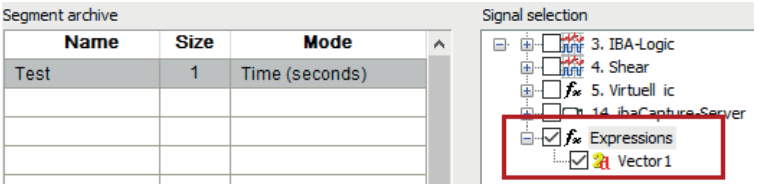
<Remove unused profiles>

Use this option to automatically delete profiles that have no signals assigned.

Signal selection

You have to assign signals to the created profiles. Select signals to be included in the extracted data. The extracted signal values are the average values of the source signal values within the selected sampling rates.

When you enable or disable a module node, you also enable or disable all signals that belong to that module. You can enable or disable single signals in the opened module node. The selected signals are always assigned to the marked profile (gray background color). When you change the profile, the associated selected signals also change. You can assign only 1 profile to a signal.



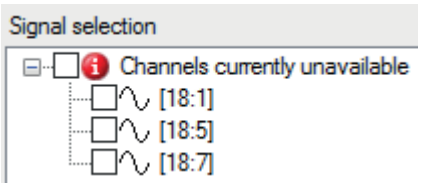
Note

If you assign a length mode to a time-based signal or a time mode to a length-based signal, then an error message appears during extraction.

The **Signal time base* mode is suitable for time-based and length-based signals.

Note

If the archive profile specifies signals that are not present in the current analysis file, they are listed under *Channels Not Currently Available*.



Extra binary channel Maximum, Minimum or Standard deviation

If you enable this option, subchannels are created, each representing the maximum, minimum or standard deviation of the source signal values within the selected sampling rates. This option can be useful for newly acquired signals.

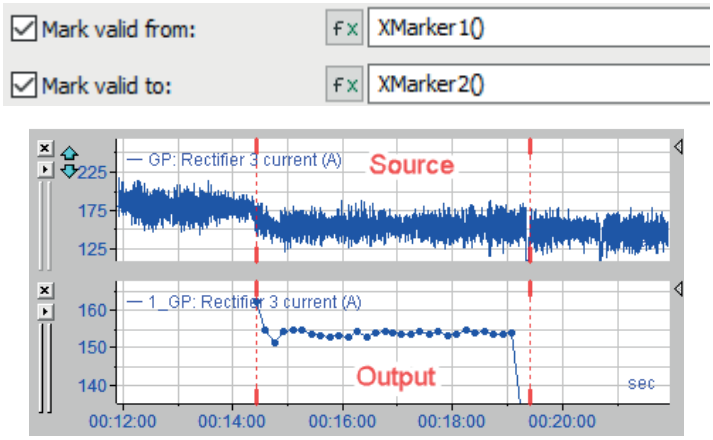


Note

If the selected sampling rate of the profile is identical to the sampling rate of the source signals, these values have no meaning.

Mark valid from/Mark valid to

If you do not want to extract the signal over the full duration, you can define the start and end of the profile range here. You can use expressions to calculate the range from your data.



If neither of the two expressions is defined despite activated options, the signal is extracted completely.

If the specified expressions cannot be evaluated, an error message appears during the extraction.

Extract video range

When exporting videos, only parts of the video may contain important information. By providing a corresponding digital signal, only the parts of the video marked by this signal are extracted to the file. This procedure helps to save storage space.

Transcode

If you enable this option, the video file is re-encoded for this archive profile. This allows you to change the video resolution, format and quality, e.g. to reduce the file size.

To do this, select an encoding profile from the drop-down menu. The profiles are loaded from the *ibaCapture* server when you activate the *Transcode* option and configured profiles are available. Without a connection to the server, you can use the standard profiles. For more information on the encoding profiles, click the <...> button.

Further options:

- *Timestamp*: If you select the *Timestamp* option, a timestamp is inserted as an overlay in the video.
- *Encode gaps*: If you select the *Encode gaps* option, video gaps are encoded as individual frames in the file, including an optional timestamp. If this option is deactivated, video gaps are inserted into the file as omissions.

This function is supported from *ibaCapture* v5.3.0 or higher.

