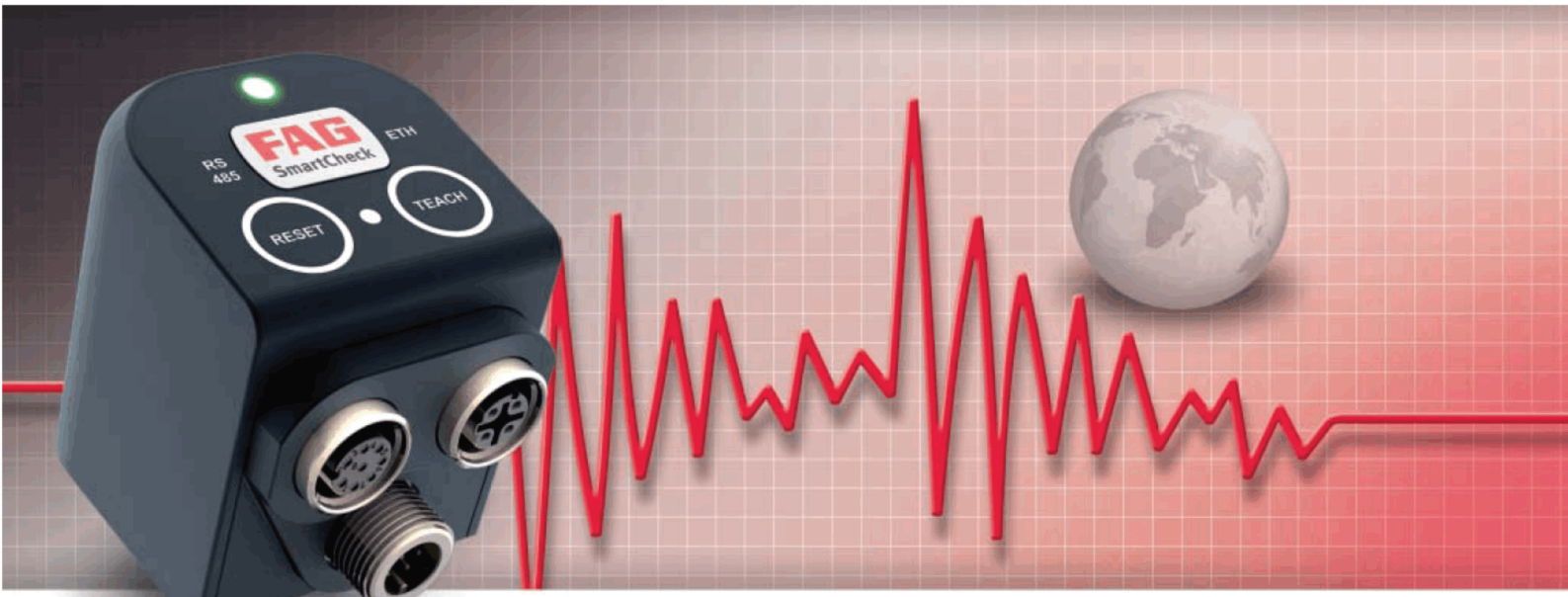
FAG



FAG SmartUtility

User manual

SCHAEFFLER

LuK INA FAG

Imprint

FAG Industrial Services GmbH
Kaiserstraße 100
52134 Herzogenrath
Germany
Telephone: +49 (0) 2407 9149 66
Fax: +49 (0) 2407 9149 59
Email: industrial-services@schaeffler.com
Website: www.schaeffler.com/services

The software uses the following third-party provider libraries under their respective licences:
Ionic.Tar, Ionic.Zip, Log4Net, Microsoft Windows API Code Pack, MongoDB, Nhibernate, NLog, Nprotobuf, Renci SSH, SharpDX, SharpSSH, Extended WPF Toolkit, ZLIB.NET
Full licence conditions for the respective libraries can be found in the software's program directory.

Version 1.10.0
Original user guide.

Contents

1 General

The FAG SmartUtility software assists you with the management functions of the FAG SmartCheck device. With this software, you perform the basic configuration of the device, load and save configurations and update the firmware. In addition, you can open the devices directly in the FAG SmartWeb software and download measurement data from the device.

You can use the FAG SmartUtility Viewer software to analyse measurement data you have downloaded from the FAG SmartCheck device. The characteristic values are shown in the user interface in a clear tree structure, the **Device overview**. You can select characteristic values from the tree structure; the corresponding data is displayed automatically as a trend in a viewer. You can check specific time signals or spectrums for this trend in two other viewers. A comprehensive range of cursor functions and settings options offer help and support.

About the FAG SmartCheck system

FAG SmartCheck is a vibration monitoring system for permanent frequency-selective monitoring. Other measurements can be captured, recorded and analysed using two integrated signals and up to three connected signals. After the analysis, the system can switch outputs and display the status using LEDs depending on user-defined alarm limits.

Inputs are available, which record additional signals, to allow the device to be integrated into a superordinate system. These signals can be used as command variables for a dependent signal analysis, e.g. to initiate time or event-controlled measurement tasks.

The FAG SmartCheck device can be used to cover a wide range of applications; the SmartCheck device can be configured to meet your own requirements using the integrated web application and FAG SmartWeb software. Multiple SmartCheck devices can be combined in a network. Regardless of the number of devices, they can be managed centrally on a PC using the FAG SmartUtility software.

With FAG SmartCheck, Schaeffler offers status monitoring that is optimised to suit your requirements.

1.1 About this guide

This guide describes how to use the FAG SmartUtility software. Read this guide carefully before using the software, and store it in a safe place.

Make sure that

- This guide is available to all users
- if the product is passed on to other users, this guide is passed on with it;
- Additions and amendments provided by the manufacturer 115 are always attached to this guide.

Further information

This software is required for the operation of the FAG SmartCheck vibration monitoring system. This system also includes the FAG SmartCheck device and the web interface and FAG SmartWeb software integrated in it, which are described in their respective manuals.

Definitions

- Product: the FAG SmartUtility software described in this manual.
- User: person or organisation with the ability to use the product

Symbols used

This symbol indicates

- Helpful additional information and
- Device settings or practical tips that will help you to perform activities more efficiently.

CAUTION

The damage that can occur is described here

Measures to prevent the damage are explained here.

Cross-reference symbol 6: This symbol refers to a page in the manual that provides further information. If you are reading the manual in PDF format on a screen, clicking the word to the left of the cross-reference symbol will take you straight to the section in question.

2 About this software

The FAG SmartUtility software is provided to you as a setup version that you need to install. You can find this version in the "FAG SmartUtility" program directory on the supplied CD-ROM. It automatically checks for the SmartUtility software requirements, such as Microsoft .NET Framework, and installs them if necessary.

Perform a software and firmware update before commissioning. You can download the latest version via the SmartUtility software or from the the downloads area of the SmartCheck microsite www.fag-smartcheck.com.

Functionality of the FAG SmartUtility software or FAG SmartUtility Light software

The following overview displays the functionality of the SmartUtility software or the SmartUtility Light software:

Wizards and functions	FAG SmartWeb	FAG SmartUtility	FAG SmartUtility Light
Status overview	✓	*	-
Measurement data viewer	✓	*	-
Real-time display	✓	*	-
Configuration	✓	*	-
User management	✓	✓ (User name, password)	-
Opening devices	-	✓	-
Download data	✓	✓	✓
Analyse data	-	✓	-
Create report	-	✓	-
Edit device settings	-	✓	✓
Downloading the configuration	-	✓	-
Sending the configuration	-	✓	-
Updating the firmware	-	✓	✓
Opening the log file directory	-	✓	✓
Opening the default directories	-	✓	✓
Manage report templates	-	✓	-
Rescan data directory	-	✓	-
Export data	-	✓	-
Import SmartWeb data	-	✓	-

* This function can be accessed via the FAG SmartUtility software.

2.1 User rights and write access

To install and operate the SmartUtility software, you require special access rights. Please contact your system administrator if you experience problems with the security requirements of your system.

User rights

To install the FAG SmartUtility software, you require administrator rights on your system.

Tip: Install the software with administrator rights and then switch back to a normal user.

Write access

The software saves settings data and log data during operation. For this reason, you require write access for the following directories for the setup version of the SmartUtility software:

Default directories

- Windows 7:
  C:\data
  C:\configuration
  C:\reports
  C:\report templates

Log file directory

- Windows 7:

C:\Users\[User name]\AppData\Roaming\Condition Monitoring

2.2 System requirements

Your system needs to meet the following requirements for you to get the most from your FAG SmartUtility software:

General system requirements

Windows 7 (32/64 Bit) Service Pack 1

The system requirements recommended by Microsoft must be met as a minimum:

- Dual-core processor
- 2 GB RAM (recommended: 4 GB RAM)
- 16 GB of hard disc space available
- DirectX 9 graphics device with WDDM 1.0 driver or higher

In addition:

- Screen resolution: 1024x768 (pixels) at 96 dpi and normal font size (recommended: 1280x800 or higher)
- Disk space for software: at least 40 MB free disk space
- Mouse: three-button mouse recommended

In addition, you require disk space for the data downloaded from the SmartCheck device. This can vary considerably according to the application type and depending on your measurement tasks.

Other requirements

- Microsoft .NET Framework 4.6.1
- DirectX 9
- VC++ Runtime 2010 and 2012
- Standard browser: Mozilla Firefox ESR 38 (recommended), Internet Explorer 11 (Internet Explorer 9 is not recommended for performance reasons)

The SmartUtility software checks whether the Microsoft .Net Framework component is on the computer and installs it automatically if necessary. If you run the installation without an Internet connection, the components are set up in English by default. The SmartUtility software is also then fully functional.

Requirements for connection to the computer

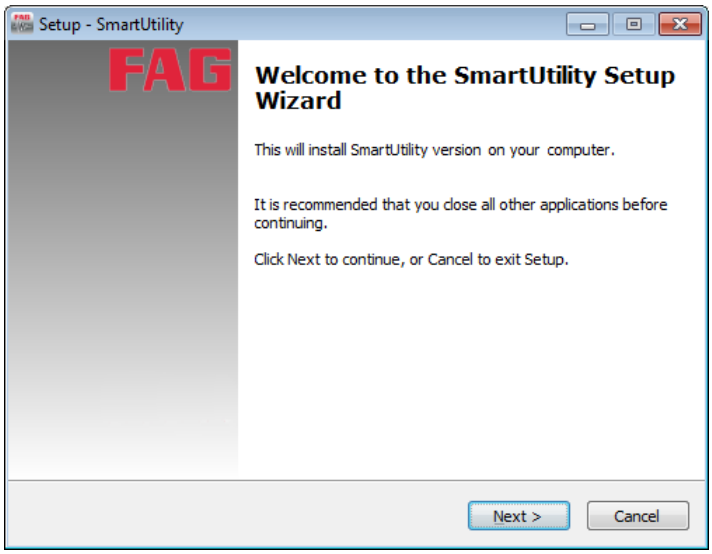
- The UDP communication protocol must be enabled on ports 19000 and 19001 in existing firewalls in your network
- The ports must also be enabled for UDP broadcasts. This can be done in the Windows 7 firewall via **Allow unicast response**.
- If the SmartCheck device has not been assigned an address via DHCP, it will have the IP address 192.168.1.100 by default. In this case, the IP address of your computer must be within the 192.168.1.x range.

Please contact your system administrator if you experience problems with the network settings.

- If an SmartCheck device cannot be opened in the browser, clear the browser cache and try again.
- If you receive a message stating that cookies are not accepted, please allow the use of cookies or enter the IP address of the SmartCheck device as an exception.
  For further information, please see the "Introduction to network basics" section in the manual.