<Reset>

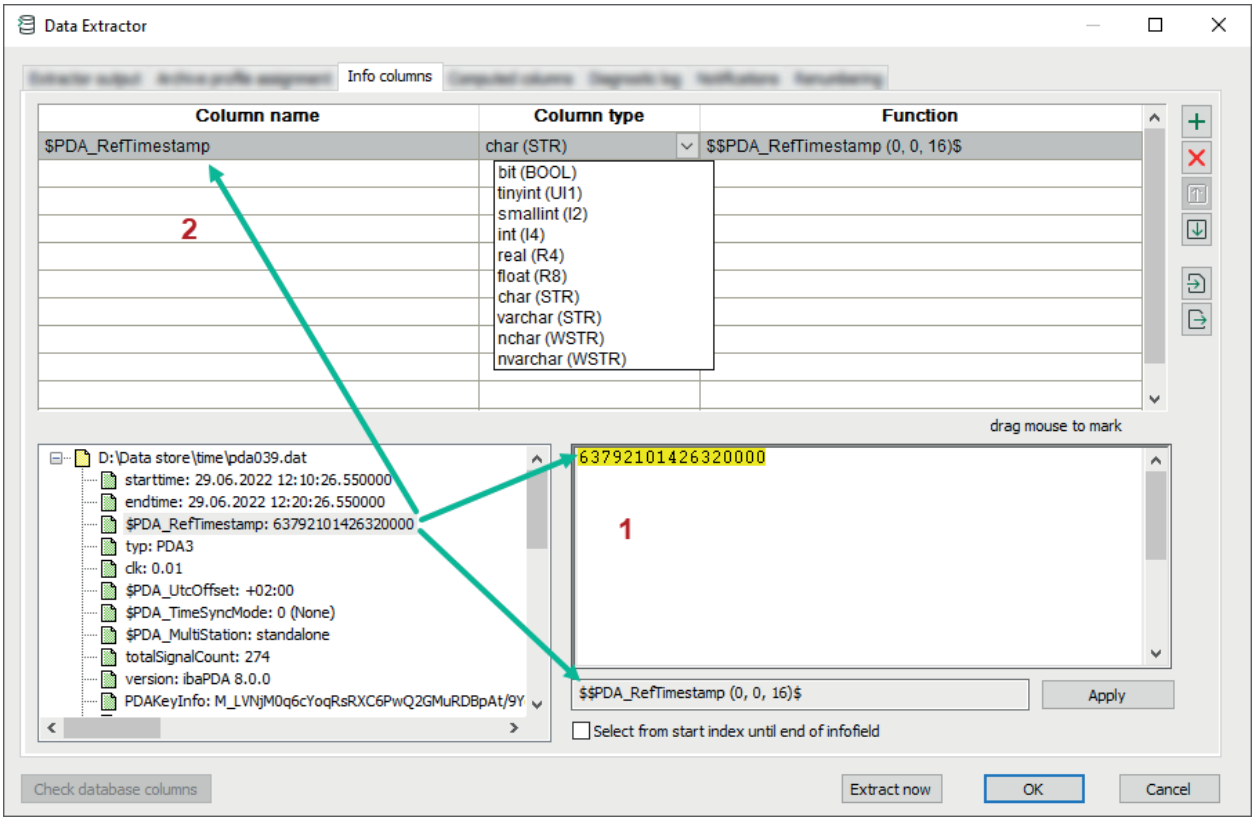
Remove all signals from the selected profile.

15.3.1.3 Info columns

Use the *Info columns* tab to extract information from the info part (e.g. start time, texts) of the source file. This data can then be found in the info part of the extracted files.

Using info fields

You have two possibilities to add info fields for extraction.



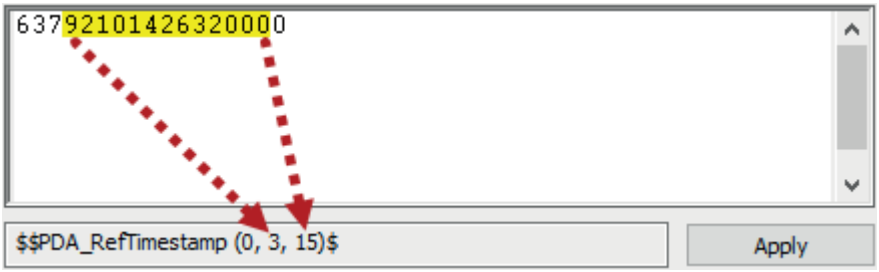
First select a line for the field information in the table (2).

- Click on the source field. The field information is transferred to the processing field (1). Select the range of characters with the mouse and click on <Apply>.
  Your selection is transferred to the table (2) and the processing field. The *Function* column shows the character selection.
- Double-click on the source field.
  The field information is completely transferred to the table (2) and the processing field (1).

Selecting a character range

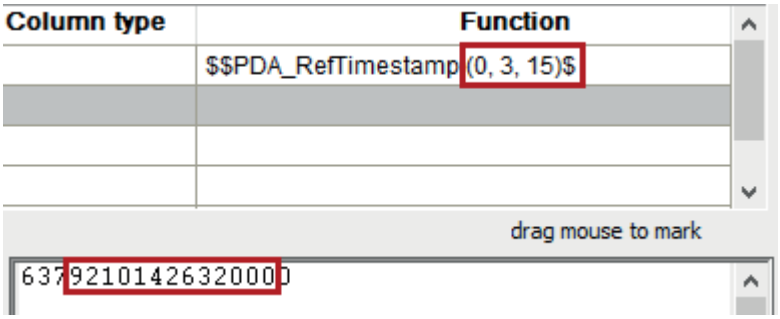
You can also select a range of characters from the contents of the info field.

1. Click the first required character, drag the selection to the last required character.



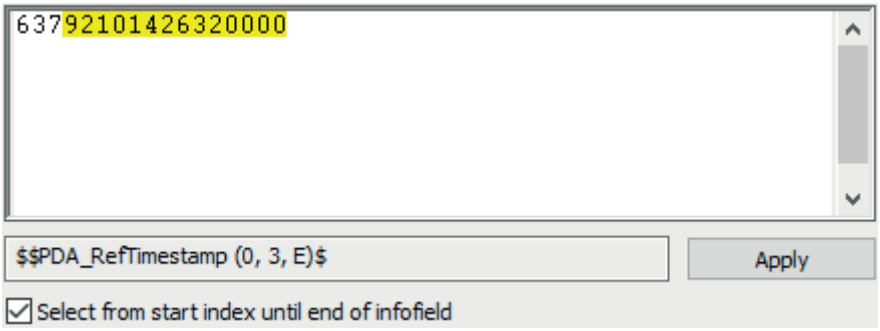
2. Click on <Apply>.

→ The excerpt from the source string is transferred to the column field.



Alternatively, you can also enter the position of the first and last character manually in the *Function* column.

Select from start index to end of info field
If you enable this option, all characters from the selected start point to the end of the info field are selected.



Editing info fields in the table

You can rename the *Column name* manually. Avoid special characters.

Column type specifies the data type and is always "char" for file extraction.

You can change the character range of the *Function* in the processing field.

For information on further editing options, see ↗ *Editing the info columns, page 302*.

Further option in the Info columns tab

Extract module names
This option is relevant only for the extraction to databases.

Remove custom info fields
By default, *ibaAnalyzer* also extracts custom info fields (formerly called technostring). However, these custom info fields can contain sensitive information. If you enable the *Remove custom info fields* option, these info fields are not extracted, but only the info fields that you have defined under *Info columns* or *Computed columns*.

15.3.1.3.1 Editing the info columns

Using the buttons on the side, you can change the order of info columns or import or export information about the columns.

	Add line	Add a new empty line above the currently selected line.
	Delete line	Delete the currently selected line.
	Move line up/down	Move currently selected line up/down.
	Import info file	Import column information from text file. The *Open text file* dialog appears. **Alternative:** Right-click on the table header and select *Import*. **Attention:** An import always overwrites all current columns.
	Export info file	Export column information to text file. The *Save text file* dialog appears. **Alternative:** Right-click on the table header and select *Export*.

Tip

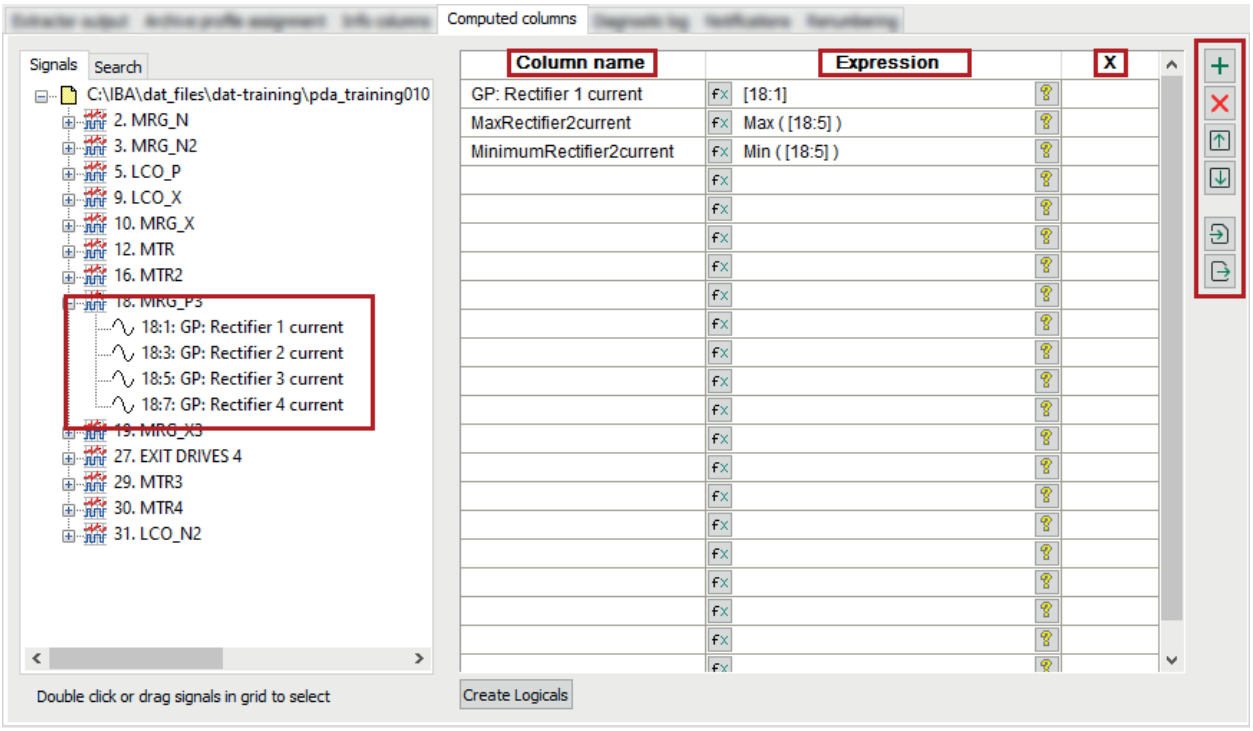


The import-export-function is very helpful if you want to use the same expressions in different places, e.g. for the database extraction or when handling large numbers of computed columns.

The exported text file is easy to edit using an editor or MS Excel.

15.3.1.4 Computed columns

Use the *Computed columns* tab to specify operations for individual signals. These operations create one value for the associated signal (e.g. the average, maximum, or minimum of all the data in one signal).



Select and prepare signals

Add the desired signal to the table either by double-clicking on the signal or using drag & drop. You can also select multiple signals in the signal tree by the standard Windows method of a mouse click + <Shift> or <Ctrl>, and drag them into the table with the mouse.

The *Column name* column initially contains the source signal's name, which you can change manually.

The *Expression* column initially contains the channel number of the source signal. You can either enter an expression or use the <fx> button to open the expression builder. The result of the expression should be a signal value.
If you do not use an operation or group function for *Expression*, then *ibaAnalyzer* automatically extracts the average (AVG) from the signal data.

The *X* column automatically shows the X-axis mode of the source signal (time-based or length-based).

You can edit the computed columns in the same way as the info columns, see ↗ *Editing the info columns, page 302*.

<Create Logicals>
At different places in *ibaAnalyzer*, you can define and manage logical expressions. This button creates a group of logical expressions from the defined computed columns. You can then manage the created logical expressions via the *Logical signals* dialog. When renaming or moving the expressions, all references to them are automatically adapted.

If you have already used the same expressions or the same names elsewhere in *ibaAnalyzer* and they are not yet grouped, you can select how you want to proceed with these expressions: ignore, overwrite other expression or replace this expression with the other one.

Special feature: Text signals

You can also use text signals in computed columns. A column with the name of the text signal and its first value is then created in the file.