2.3 Installing the software

Open the FAG SmartUtility program directory on the supplied CD-ROM. Double-click the following file: **SmartUtility.exe**. This starts a wizard, which will guide you through the individual installation steps:



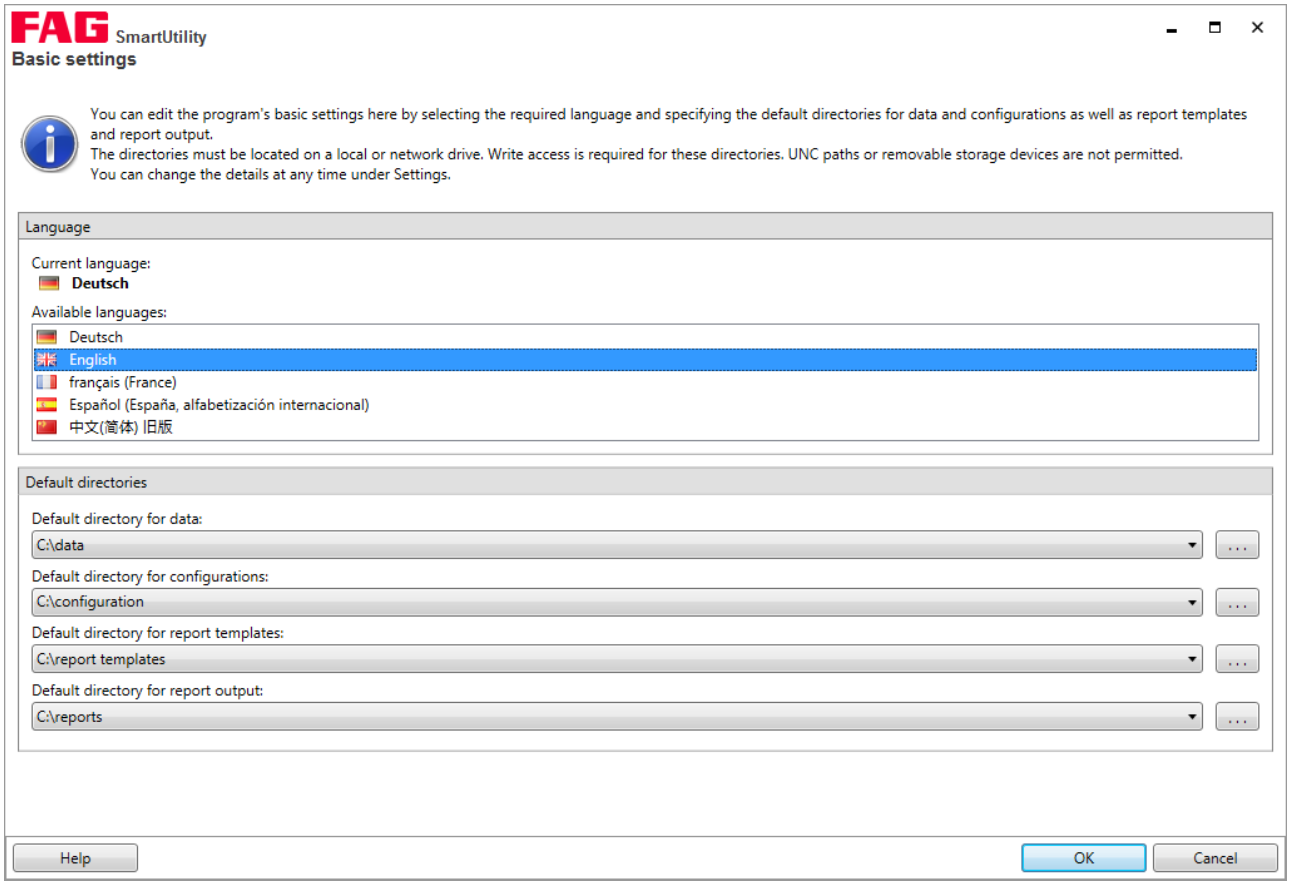
This includes—depending on your system configuration—the following steps:

- **Select installation language:** Here you can select the language to be used during the installation of the SmartUtility software.
- **Install Microsoft .NET Framework:** If the wizard does not find these components on your computer, it will start installing them after confirmation.
- **Select target directory:** You can either accept the suggested directory or enter a different directory. The FAG SmartUtility software is saved in the following directory by default: C:\Programs\FAGSmartUtility.
- **Select folder in Start menu:** Here you have the option to change the SmartUtility entry in the Start menu to suit your requirements.
- **Restart computer:** In the final step of the wizard, you have the option to restart the computer immediately or to carry out the restart at another time. We recommend always restarting the computer if the DirectX or Microsoft .NET components have been installed.

Links are created in the start menu and on the desktop during the installation; these links can be used to start the FAG SmartUtility software. When you start the software for the first time after installation, the **Basic settings** wizard opens and guides you through the most important configurations in three steps. Details on this can be found under **First start-up** 10.

3 First start-up

You can start the FAG SmartUtility software via links in the Start menu and on the desktop. During the first start-up, the **Basic settings** page opens first. This page appears in the language that you selected for the installation:



Here you can define the following basic configurations:

- **Language:** Here you can specify the language in which the user interface is displayed. Click on an available language.
- **Default directories:** Here you can specify the directories to be used by default for downloaded data, configurations, report templates and report output.

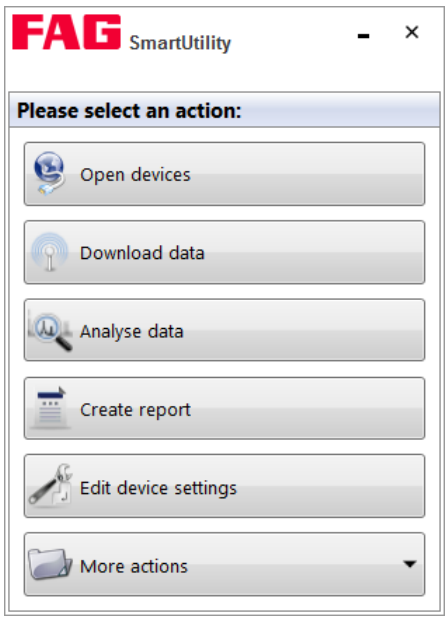
- Set up the default directories of the SmartUtility software on a central computer that is available at all times. The analysis of downloaded data in the Viewer software is only useful when a trend for all the collected data can be created. To this end, it is necessary that all data is collected in the same directory.
- Even if several users access the SmartUtility software, ensure that the data is always downloaded to the same default directory for data.

When you close the **Basic settings** page with **OK**, the SmartUtility software is opened directly and all functions are available. When you close the page with **Cancel**, the default settings are adopted automatically. You will then need to start the program again via links in the Start menu and on the desktop.

You can change any program settings that you specify on the **Basic settings** page at a later date under **Settings** 108.

4 The user interface at a glance

The SmartUtility software user interface consists of buttons that are used to open the wizards for the most important actions. In addition, you can find the **More actions** list box here, which includes additional functions and the settings for the SmartUtility software:



The wizards guide you through the individual steps with the aid of detailed instructions. For this reason, you will only find general information and important additional information regarding the wizards under **Wizards and functions** 12. The additional **Settings** 108, which are not wizard-supported, are described in more detail. Here you can specify important settings for SmartUtility.

The – button minimises the SmartUtility window. To exit the software, click on ✕. Only exit the software after a process has been fully completed; otherwise, measurement data could be lost.

5 Wizards and functions

If you want to use the FAG SmartUtility software to access the FAG SmartCheck device, the following requirements must be met:

- The SmartCheck device must be started and must be in measuring mode.
- The SmartCheck device must be on the network or directly connected to your computer via Ethernet cable.
- The UDP communication protocol must be enabled on ports 19000 and 19001 in existing firewalls in your network
- If the SmartCheck device has not been assigned an address via DHCP, it will have the IP address 192.168.1.100 by default. In this case, the IP address of your computer must be within the 192.168.1.x range.

Further information can be found in the FAG SmartCheck user documentation. This documentation can be found on the supplied CD ROM.

Please contact your system administrator if you experience problems with the network settings.

Accessing the SmartCheck devices in the wizards

The name of every SmartCheck device is **FAG** SmartCheck by default. If you want to integrate multiple SmartCheck devices into your system, it is important that you give each device a unique name. You can only identify devices without a unique name in the Wizard device list 12 via the IP address.

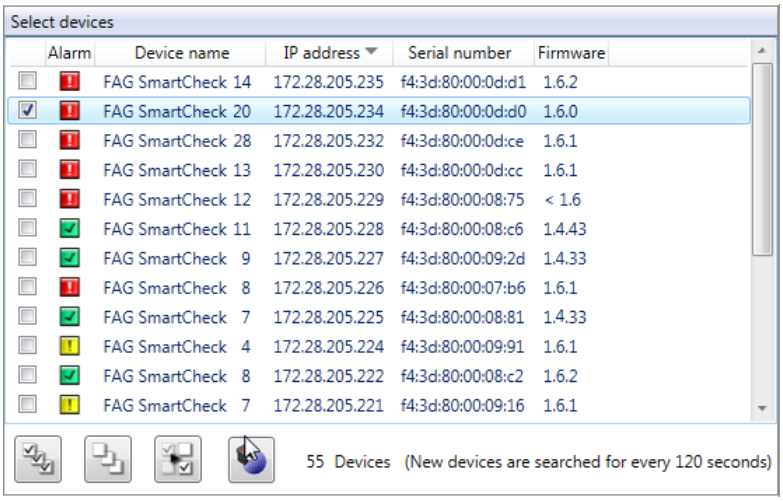
Configure the name using the **Edit device settings** 86 wizard. Within the wizard, you can find your system's SmartCheck devices either in the automatically generated list 12 or by manually entering 14 the network parameters of the corresponding SmartCheck device.

If you have activated user management in the FAG SmartWeb software, you also have to specify the user name and password 111 for each SmartCheck device in the **Settings** 108. You can find further information on user management in the FAG SmartWeb user documentation on the supplied CD-ROM.

If a device is started via the FAG SmartWeb software in the Maintenance System, you will be unable to open it using SmartUtility, download data or update the firmware. Devices that have been started in the Maintenance System are marked with the alarm symbol M in the SmartUtility wizards. You will also receive an error message: "A communication error occurred: The operation could not be executed. Check whether the device's Maintenance System has started."

List of FAG SmartCheck devices

In the first step - the **Select devices** step - all SmartUtility software wizards display the SmartCheck devices found on the network:



The following information and functions are to be found here:

- The list columns provide information on the alarm status, name, IP address, serial number and firmware version of the individual SmartCheck devices.

- The alarm symbols show you at a glance:
  - Green: There is no alarm
  - Yellow: One or several characteristic values have triggered a pre-alarm
  - Red: One or several characteristic values have triggered a main alarm
  - White: The alarm status is still unknown, e.g. because the SmartCheck device has not yet carried out any measurements.
  - : The device is in the Maintenance System (maintenance mode).
- The alarm symbols are updated on a regular basis.
- You can set any column of the list as a sort criterion by clicking in the column title. Click a second time to change the sort order, i.e. from ascending to descending or vice-versa. The current sort order is displayed by the symbols ▲ for ascending and ▼ for descending.
- Sorting by column is retained, even if you close the wizard and re-open it.
- You can adjust the column width.

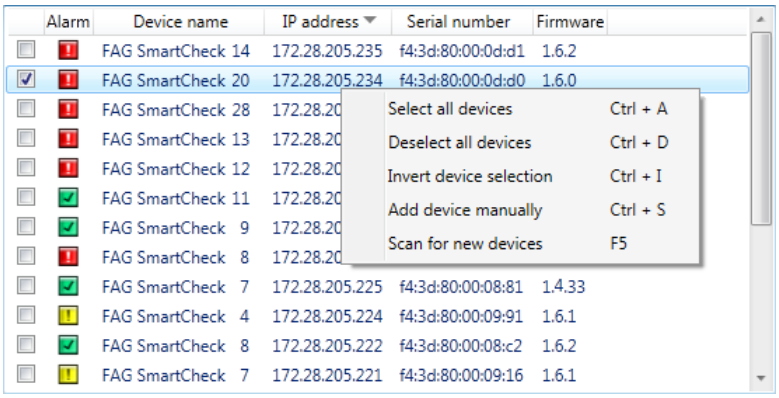
- If the list does not contain any entries or a device is missing, you can update the list. To do so, press the **F5** key or add the device manually (see below). If there are still no devices displayed, make sure:
  - that the device is in measuring mode and connected to the network;
  - that you are connected to the network;
  - that the settings in the SmartUtility software are correct.
- Manually added devices are then displayed even when unavailable. They are then highlighted in light grey.

Select FAG SmartCheck devices

In the wizards, you can select one or more SmartCheck devices for each action. To do so, activate the tick ☑ in front of the respective device name in the list.

Below the list, you will also find symbols to filter the selection:

Symbol	Explanation
	Here you can select all the devices.
	Here you can deselect all the devices.
	Here you can invert the selection of the devices.
	Here you can select all the devices for which a pre or main alarm has been triggered. This option is only available for functions that work with files on the hard drive, such as **Analyse data** and **Create report**.
	Here you can select all the devices for which a main alarm has been triggered. This option is only available for functions that work with files on the hard drive, such as **Analyse data** and **Create report**.
	Here you can manually add a device via the TCP/IP address or via a UNC path.
Context menu	Right-click in the area of the device list to use the options described above via the context menu: **Select all devices, Deselect all devices, Invert device selection** and **Add device manually**. There, you will also find the command **Scan for new devices**, with which you can update the device list:

Symbol	Explanation
	

The following hotkeys can also be used for the options described above:

Ctrl+A Select all devices
Ctrl+D Deselect all devices
Ctrl+I Invert device selection
Ctrl+S Add device manually
F5 Scan for new devices

Updating the device list

The list of SmartCheck devices is updated automatically within a specified search interval. You can adjust this in the Settings 109. You can also update the list by pressing the **F5** key.

Add device manually

This function is particularly important if a SmartCheck device cannot be found via UDP. A possible reason for this is that the device is behind a router. The TCP protocol is used to find manually added devices for the device list. If a proxy has been entered in Internet Options on the Control Panel, this will also be used.

To add a device manually, click in the **Select device** step:

Here you have the following options:

- Enter the **address** (IP address or UNC path) and the relevant **port number** of the SmartCheck device.

- If the device is password-protected, specify the **user name** and **password**.
- You can also activate the **Use default user and password** option to log on with your user data.
- Click **Verify** to test the connection to the SmartCheck device.