If you want to extract a specific value from a text signal with several values in the data file, you have to select this value explicitly using a suitable signal expression.

Example: Selection of the third value (counting starts at 0)

```
XCutRange([text signal], XFirst([text signal],2), XFirst([text signal],2) + 0.01)
```

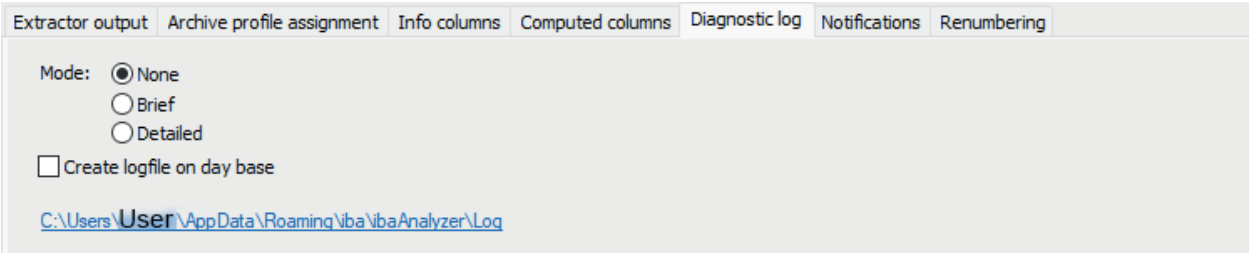
15.3.1.5 Diagnosis and notification

In a production environment it is necessary to monitor automated processes, to log processing information for diagnostic purpose and to generate notifications.

15.3.1.5.1 Diagnostic log

The diagnostic log is useful for checking progress when automatic extraction is operating (e.g. triggered by *ibaDatCoordinator*). If you select a log mode, *ibaAnalyzer* creates a log file and updates it with each extraction.

The linked path takes you to the directory containing the log files.



Mode

- *None*: *ibaAnalyzer* does not create log files of the extraction.
- *Brief*: The log file only contains the filenames of the extracted files.

```
24-May-17 13:45:54: Start extract file 'D:\IBA\dat files\dat-training\pda_training021.dat'
24-May-17 13:46:57: Start extract file 'D:\IBA\dat files\dat-training\pda_training021.dat'
```

- *Detailed*: The log file contains the filenames of the extracted files and individual channels.

```
24-May-17 13:16:10: Start extract file 'D:\IBA\dat files\dat-training\pda_training021.dat'
24-May-17 13:16:10: Start extraction to file 'd:\dat\pda_training021_de.dat'
24-May-17 13:16:10: Start extract channel 18:1 GP: Rectifier 1 current with profile 100ms
24-May-17 13:16:10: Start extract channel 18:3 GP: Rectifier 2 current with profile 100ms
24-May-17 13:16:10: Start extract channel 18:5 GP: Rectifier 3 current with profile 100ms
24-May-17 13:16:10: Start extract channel 18:7 GP: Rectifier 4 current with profile 100ms
24-May-17 13:16:10: Extract completed
24-May-17 13:16:35: Start extract file 'D:\IBA\dat files\dat-training\pda_training021.dat'
24-May-17 13:16:35: Start extraction to file 'd:\dat\pda_training021_de_00.dat'
24-May-17 13:16:35: Start extract channel 18:1 GP: Rectifier 1 current with profile 100ms
24-May-17 13:16:35: Start extract channel 18:3 GP: Rectifier 2 current with profile 100ms
24-May-17 13:16:35: Start extract channel 18:5 GP: Rectifier 3 current with profile 100ms
24-May-17 13:16:35: Start extract channel 18:7 GP: Rectifier 4 current with profile 100ms
24-May-17 13:16:35: Extract completed
```

Create logfile on day base

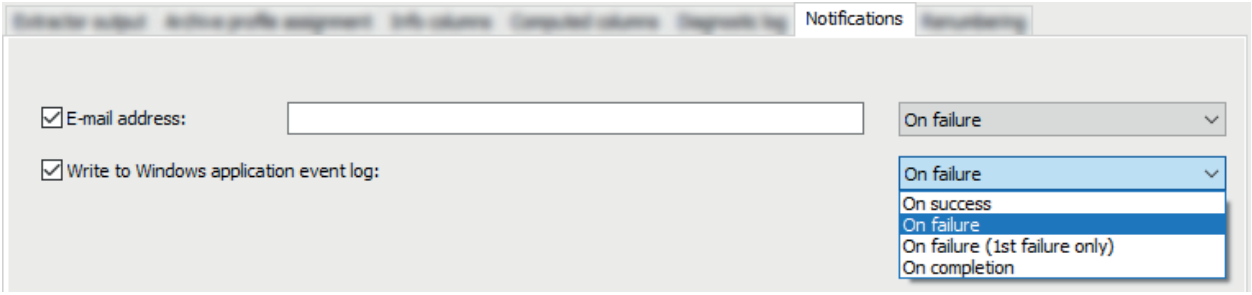
ibaAnalyzer creates a log file for each day, e.g. `DataExtractorLog_23_02_2026.txt`.

Note

If logging is activated permanently, you have to implement a cleanup strategy for the log files externally (not part of *ibaAnalyzer*).

15.3.1.5.2 Notifications

On the *Notifications* tab, you can choose between two communication methods how to receive notification or record events.