As soon as the connection to the device has been established, the **serial number** and **device name** are displayed.

Manually input devices are retained in the SmartUtility software and are also available in the other wizards of the same software.

If you no longer need to use a device, you can remove it from the device list. Further information on this can be found in the Settings 111 section.

Further information on the password protection of SmartCheck devices can be found in the **User management** chapter in the FAG SmartWeb user documentation.

5.1 Open devices

With this wizard, you can open the FAG SmartCheck devices in your default browser with FAG SmartWeb. Proceed as follows:

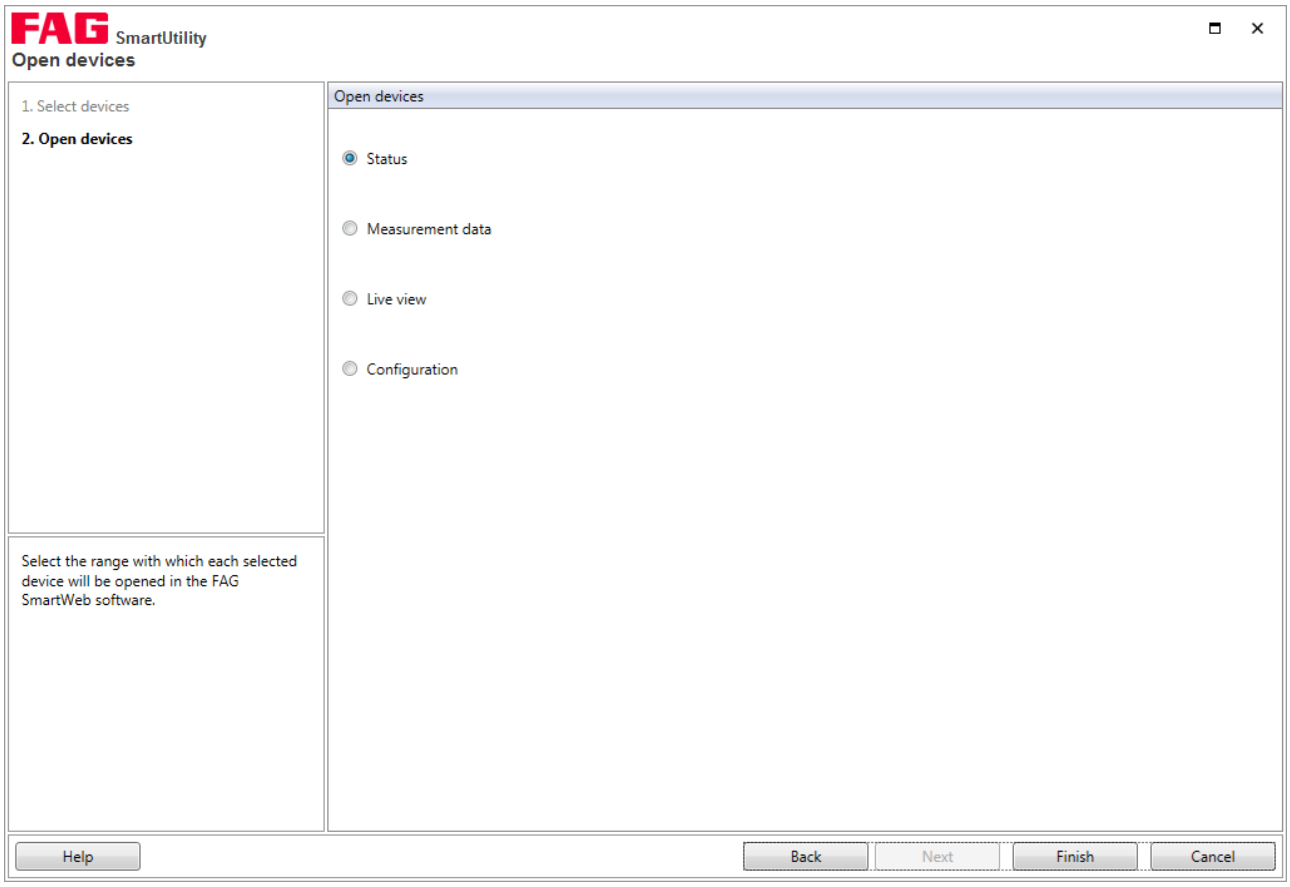
Step 1:

Select the required SmartCheck device or devices.

Step 2:

In the next step, select the action with which each selected device should report in the FAG SmartWeb software. This determines the page at which the device's internal SmartWeb software should open. Each selected SmartCheck device is opened in its own tab in the default browser:

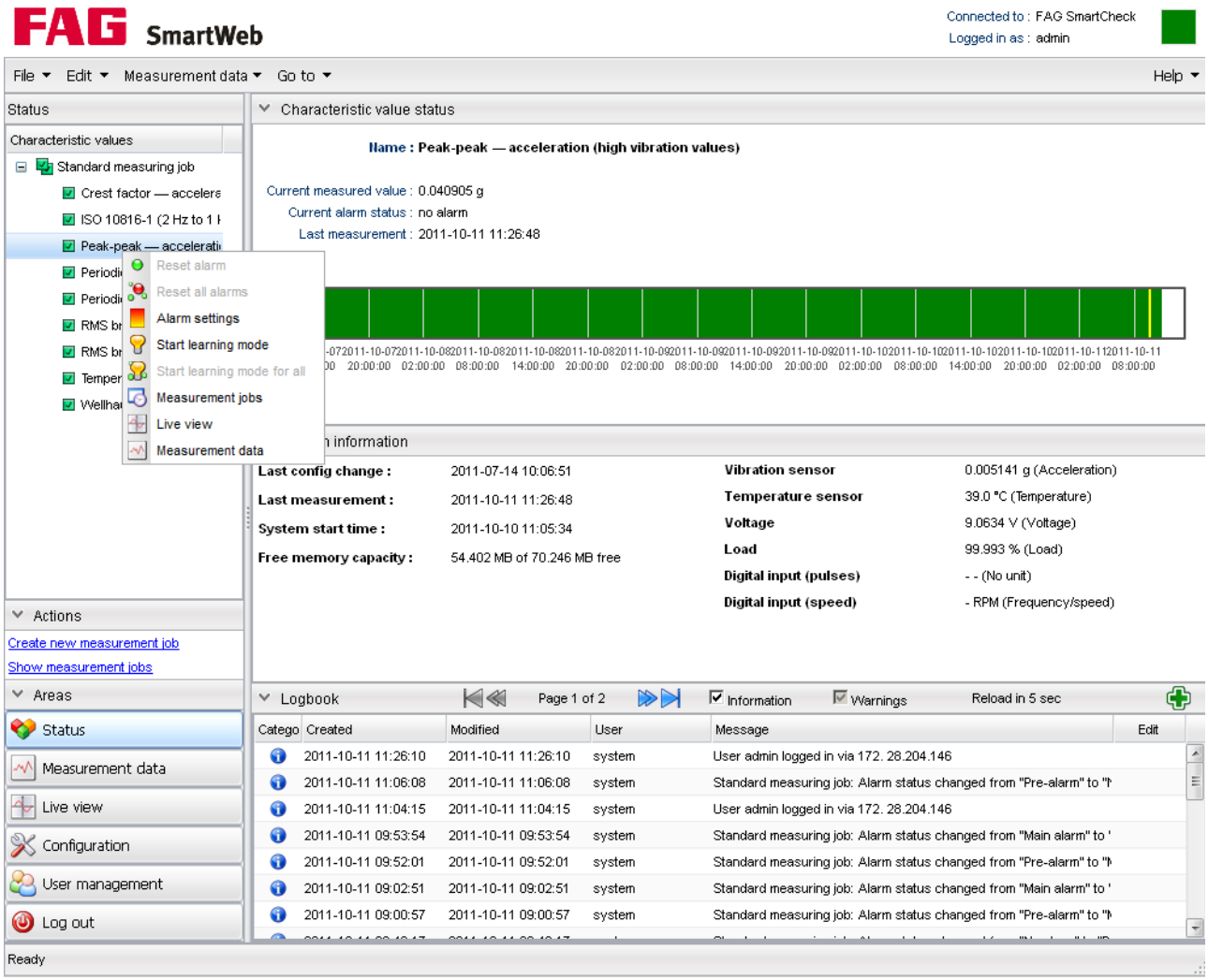
- **Status:** Here you can find a general overview of the status of the SmartCheck device.
- **Measurement data:** Here you can view the trends and time signals of a characteristic value and carry out an initial analysis.
- **Live view:** Here you can view the relevant signal for each input and the scaling factors created for them.
- **Configuration:** Here you can define basic settings for your SmartCheck device and specify measurement jobs.



If you cannot access the SmartWeb software, please check whether all requirements for starting up the software have been met. For more information, please see the section entitled **First start-up** in the handbook for the SmartWeb software.

Result:

The following screenshot shows a sample status page of a SmartCheck device in the SmartWeb software:



Further information on the application of the FAG SmartWeb software can be found in the FAG SmartWeb user documentation. This is available as a PDF on the supplied CD-ROM.

If you are working on a computer with several SmartCheck devices, these devices should have identical firmware versions installed. If different firmware versions are installed, this can have undesired effects within the browser.

5.2 Download data

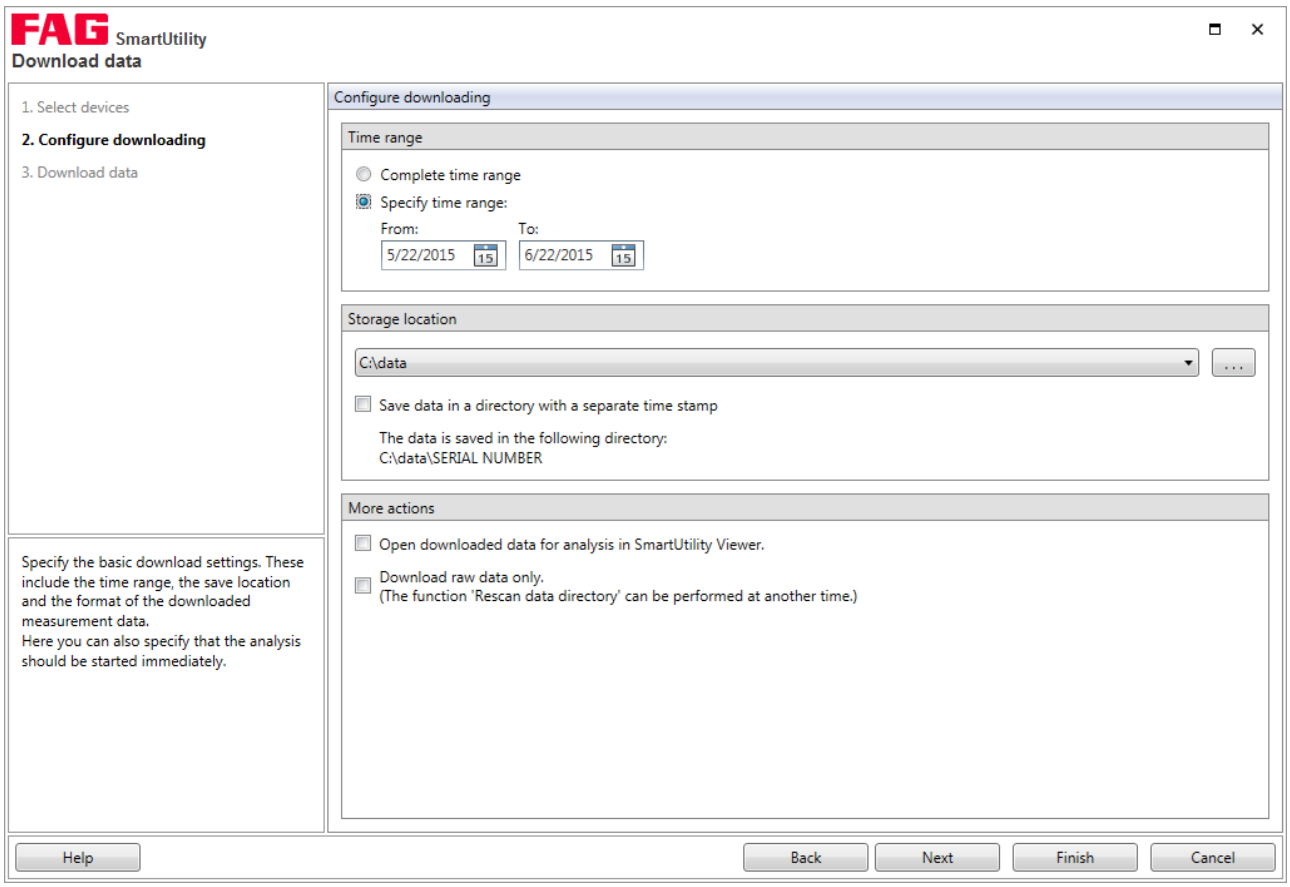
With this wizard, you can download the measurement data from the FAG SmartCheck devices. Proceed as follows:

Step 1:

Select the required SmartCheck device or devices from which you want to download the data.