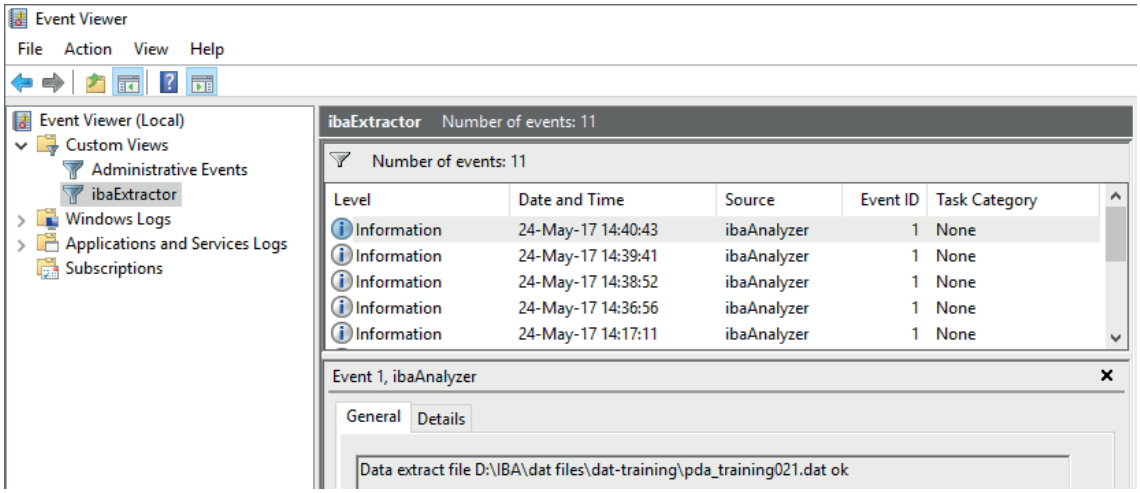


E-mail address
Enter the address and select the send mode.

Note

To send e-mails, a standard e-mail client (e.g. MS Outlook, Outlook Express, etc.) must be installed on the computer.

Write to Windows application event log
Events are shown in the Windows Event Viewer. Select the send mode.



Send modes for each communication method

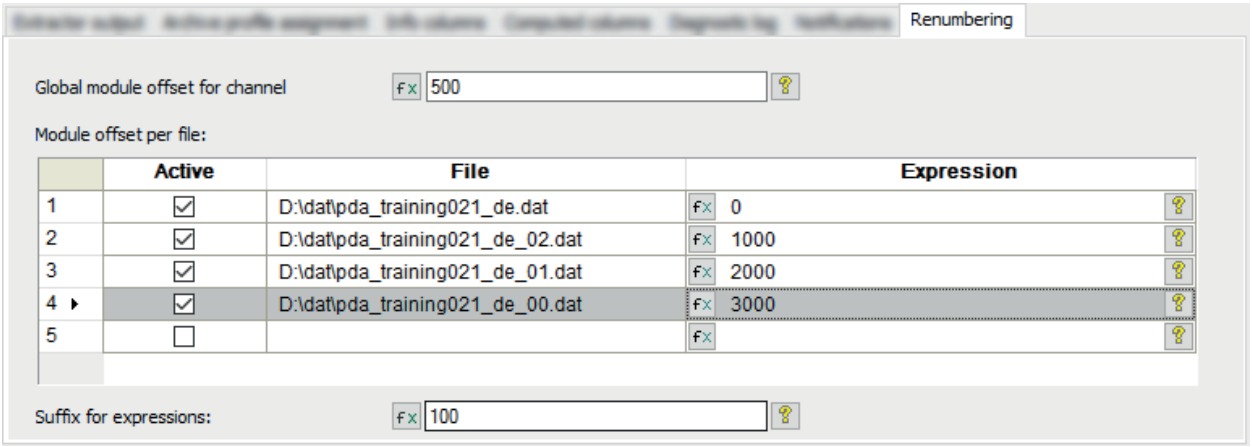
- *On success*
  Notification when extraction is successful.
- *On failure*
  Notification when any extraction has failed.
- *On failure (1st failure only)*
  Notification when the extraction has failed for the first time.
- *On completion*
  Notification when the extraction has been completed, success or failure is irrelevant.

15.3.1.6 Renumbering

If multiple files are open and you want to extract multiple signals from multiple files, it is likely that some channels have the same ID number. The channels must therefore be renumbered, otherwise they cannot be uniquely identified in the exported media. *ibaAnalyzer* can perform this renumbering automatically, but will issue a warning if this is required before proceeding with the extraction.

The channel IDs consist of a module number and a position within the module.

The *Renumbering* tab allows you to specify an offset to the module numbers for each file, so you can prevent ID collisions and avoid automatic renumbering.



Global module offset for channel
This number is added to the data file channel numbers.

Active column
Only active files receive a channel offset. Extraction of channels in inactive files results in a notice.

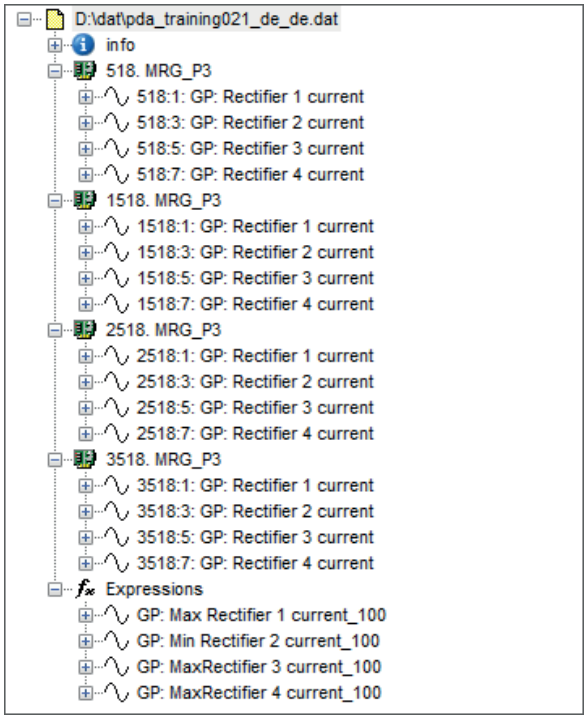
File column
The open data files are listed automatically in the File column.

Expression column
Enter the offset for each data file in this field.

Suffix for expressions
This applies only to expressions, names and comments created in *ibaAnalyzer*. The specified suffix is appended to the expression names.

Examples for extracted file

Extracted data with renumbering (based on the settings above) is displayed in the extracted file as follows.



15.3.2 File extraction via the command line

Note

To use the command line switches for extraction, you need the *ibaDatCoordinator-File-Extract* license.

If you run *ibaAnalyzer* via the command line, some of the command line switches for *ibaAnalyzer* are especially suitable for file extraction. This allows you to automate post-processing. The used syntax is the following:

```
ibaAnalyzer.exe datfilename1 [datfilename2] … [datfilenamen] [pdo-
filename] [/switch]
```

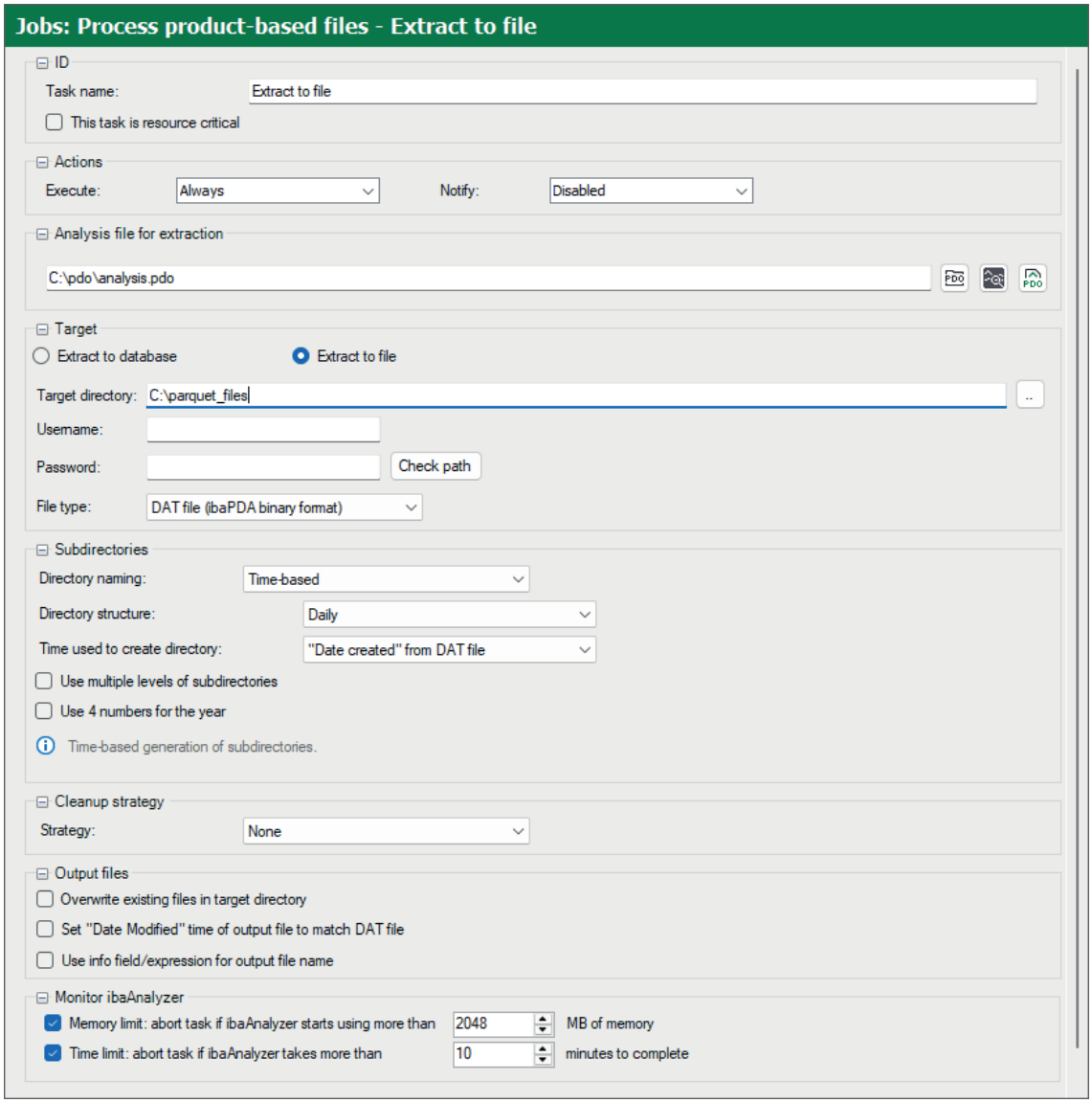
ibaAnalyzer then opens with various data files, an analysis and various options, which are described below. In each case, *ibaAnalyzer* automatically opens all specified data files and executes the assigned tasks. Note that in some cases the analysis is mandatory, for example if the information required for the task is only available in the analysis file.

The following command line switches are useful for file extraction. For a detailed description of the switches, see Part 1, *Using switches in the command line*.

- /extract[:filename]
- /append
- /print
- /report[:filename]

15.3.3 Automatic extraction using ibaDatCoordinator

With the *ibaDatCoordinator* tool and the *ibaDatCoordinator-File-Extract* license, you can automate the data extraction into other file formats using an extract-to-file task.



Here, some settings for the extraction function of *ibaAnalyzer* in combination with the extract-to-file task in *ibaDatCoordinator* are described.

Analysis file for extraction
Specify the path and the filename of the PDO file in this field or select the file using the browser button. To open the analysis in *ibaAnalyzer*, click on the *ibaAnalyzer* button.