Step 2:

Next, determine the time range for which the data is to be downloaded. Also specify the save location and format for the data:



Here you have the following options:

Time range

This is where you determine the time range for which the measurement data is to be downloaded:

- **Complete time range:** All measurement data is downloaded.
- **Set time range:** Only the measurement data for the selected time range is downloaded. If you activate this option, the default setting is one month (i.e. the measurement data for the past month up to today is downloaded).
  You can enter the dates for your desired time range in the **From** and **To** fields or select the dates from the calendar.

Storage location

Here you determine the directory in which the measurement data is to be saved. You have the following options:

- The list shows the default directory for data by default. This directory is specified either when you first launch the software or subsequently under **More actions > Settings > General** 108.
- Click ... to select another directory and specify this as the save location.
- In the selected directory, a subdirectory is automatically and additionally created with the serial number of the appropriate SmartCheck device for the data. If you also wish to sort by time stamp, activate the **Save data in a directory with a separate time stamp** option.
  The time stamp directory appears in the directory tree one level above the serial number directory.
- Finally, you will find a directory path that matches your settings. This path is updated directly each time the settings are changed.
  The terms time stamp and serial number are simply used here as placeholders and will be replaced by the actual values.

More actions

This is where you determine the format in which the measurement data is to be downloaded:

- **Open downloaded data in the** SmartUtility Viewer **for analysis**: The data is downloaded, converted for analysis and shown directly in the **SmartUtility Viewer**.
- **Download raw data only**: The data is downloaded as raw data and saved. To

analyse this data, you need to convert it in a subsequent step using the **More actions > Rescan data directory** 104 wizard.
This option is provided in case you wish to speed up the download.

- If both options are deactivated, the measurement data is downloaded, converted and saved in the specified directory. It is not opened in SmartUtility Viewer for analysis.

Step 3:

The data is downloaded. Wait until the process is fully completed.

CAUTION

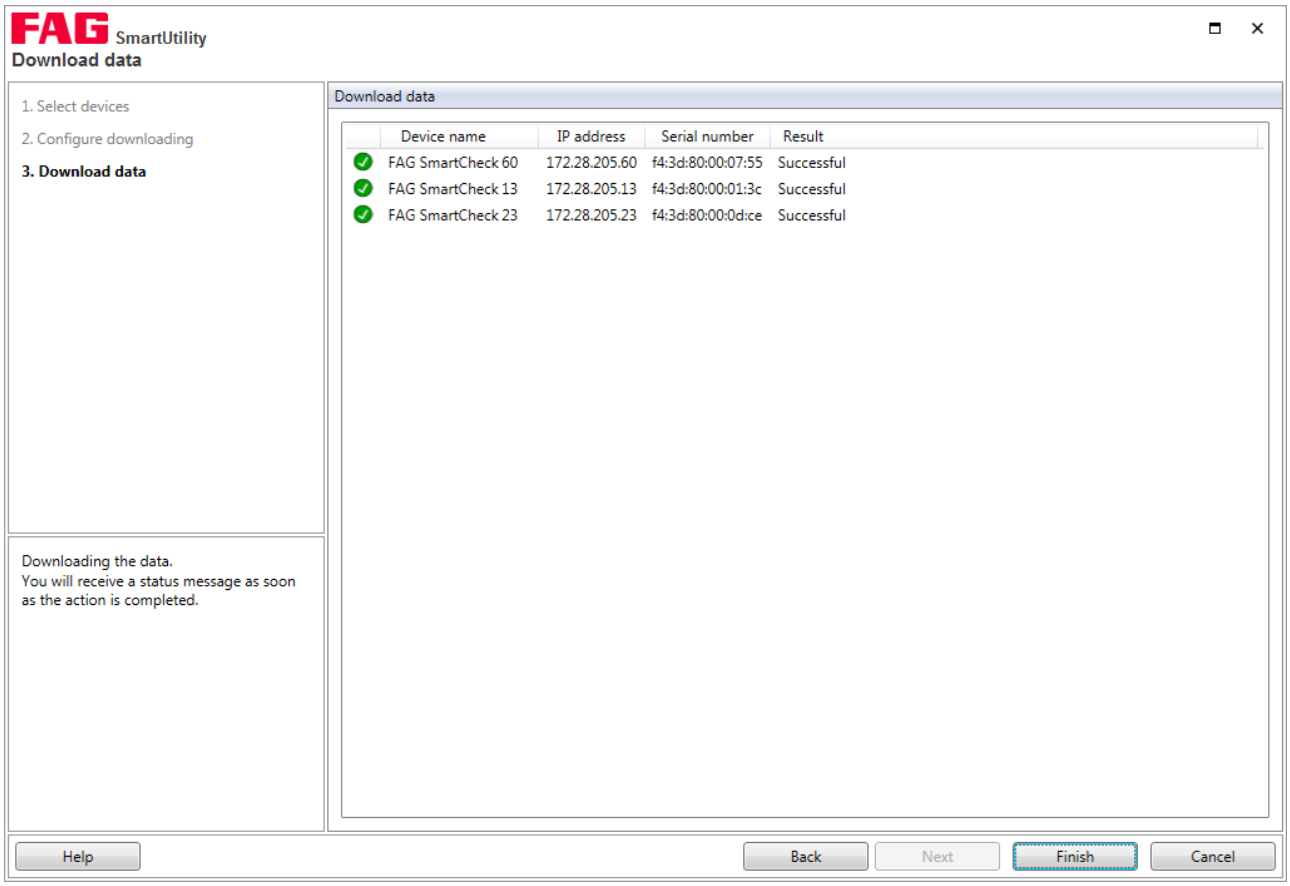
Cancelling the process prematurely may result in the loss of data!

If you cancel the process, all the saved measurement data and trends may be damaged.

Conversion of the measurement data into another format and calculation of the trends can take some time, depending on the configuration. Always wait until the process is finished. If you cancel the process prematurely, all the measurement data collected until that point may be damaged. This measurement data can then no longer be analysed in the SmartUtility Viewer.

Result:

As soon as the action is completed, you will receive a corresponding status message. The dialogue looks like this:



The downloaded measurement data can be found in the Default directory for data 108 in the SmartUtility software or in the directory that you specified in **Step 2**. An individual subdirectory is created for each SmartCheck device. The name of this subdirectory is the serial number of the device.

If you want to burn the downloaded measurement data onto a CD ROM or DVD, you will need to compress the data beforehand.

5.3 Analyse data

With this wizard, you can open the measurement data you have downloaded from SmartCheck devices in order to analyse it in the SmartUtility Viewer software.

This version of the FAG SmartUtility Viewer software includes the introduction of a new file format. Data in the previous file format can no longer be loaded.

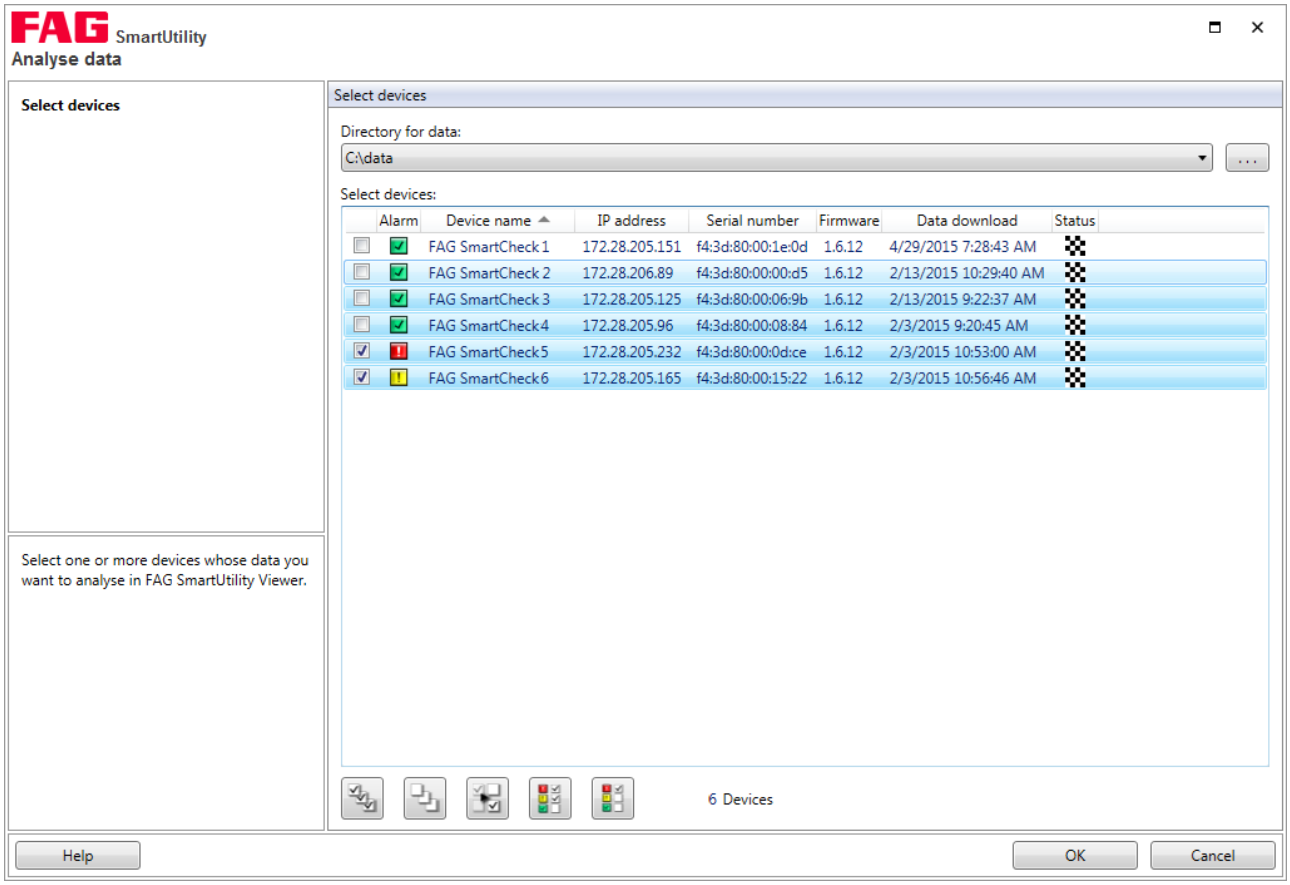
You have the following options to continue working with this data:

- Open the FAG SmartUtility software and perform the action **Rescan data directory**; this action converts data that has already been downloaded into the new file format, without downloading new data from the FAG SmartCheck device. Details on this can be found in the "Rescan data directory" section in the manual for the FAG SmartUtility software.
- Reimport the required data from the SmartCheck device.

Proceed as follows:

Step:

1. Select the directory in which the data is located. This may be the default directory 108 or a directory you have selected yourself.
2. Select the SmartCheck device that contains the data you want to analyse. You can also select multiple devices:



In the **Status** column you will find one of several possible data status symbols. If you move the mouse over it, you will obtain information about the status of the data. The following status messages are possible:

Symbol	Tooltip text	Description
	Data not yet converted to the new format.	The data comes from a device with firmware version 1.0.x or 1.4.x. The SmartUtility Viewer software cannot be opened with this data.
	In this directory there is data that has not yet been	The directory contains a database of version 1.6.x,

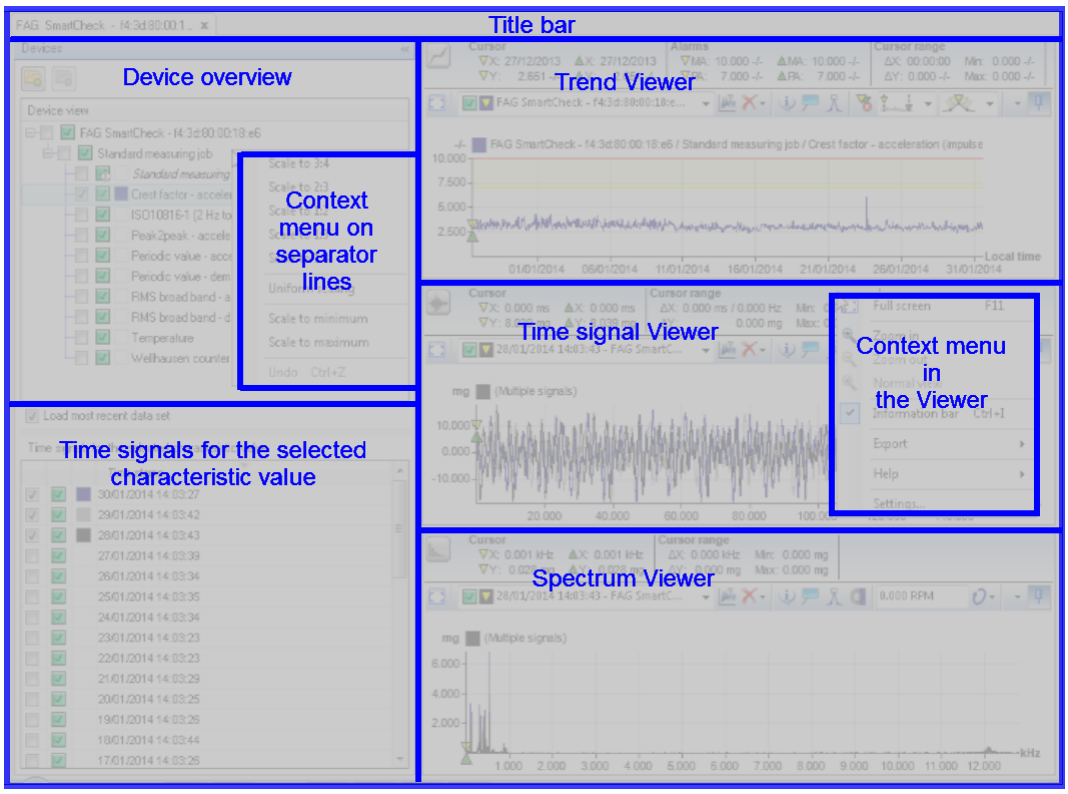
	converted to the new format. To ensure that the data can be shown in the Viewer, please perform the "Rescan data directory" action under "More actions".	the data from which can be shown without issue in the SmartUtility Viewer software. In addition, the directory also contains *.scd1 files and/or data from a device with the firmware version 1.0.x or 1.4.x . This data is not included in the 1.6.x database. You must convert this data to open it with the SmartUtility Viewer software.
	Data not yet converted to the new format. As the alarm information is no longer available, the alarm status is reset upon conversion.	The directory contains *.scd1 files or data from a device with firmware version 1.0.x or 1.4.x . The SmartUtility Viewer software cannot be opened with this data. The alarm status is lost during conversion.
	The data has already been converted to the new format without the alarm status. The alarm status cannot be adopted, as the original data is no longer available. The alarm status has been reset automatically. A repeat conversion is not possible.	The data comes from a device with the firmware version 1.0.x or 1.4.x and has already been converted. A **VHF** folder does not exist, so there is no information about the alarm status. However, a database for version 1.6.x is available, meaning that the data can be opened in the SmartUtility Viewer software.
	The data has already been converted to the new format, but the alarm status was not adopted. A repeat conversion is possible, during which the alarm status would be adopted.	The data comes from a device with the firmware version 1.0.x or 1.4.x and has already been converted. The **VHF** folder is available, so there is information about the alarm status. A database for version 1.6.x is also available, meaning that the data can be opened in the SmartUtility Viewer software. During a conversion, the alarm status is also updated.
	The data has been successfully converted to the new format. A repeat conversion is not necessary.	The data can be opened without issue and including the alarm status in the SmartUtility Viewer software.

Result:

After you have clicked on **Finish** the data is opened directly in the SmartUtility Viewer software and is available for analysis.

5.3.1 User interface overview

The user interface of the FAG SmartUtility Viewer software can be divided as follows:



Here you have the following options:

Title bar

The buttons to minimise, maximise and close the program window are located in the right-hand corner.

Device overview

This is a list of the characteristic values whose measurement data you have downloaded via the FAG SmartUtility software program and opened for analysis. If you activate a checkmark, i.e. highlight the characteristic value, the associated trend is displayed in the Trend Viewer. If you click on a characteristic value, i.e. select the characteristic value, the associated time signals are displayed bottom left under **Time signals for the selected characteristic value**.

You will find further information on the device overview and its functions in the **Device overview** 22 section.

Time signals for the selected characteristic value

The content of this list depends on which characteristic value you have selected in the **Device overview**. You will always see the time signals for the selected characteristic value here. Details about working with the time signals overview can be found in the **Time signals for the selected characteristic value** 26 section.

Trend Viewer

You can see the trend for the characteristic value you activated with a checkmark in the **Device overview** in the Trend Viewer. You can learn how to work with the Trend Viewer in the relevant sections in the **Viewer and diagram** 29 chapter.

Time Signal Viewer

In the Time Signal Viewer, you will see the time signals that you have activated with a checkmark in the list of **Time signals for the selected characteristic value**. You can learn how to work with the Time Signal Viewer in the relevant sections in the **Viewer and diagram** 29 chapter.

Spectrum Viewer

In the Spectrum Viewer, you will see the spectrums for the time signals that you have activated with a checkmark in the list of **Time signals for the selected characteristic value**. You can learn how to work with the Spectrum Viewer in the relevant sections in the **Viewer and diagram** 29 chapter.

Context menu in the viewer

You can right-click in any Viewer to open a context menu with the following functions:

Full screen Use this command to switch the Viewer to full screen size 29 or to return it to the

	FAG SmartUtility Viewer software interface.
Zoom in	Use this command to zoom gradually 29 into the diagram for the active Viewer.
Zoom out	This command undoes the last zoom step 29 in the diagram for the active Viewer.
Normal view	This command undoes all of the zoom steps 29 in the diagram for the active Viewer.
Information bar	Use this command to hide or show the viewer's information bar.
Help	This is where you can find SmartUtility Viewer **Help**; the **About** command also offers detailed information on the version of the FAG SmartUtility Viewer software.
Export	Use these commands to save or copy a diagram in different formats. You will find further information on these commands in the **Exporting diagrams** 68 section.
Settings	This command opens a dialogue box with numerous settings options. You can use these to adjust the FAG SmartUtility Viewer software to suit your needs. You will find further information in the **Change program settings** 72 section.

Context menu on separator lines

Right-click on the line separating two areas from each other to call up a context menu. This contains functions that allow you to define the size of each area and their relation to each other. The functions relate to the area above a horizontal separator line, or the area to the left of a vertical separator line. The only exception to this rule is the function **Apply uniform scaling**: if you use this function on a horizontal separator line between viewers, it will apply to all three viewers, i.e. display all three with the same size.

The following functions are available to you:

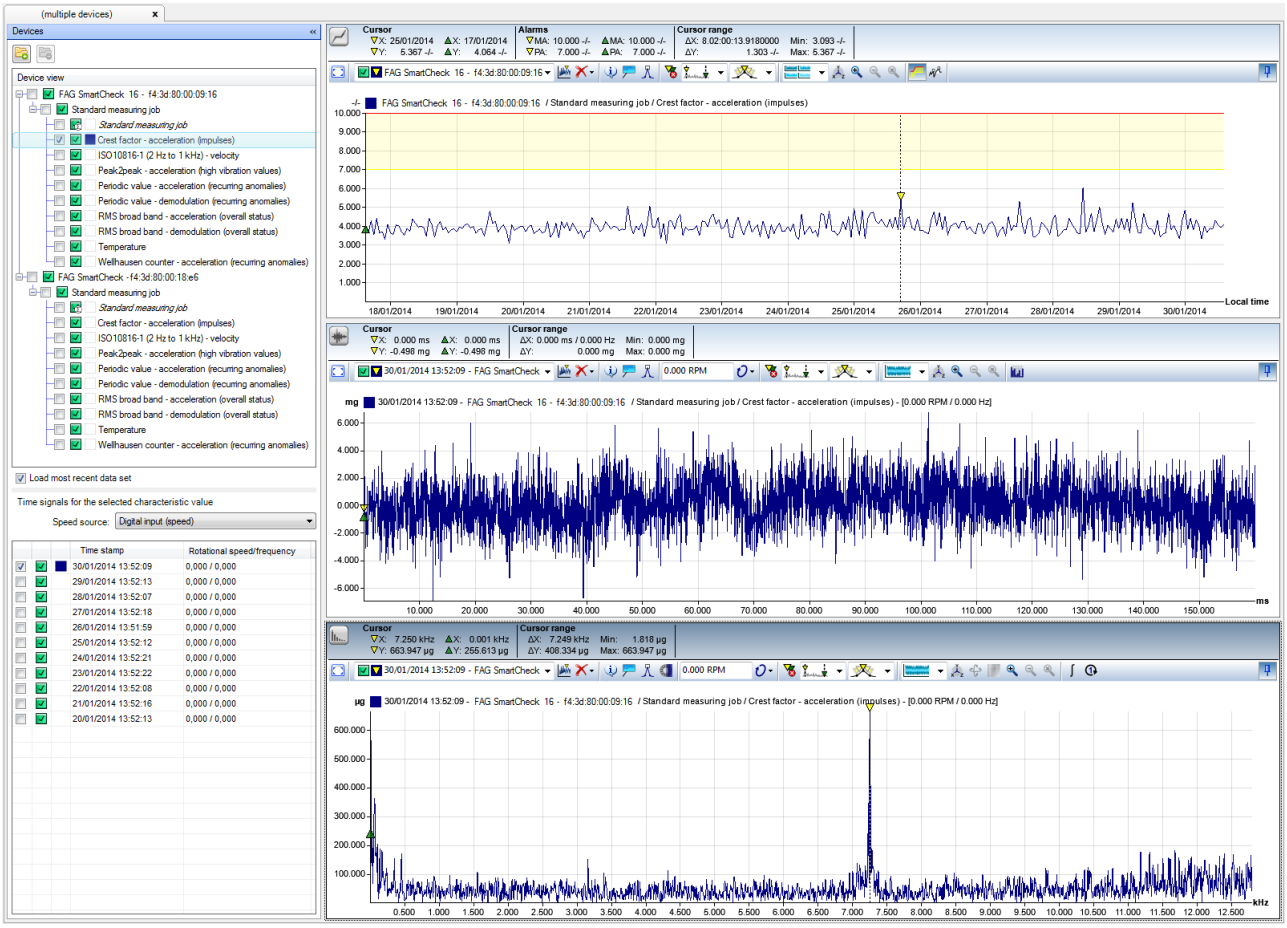
- **Scale to 3:4**: This scales the area to 3/4 of the available space.
- **Scale to 2:3**: This scales the area to 2/3 of the available space.
- **Scale to 1:2**: This scales the area to 1/2 of the available space.
- **Scale to 1:3**: This scales the area to 1/3 of the available space.
- **Scale to 1:4**: This scales the area to 1/4 of the available space.
- **Scale equally**: This scales neighbouring areas to equal sizes.
- **Scale to maximum/minimum**: This scales the area to the maximum possible size.
- **Undo (CTRL+Z)**: This key combination undoes the last change.

You can also scale the areas manually:

1. Move the mouse over the separator line that you want to move.
2. Once the mouse pointer changes into a double-headed arrow , click and hold the left mouse button and pull the line to the new position.

5.3.2 Device overview

Once you have started the FAG SmartUtility Viewer software from within the FAG SmartUtility software, you will find the **device overview** with characteristic values at the top left. Both the list of **Time signals for the selected characteristic value** and the viewers are empty. If you activate a checkbox in front of a characteristic value, the trend is loaded in the Trend Viewer. In addition, the list of **Time signals for the selected characteristic value** fills up. The latest time signal is shown in the Time Signal and Spectrum Viewer, if the option **Load most recent data set** has been activated:



You will find the following functions and information in the **device overview**:

Measurement value details:

The individual **device overview** levels offer detailed information on the measurement values in question:

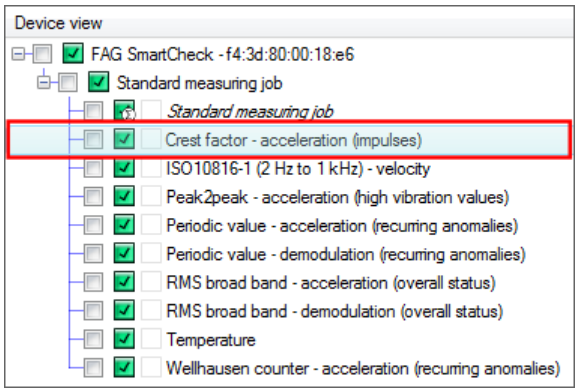
- **Top level:** Contains the name and serial number of the FAG SmartCheck device that provided the data.
- **Middle level:**
  Contains the name of the **measurement jobs**, which have saved the data.

  On this level you will also find the section **Measurement triggers and conditions**, if data has been saved for this.
- **Last level:**
  In the first instance, you will find the alarm characteristic value under **Measurement jobs** that you have defined in your measurement job. This is followed by the characteristic values of your measurement job, i.e. the characteristic values that were used to calculate the overriding alarm characteristic values.

  You will find the individual triggers and conditions under **Measurement triggers and conditions**.

Select characteristic value

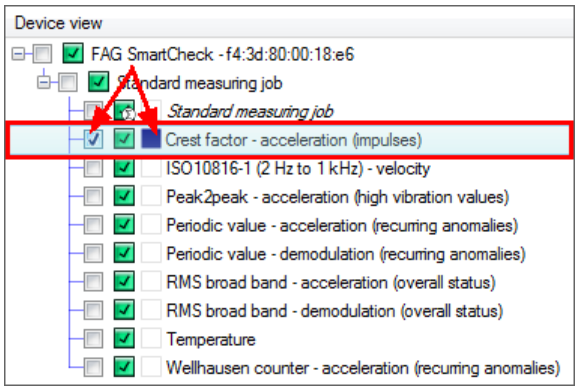
Select a characteristic value by clicking on the characteristic value in the **Device overview**. The characteristic value is then highlighted in colour:



The time signals for the characteristic value highlighted in colour are shown in the list **Time signals for the selected characteristic value**. To load time signals into the Viewer you must activate them with a checkbox in this list. You can find details on this in the **Time signals for the selected characteristic value** 26 section.