Target/Target directory
Select the *Extract to file* option. Specify the path in this field or select the file using the browser button. If username and password are required to access the (remote) directory, you can enter them there as well. You can check the connection using the <?> button.

The *Extract to database* option requires an *ibaDatCoordinator-DB* license.

File type
Select the required file type. The available file formats are described in ↗ *Supported file formats, page 276*.

Date modified
For archiving and sorting it may be necessary to manipulate the "date modified" tag of the resulting file. Enable the option *Set 'Date Modified' time of output file to match DAT file* to have the same modification date as in the original file.

Other documentation



More detailed descriptions of other available options or for the general use of the *ibaDatCoordinator* software can be found in the corresponding manual.

For information about extracting data into databases, see part 4 *Database interface*.

15.4 Export of text signals to an ASCII file

In addition to numerical values, you can also export text signals to an ASCII file. Like other signals, text signals are also included in the signal tree, and you can select them for export.

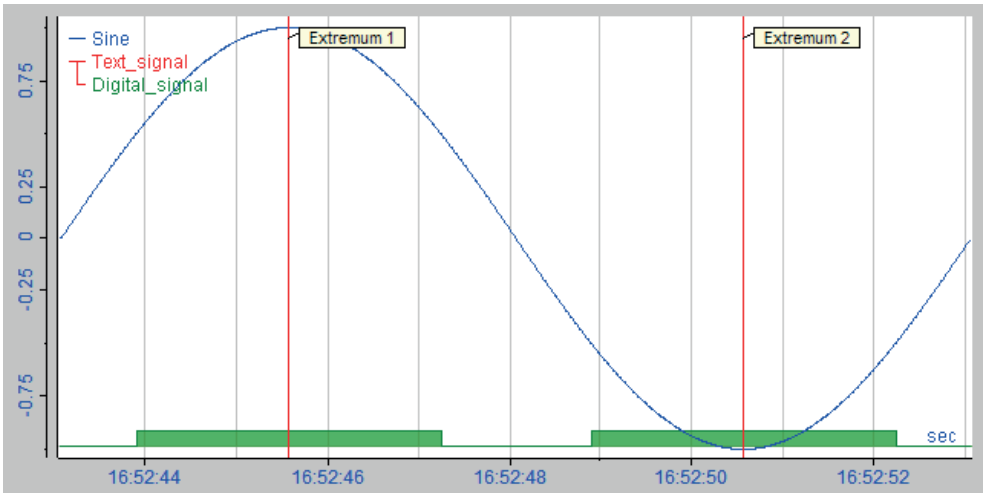
In the exported ASCII file, every line corresponds to a particular time stamp. Each time stamp differs from the first by an integer multiple of the set time base.

In the case of text signals, the time stamp is rounded down so that the text is written in the line with the last time stamp that is less than or equal to the original time stamp of the text sample. As there are usually more time stamps than text samples, the columns for text signals remain largely empty.

Example
The following example shows an exported ASCII file with an analog signal, a text signal and a digital signal, exported with a time base of 1 s.

Time	[0:0]	[0:1]_text	[0.0]
time	Sine	Text_signal	Digital_signal
sec			
30.08.2010 16:52:43.070000	0		0
30.08.2010 16:52:44.070000	0.587785		1
30.08.2010 16:52:45.070000	0.951057	"Extremum 1"	1
30.08.2010 16:52:46.070000	0.951056		1
30.08.2010 16:52:47.070000	0.587785		1
30.08.2010 16:52:48.070000	-8,74E-08		0
30.08.2010 16:52:49.070000	-0.58779		1
30.08.2010 16:52:50.070000	-0.95106	"Extremum 2"	1
30.08.2010 16:52:51.070000	-0.95106		1
30.08.2010 16:52:52.070000	-0.58779		1

The signals are displayed *ibaAnalyzer* as follows:



15.5 Export of graph signals to text file

You can also export signal values directly from a graph to a text file. The signal values are stored in the text file in columns.

Unlike with the export function via the *File* menu, you can also export the signals if the graph is length-, frequency- or inverted length-based.

Procedure

1. Open the context menu in the graph with the desired signals and select *Export signal data to text file*.

→ The dialog for exporting curves opens up.

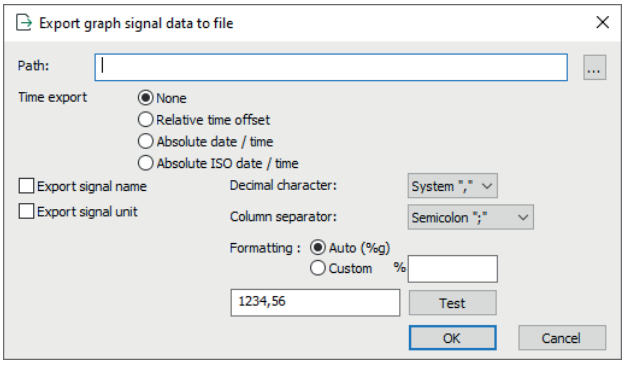
2. Enter a path and a file name for the export file.
3. Specify the other settings for the export.
4. Once you have made the desired settings, click <OK>.

→ *ibaAnalyzer* exports the signal trend values to a text file.

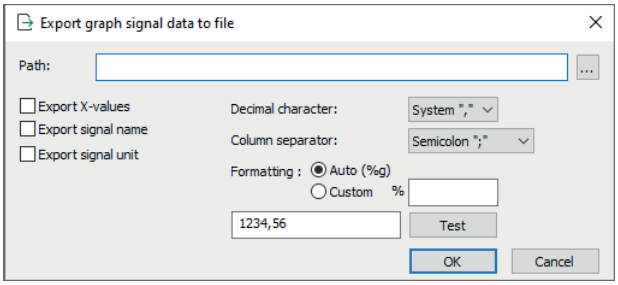
Settings in the dialog

The dialog for length-, frequency- or inverted length-based signals differs from the dialog for time-based signals.