Activate characteristic value

You activate a characteristic value by positioning a checkmark in front of it. The characteristic value is then marked with the alarm symbol and highlighted in colour , based on which you can also identify it in the Trend Viewer diagram:



You can find the following information:

- The trend for the characteristic value is loaded in the Trend Viewer.
- The last available time signal opens up in the Time Signal Viewer, provided the **Load most recent data set** option has been activated. The relevant spectrum is shown in the Spectrum Viewer.
- You can load more trends in the Trend Viewer by activating more characteristic values.
- The time signals for the characteristic value are shown in the list of **Time signals for the selected characteristic value**, provided the characteristic value has also been selected, i.e. highlighted in colour.
- As soon as you select a different characteristic value, its time signals are shown in the list. You can find details on this in the **Time signals for the selected characteristic value** 26 **section.**
- If you deactivate the **Load most recent data set** option, the latest time signals for the activated characteristic values are not loaded.

Alarm status

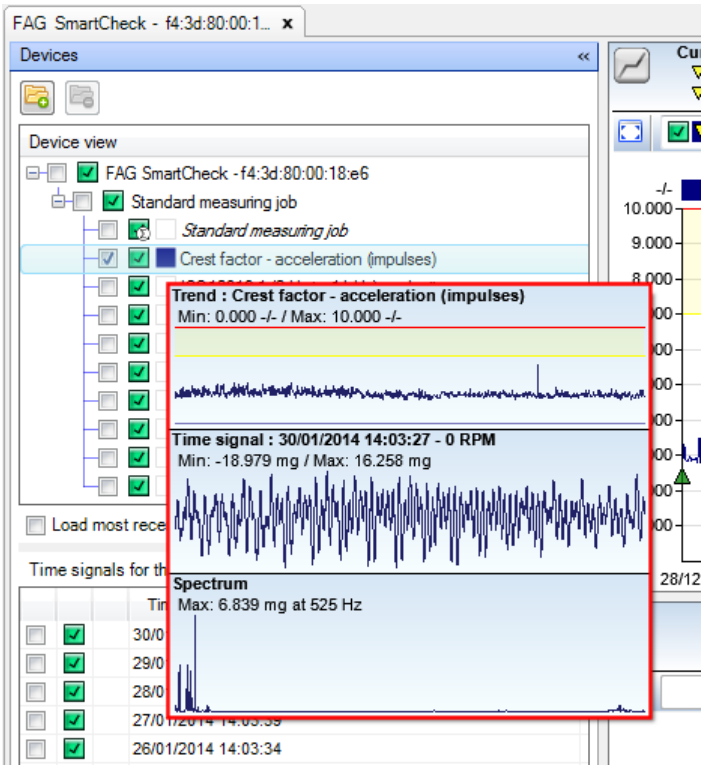
This symbol displays the alarm status on every level. The symbol is shown in different colours depending on the status:

- : The measurement was taken but the device was still in the learning phase.
- : No alarm
- : Pre-alarm
- : Main alarm

On the level of individual characteristic values, the symbol relates only to the characteristic value in question; in the higher levels its relates to all subordinate characteristic values; the program always shows the most

critical alarm status. For instance, if there is a characteristic value with a main alarm, then the main alarm status is set for the entire measurement job.

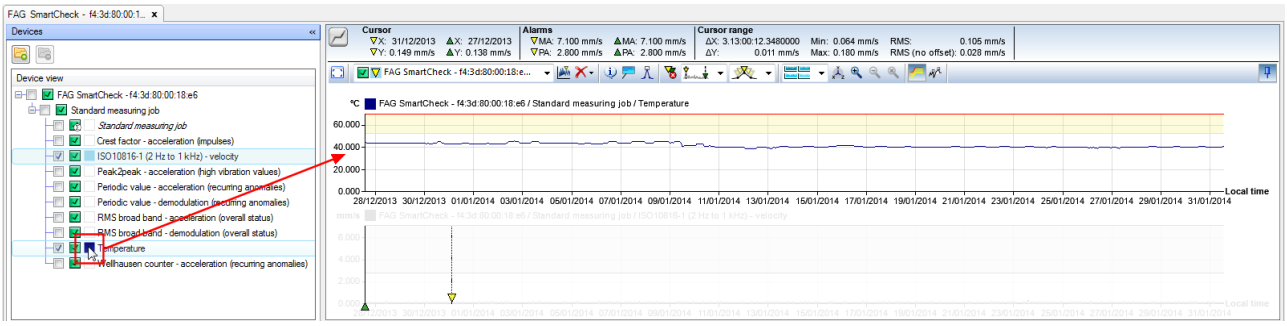
If you move the mouse over the symbol, you will see a preview of the trend and of the last available time signal and spectrum. You will also find brief information here, such as the time stamp or the minimum and maximum values for the Y-axis:



Colour identification

The colour box shows you the colour used to identify the trend in the Trend Viewer. This allows you to differentiate between several loaded trends.

If you move the mouse over the coloured box, the associated trend is highlighted in the Trend Viewer, while the other trends are greyed out:

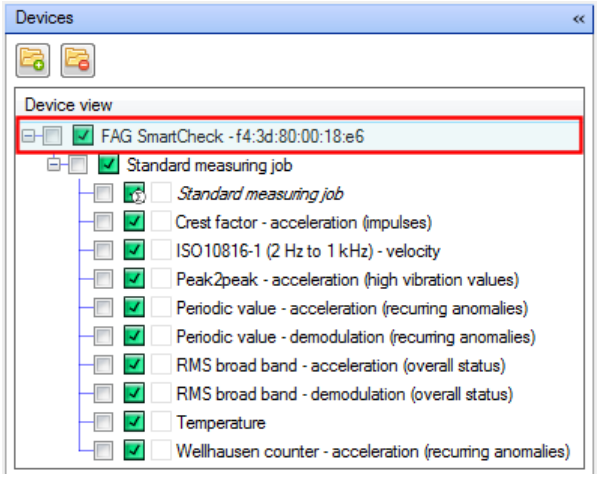


Button

Click this button to open further measurement data in *.SUVINFO format or *.DB format from another FAG SmartCheck device. This function automatically opens the directory from where the last device was opened; you can search for another directory if your data is not located here.

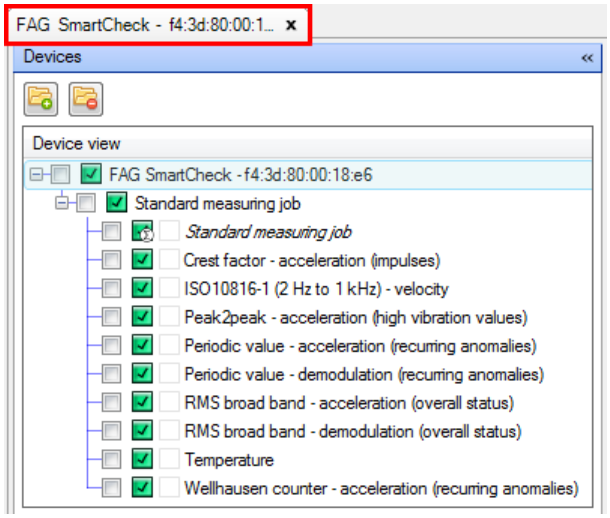
Button

Click this button to remove the FAG SmartCheck device selected in the device overview, together with all the measurement data, from the overview. This function is only available if you have selected the top level, i.e. the name and serial number of the FAG SmartCheck device:



Context menu in the title bar

When you open the FAG SmartUtility Viewer software, you will find a tab containing the device name in the title bar of the **device overview**:



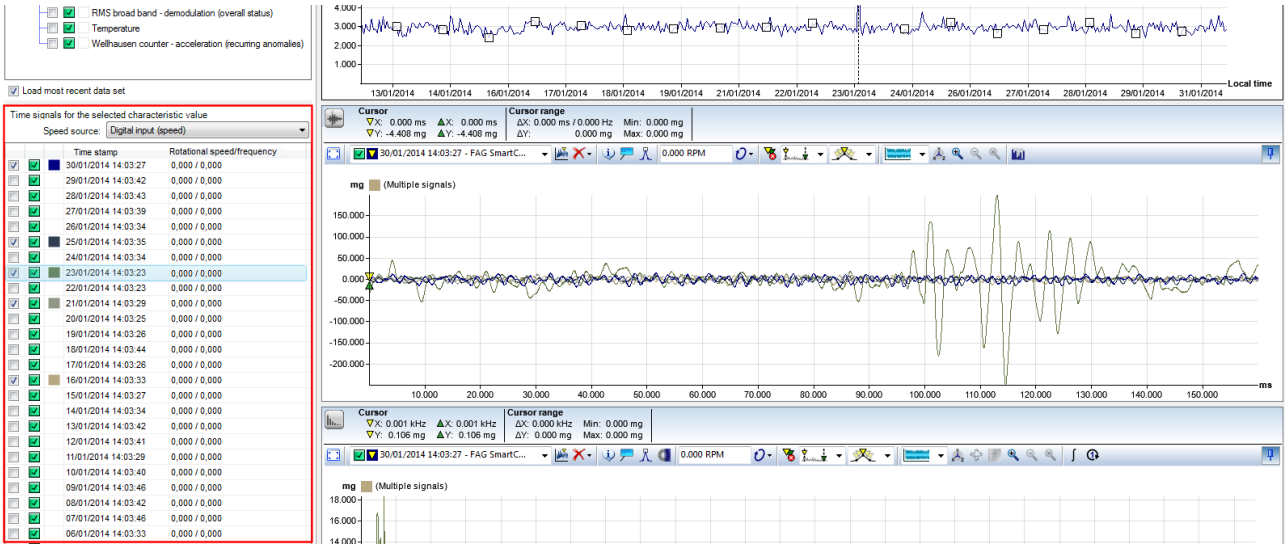
Right-click to open a context menu where you can add new tabs and manage existing tabs:

- **Add new tab**: Adds up to 32 new tabs.
- **Close tab**: Closes the active tab.
- **Close all tabs**: Closes all tabs.
- **Close all tabs except this one**: Closes all tabs except for the active tab.
- **Reset scaling in all tabs:** Resets the original size of the individual working areas in all the tabs. This means that any scaling that you carried out in the context menu on the separator lines 22 or manually is reversed.

The naming of a tab occurs automatically: As long as no FAG SmartCheck device with measurement data is loaded, the tab is given a number. Otherwise you will find the name of the FAG SmartCheck device here, whose measurement data is loaded. If you have opened measurement data from several FAG SmartCheck devices for analysis in one tab, you will see the note **(several devices)**.

5.3.3 Time signals for the selected characteristic value

The list of **Time signals for the selected characteristic value** can be found on the left under the **Device overview**. The content of the list depends on which characteristic value is selected in the device overview 23. The time signals you activate here with a checkmark are displayed in the Time Signal Viewer and in the Spectrum Viewer:



You have access to the following information and functions:

Speed source

Here, you can select the appropriate speed source from all speed sources defined in the measurement configuration. For gearboxes with a transmission, for example, you can select a second speed source here. The relevant speed and rotational frequency are displayed in the list of time signals.

Selecting a time signal

You select a time signal by activating a checkbox at the start of the line. The activated time signal is then displayed in the Time Signal Viewer and in the Spectrum Viewer.

Selecting multiple time signals

To select several time signals at the same time, perform the two steps described below:

Step 1: Click to select the required time signals

You have the following options for selecting the required time signals:

- Hold down CTRL and click on the required lines.
- To select a consecutive range of time signals, click on the first time signal in the desired range, hold the SHIFT key and then click on the last time signal in the desired range.

Selected time signals are highlighted in colour.

Step 2: Use the check box to select time signals

- Select one of the lines highlighted in colour with a checkmark. All the other lines highlighted in colour will also be selected.

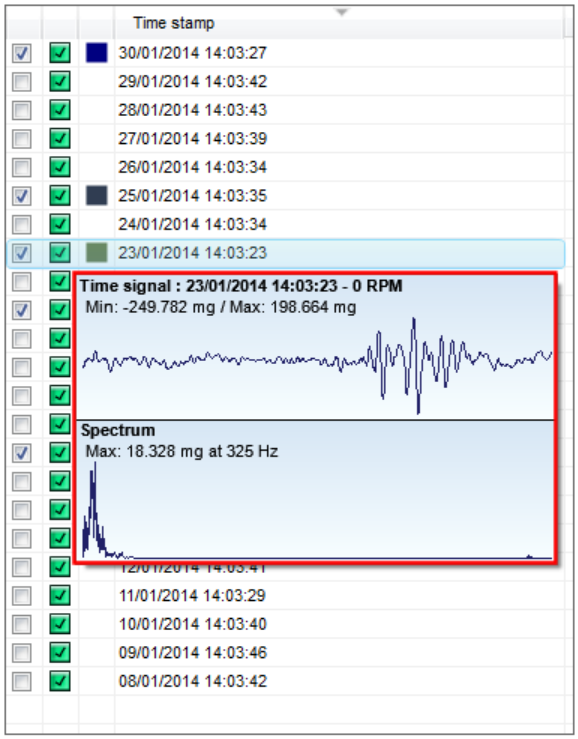
You can use the same method to remove a multiple selection.

Alarm status 

This symbol indicates the alarm status for the data:

-  : The measurement was taken but the device was still in the learning phase.
-  : No alarm
-  : Pre-alarm
-  : Main alarm

If you move the mouse over a symbol, you will see a preview of the time signal and the spectrum:



Colour identification

The coloured box indicates the colour used to display the data in the diagram. In this way you can differentiate between several loaded time signals or spectrums.

If you move the mouse over the coloured box, the associated signal is highlighted in the Time Signal and Spectrum Viewer, while the other signals are greyed out:

Time stamp

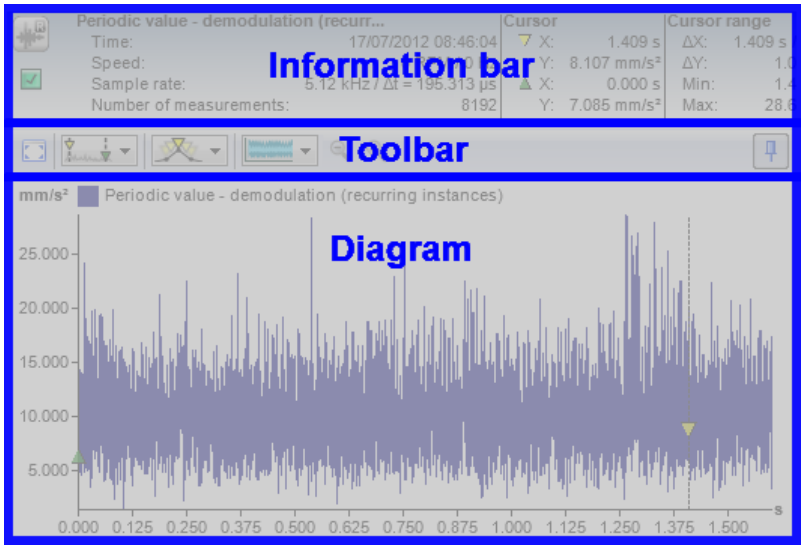
This is where you will find detailed information about when the time signal was saved.

In the list of **Time signals for the selected characteristic value** you can use the columns as a sort criterion**:**

Set any column as a sort criterion by clicking in the column title. Click a second time to change the sort order, i.e. from ascending to descending or vice-versa. The current sort order is displayed by the symbols ▲ for ascending and ▼ for descending.

5.3.4 Viewer and diagram

All of the FAG SmartUtility Viewer software viewers are identical in design and offer you the same basic functions and adjustment options, with a few exceptions. Each viewer consists of the information bar 29, toolbar 30 and diagram 32 areas:



You can find information on each area of the viewer in the following sections.

Information bar

The upper area of the Viewer contains basic information on the position of the base cursor 51 and measure cursor 51, and on the values of the cursor range.

You can access the following information and functions via the information bar:

This symbol indicates a time signal.

This symbol indicates a spectrum.

This symbol indicates a trend.

Cursor

The **Cursor** range contains the X and Y positions for both the base cursor and the measure cursor. You can also see the corresponding symbols for both cursors that are used to indicate them in the diagram.

The X and Y values are adjusted automatically when you change the position of the cursor in the diagram.

Alarms

Trend Viewer only

Here you will find the alarm limits for the main alarm and pre-alarm.

Cursor range

This shows you the difference between the base cursor and measure cursor, together with the smallest (**min.**) and largest (**max.**) values in the cursor range.

The difference values are adjusted automatically when you change the position of the cursor in the diagram.

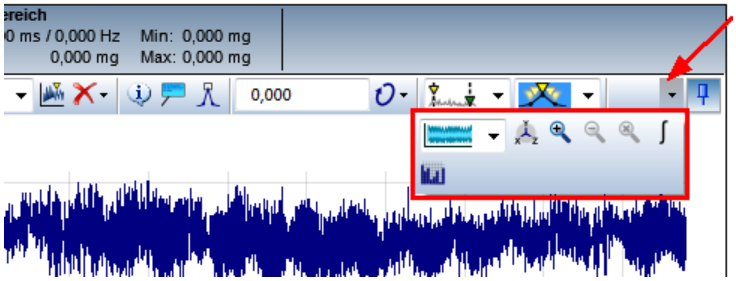
If you do not need the information bar or need more space to display the diagram, you can hide it using one of the following methods:

- Right-click to open the context menu for the Viewer 21 and select the **Information bar**. You can also use this function to display the information bar again.
- If you want to hide the information bars for all Viewers when you start the FAG SmartUtility Viewer software, you will need to adjust the Viewer program settings:
  1. Right-click to open the context menu for the Viewer 21 and select **Settings**.
  2. Click on **Viewer** 73 in the left-hand area.
  3. In the **Visible elements on startup** section, uncheck the **Information bar** option. The information bar will not appear in any of the viewers when the program is next started.

Toolbar

You can gain access to all the functions for data analysis and working on the diagram in the toolbar. The following overview shows you which functions you can call up via the selection lists and symbols. If a function is not available for all the viewers, you will find a note to this effect.

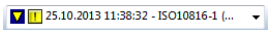
If not all of the symbols on the toolbar can be displayed, for example if the Viewer is too narrow, you will find the ▾ symbol on the right edge of the toolbar. Click on this to show the hidden toolbar functions:



Click this symbol to show the viewer in full screen mode.

To reintegrate the Viewer into the FAG SmartUtility Viewer software interface, click on the symbol.

Alternatively, you can access both commands by right-clicking in the context menu for the Viewer 21.



Use this list to determine the active signal, which is influenced by cursor functions, for instance. In each list entry you can find the following information:

- : The coloured box identifies the loaded trend or the loaded signal. You will find the same colour identification on the left in the **device overview** or in the list of **Time signals for the selected characteristic value**.
- : The yellow triangle identifies the currently active trend, the active time signal or the active spectrum.
- : This symbol indicates the alarm status of the data:
  - : The measurement was taken but the device was still in the learning phase.
  - : No alarm
  - : Pre-alarm
  - : Main alarm
- More details give you information about the time stamp, the name of the measurement configuration and the speed or frequency.

Click here to display just the active signal. Clicking again shows all the loaded signals.

Use this list to remove selected signals from the diagram. You have the following options:

- **Remove current signal:**

  The active signal is removed from the diagram. The other signals continue to be displayed. New active signal is the first signal in the list.
- **Remove all signals:**

  All the signals are removed from the diagram. The viewer is then empty.
- **Remove all other signals:**

  Only the active signal remains visible, all other signals are removed from the diagram.

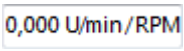
Click here to obtain precise information about the active signal. You can find details on this in the **Show signal properties** 41 section.

Click here to create a new comment about the active signal or to view or manage existing comments. You can find details on this in the **Comments** 41 section.

Click here to show the highest peaks for the active signal. You can find details on this in the **Show highest peaks** 43 section.

Spectrum Viewer only

Click here to show frequency bands or bearing frequencies. You can find details on this in the Show frequency bands 44 section.



Time Signal and Spectrum Viewer

You can directly input the speed for the active signal in this field. Other speed functions are available to you by clicking on  .

Time Signal and Spectrum Viewer

Click here for further speed/frequency settings. You can find details on this in the **Set speed/frequency** 51 section.

Click here to remove the base cursor, measure cursor and the symbols for cursor functions in the diagram. Clicking again shows them all again.

Click here to select cursor functions or to specify basic settings for all cursors and cursor functions. You have the following options:

- **Basis analysis** 53: Selects the **Basis analysis** cursor function.
- **Gear mesh** 54: Selects the **Gear mesh** cursor function. This function is only available in the Spectrum Viewer.
- **Harmonics** 55: Selects the **Harmonics** cursor function.
- **Sidebands** 56: Selects the **Sidebands** cursor function. This function is only available in the Spectrum Viewer.
- **Harmonics with sidebands** 56: Selects the **Harmonics with sidebands** cursor function. This function is only available in the Spectrum Viewer.
- **Speed** 57: Selects the **Speed** cursor function.
- **Cursor settings** 51**:** Select this option to perform general cursor settings and settings for the individual cursor functions.

  You can find details in the **Set cursor** 51 section and in the sections on the relevant cursor functions.

Click here to determine how the precise position of the base cursor will be determined when pulling in the diagram. You can find details on this in the **Position cursor** 58 section.

Click here to determine how the data should be displayed in the diagram. You can find details on this in the **Select diagram view** 59 section.

Click here to set the X, Y and, if applicable, the Z axes. You can find details on this in the **Edit axes settings** 62 section.

Spectrum Viewer only

Click here to set the camera settings for multi-dimensional display options. You can find details on this in the **Edit camera settings** 63 section.

Spectrum Viewer only

Click here to set the spectrogram display options. You can find details on this in the **Change**

spectrogram settings 63 section.

These buttons provide the following functions:

: Zooms into a step. One step corresponds to 10% of the axis limits.

: Undoes the last zoom step.

: Returns the diagram to normal view.

Trend Viewer only

Click here to show the alarm limits for pre-alarm (yellow line) and main alarm (red line). By clicking again, you remove them again.

If the alarm limits are not shown, the scale adapts to the highest peaks.

Trend Viewer only

Click here to show the time signal markers. By clicking again, you remove them again.

Time Signal and Spectrum Viewer

Click here to integrate signals. You can find details on this in the Integrate signals 64 section.

Time Signal Viewer only

Click here to calculate the spectrum/spectrogram from the signal. You can find details on this in the **Calculate spectrum** 66 section.

Spectrum Viewer only

Click here to calculate order or frequency spectrums. You can find details on this in the **Calculate order spectrum** 67 section.

Click on this button to hide the toolbar. As soon as you move the mouse over the toolbar it will become visible again, enabling you to use its functions.

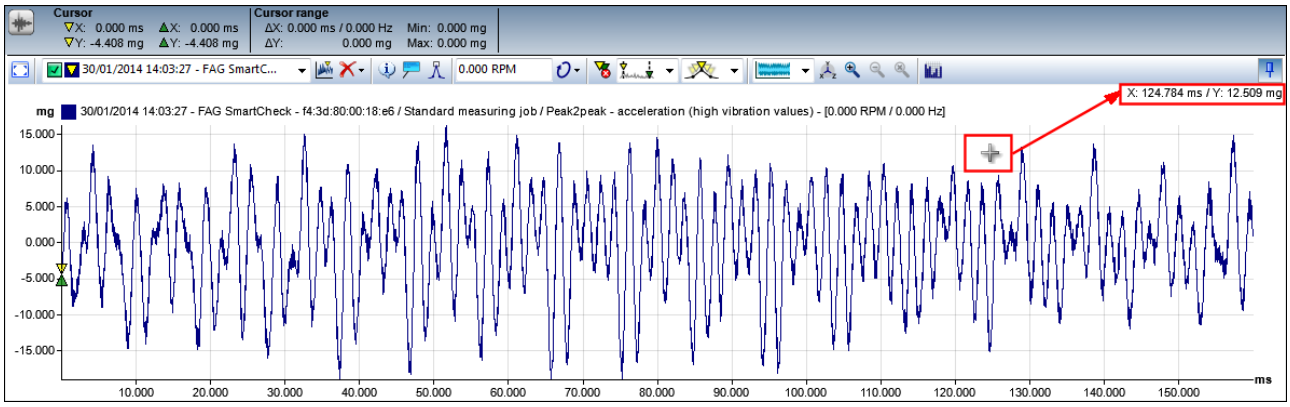
To make it permanently visible again, you will need to click in the toolbar.

If you want to hide the toolbars for all Viewers when you start the FAG SmartUtility Viewer software, you will need to adjust the Viewer program settings:

1. Right-click to open the context menu for the Viewer 21 and select **Settings**.
2. Click on **Viewer** 73 in the left-hand area.
3. In the **Visible elements on startup** section, uncheck the **Toolbar** option. The toolbar will not appear in any viewer when the program is next started.