Dialog for time-based signals



Dialog for length-, frequency- or inverted length-based signals



Path
Enter a path and a file name for the export file.

Time export mode (only for time-based signals)
Selection as described in ↗ *Text file (CSV), page 278*.

Export X-values (only for length-, frequency- or inverted length-based signals)
If you enable this option, an additional column containing the X-axis values will be created in the export file.

For all other options and settings, see ↗ *Text file (CSV), page 278*.

15.6 Export of graph signals to clipboard

You can also export signal values directly from a graph to the Windows clipboard. This makes it very easy to transfer the signal values from a graph to other software such as Microsoft Excel.

Unlike with the export function via the *File* menu, you can also export the signals if the graph is length-, frequency- or inverted length-based.

Procedure

1. Open the context menu in the graph with the desired signals and select *Export signal data to clipboard*.

→ The dialog for exporting curves opens up.

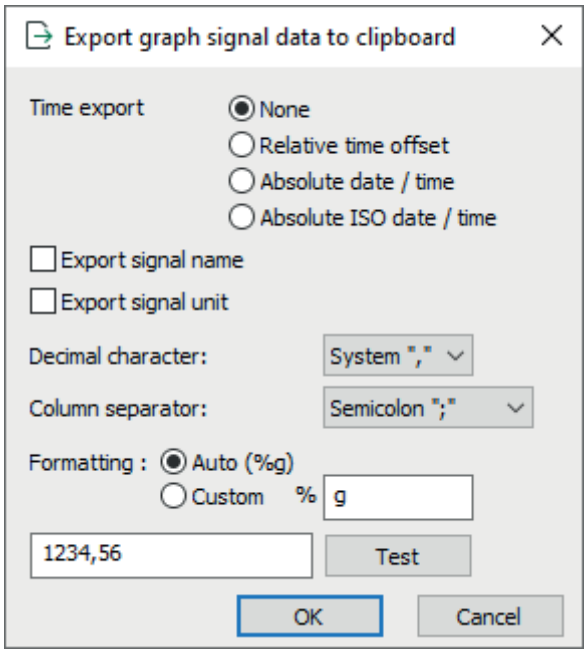
2. Specify the other settings for the export.
3. Once you have made the desired settings, click <OK>.

→ *ibaAnalyzer* exports the signal trend values to the Windows clipboard.

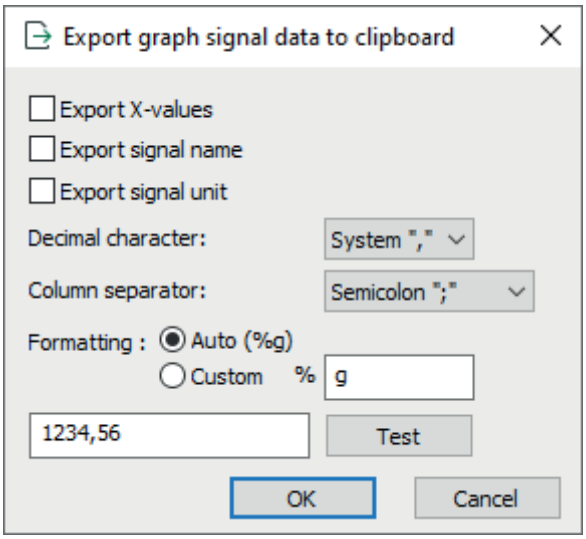
Settings in the dialog

The dialog for length-, frequency- or inverted length-based signals differs from the dialog for time-based signals.

Dialog for time-based signals



Dialog for length-, frequency- or inverted length-based signals



Time export mode (only for time-based signals)
Selection as described in ↗ *Text file (CSV), page 278*.

Export X-values (only for length-, frequency- or inverted length-based signals)
If you enable this option, an additional column containing the X-axis values will be exported to the clipboard.

For all other options and settings, see ↗ *Text file (CSV), page 278*.

16 Report generator

ibaAnalyzer-Reportgenerator is an integral part of *ibaAnalyzer* for the generation of custom reports. With the *ibaAnalyzer-Reportgenerator*, you can freely design and create analysis, quality, production, and error reports in various output formats.

You can use an analysis report to display the graphs from *ibaAnalyzer* as well as a wide variety of process data (e.g. text channels, calculated values, etc.). You can also create graphic objects (squares, circles, images, etc.), editable text fields, barcodes, diagrams, and tables.

You can create a report for a specific workpiece and then print it or export it to a file (e.g. a PDF file). It is also possible to automatically generate and output a report with *ibaDatCoordinator*.

Other documentation



A detailed description of the functions and application can be found in the *ibaAnalyzer-Reportgenerator* manual.

17 Support and contact

Support

Phone: +49 911 97282-14

Email: support@iba-ag.com

You can find the contact information for our regional offices or representatives on our website:

www.iba-ag.com/en/contact

Note

If you need support for software products, please state the number of the license container. For hardware products, please have the serial number of the device ready.

Product Security Incident Response Team (PSIRT)

Current advisories: www.iba-ag.com/en/security

Email: psirt@iba-ag.com

Contact

Headquarters
iba AG
Gebhardtstrasse 10–20
90762 Fuerth
Germany

Phone: +49 911 97282-0

Email: info@iba-ag.com

Mailing address
iba AG
Postbox 1828
D-90708 Fuerth, Germany

Delivery address
iba AG
Gebhardtstrasse 10
90762 Fuerth, Germany