Diagram

How the diagram is displayed initially depends on the type of viewer, i.e. whether the diagram is viewed in the Trend Viewer, the Time Signal Viewer or the Spectrum Viewer. Information on the mouse position is provided in the same manner in all diagrams: As soon as you move the mouse over a diagram, information on the mouse position is displayed in the top right corner:



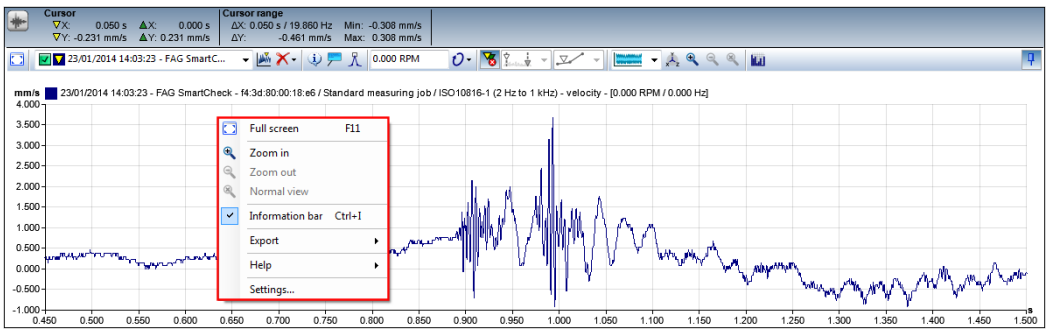
Generally speaking, you can change the display directly in the diagram as follows:

- **Base cursor positioning:** Click on the required location in the diagram. The base cursor automatically jumps to that position.
- **Position measure cursor:** Hold the SHIFT key and click at the required position in the diagram. The measure cursor automatically jumps to this position.
- **Move the base cursor** 51 **or measure cursor** 51: Move the mouse over the cursor guide until the mouse pointer turns into a double-headed arrow . Now press and hold the left mouse button and drag the cursor to the desired position.
- **Zoom functions**: The mouse and keyboard offer numerous options for zooming in and out of the diagram view. To zoom into the diagram, for instance, click inside the diagram, press and hold the left mouse button and drag the mouse over a section of the diagram. Use the back key to undo a zoom step. For detailed information on possible zoom functions, see **Appendix I: Zooming in the diagram** 78.

You can specify further settings for diagram display at the following locations in the FAG SmartUtility Viewer software:

- Use View options 59 to determine how the data should be displayed in the diagram, for instance as a **list** or as a **matrix**.
- Use your selection and marking in the device overview 22 and in the list of time signals 26 to determine which data is loaded and displayed in the diagram.
- Use the cursor options 51 to determine, for instance, which cursor and cursor functions are displayed in the diagram.
- Use the axes settings 62 to determine the unit and scaling of the diagram axes.
- Use the comments functions 41 to add and edit comments; you can also determine whether existing comments should be displayed in the diagram.
- You can adjust numerous general diagram display settings in the **Settings** 72 dialogue box; for instance, you can define the colours for the data displayed, symbols for cursors and cursor functions, and the unit profile to be used for axis scaling.

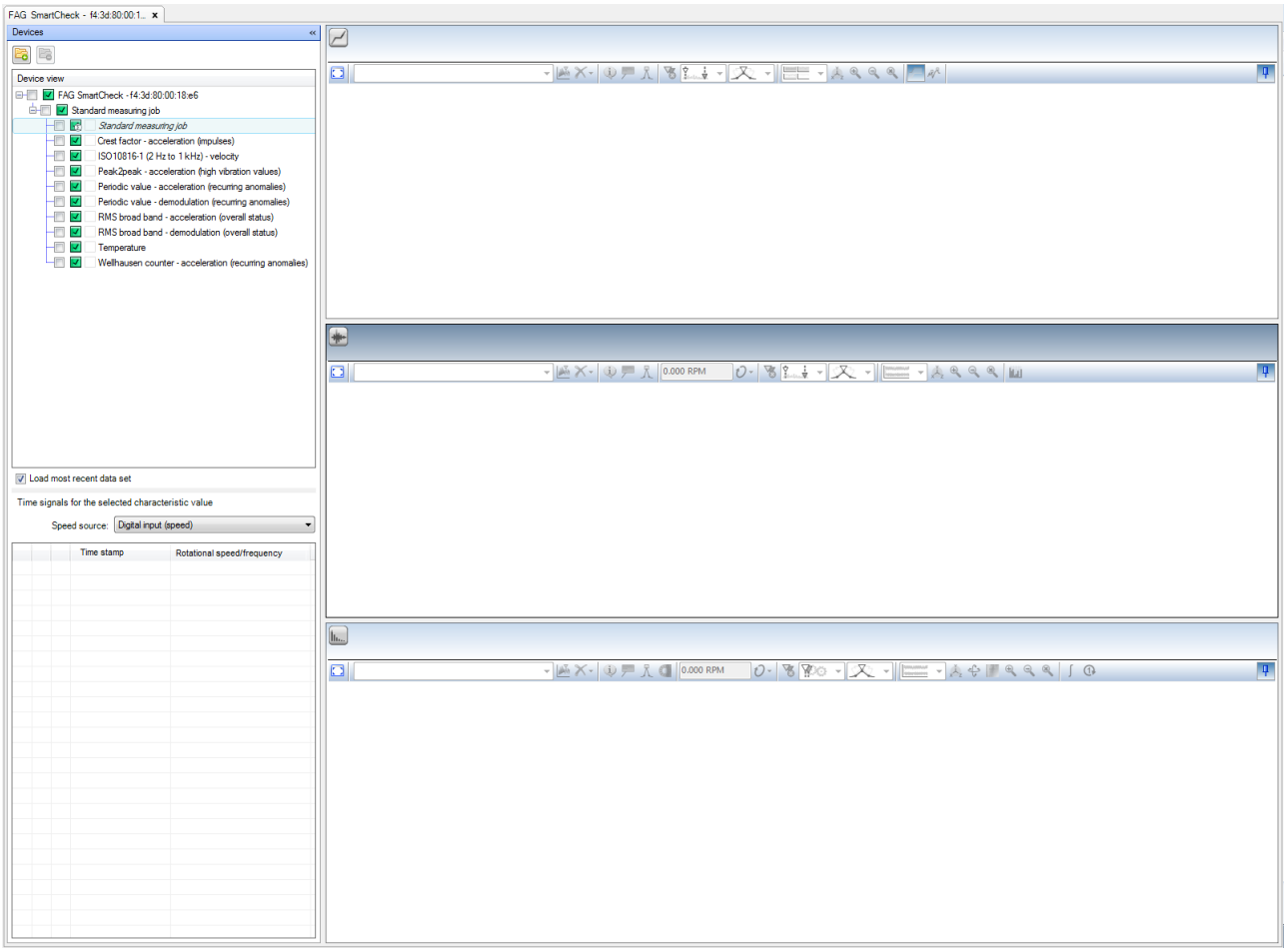
Right-clicking in any of the Viewers will open a context menu with access to important cross-program functions; the example shows the context menu in the Time Signal Viewer:



You will find a brief explanation of these functions and links to additional information in the **User interface overview** 21 section.

5.3.4.1 Opening and removing data

If you successfully execute the **Analyse data** wizard in the FAG SmartUtility software, the FAG SmartUtility Viewer software will open automatically. In the **Device overview** 22 at the top left, you will then see the characteristic values for the measurement data available to you for analysis. The list of **Time signals for the selected characteristic value** at the bottom left as well as the Trend Viewer, Time Signal Viewer and Spectrum Viewer are still empty:



In the following sections you will learn how to select the data for analysis and open it in the individual viewers, and how to remove the selected data again:

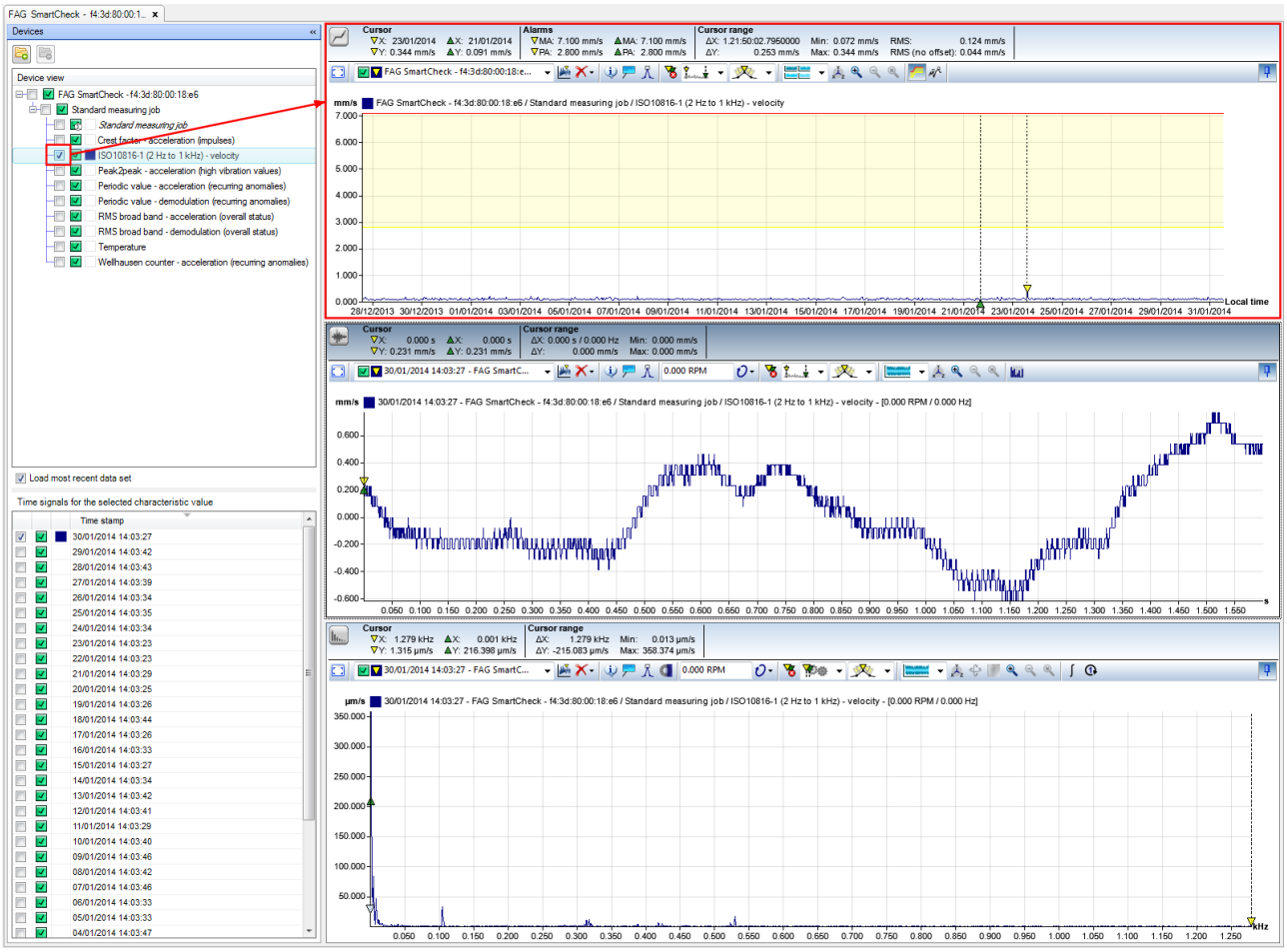
- Open one trend 34 (Trend Viewer)
- Open several trends 35 (Trend Viewer)
- Automatically open last available time signal 35 (Time Signal and Spectrum Viewer)
- Open one time signal 36 (Time Signal and Spectrum Viewer)
- Open several time signals at the same time 38 (Time Signal and Spectrum Viewer)
- Open time signals from the Trend Viewer 38 (Time Signal and Spectrum Viewer)
- Open spectrum from the Time Signal Viewer 40 (Spectrum Viewer)
- Remove data from the viewers 40

In the device overview and in the list of **Time signals for the selected characteristic value** you can display a preview of the associated signals. Move the mouse over the appropriate alarm symbol.

Open one trend (Trend Viewer)

1. Activate the characteristic value in the device overview, whose trend you want to load in the Trend Viewer.

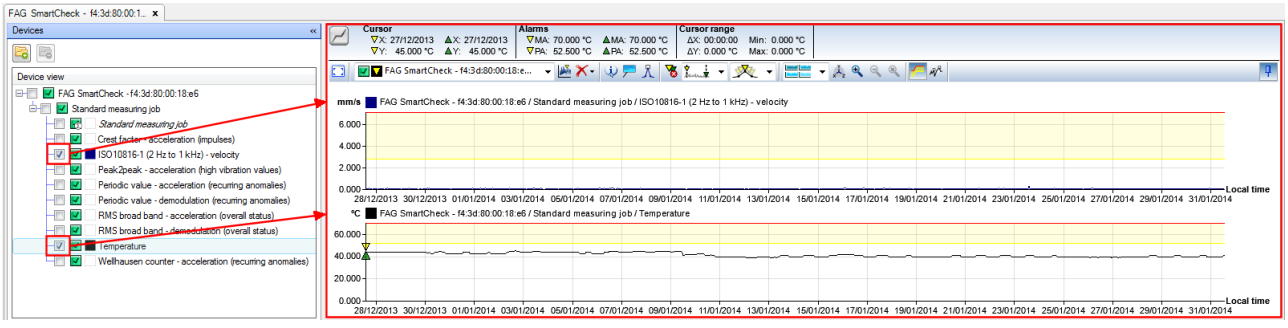
2. The trend is loaded in the Trend Viewer:



The list of **Time signals for the selected characteristic value** is also displayed. If the option **Load most recent data set** has been activated, the last available time signal will also be loaded in the Time Signal/Spectrum Viewer.

Open several trends (Trend Viewer)

You can compare the trends of several characteristic values with each other by going to the **device overview** and activating characteristic values 24 several times. The associated trends will then all be displayed in the Trend Viewer. The type of display depends on which diagram view you have selected 59:

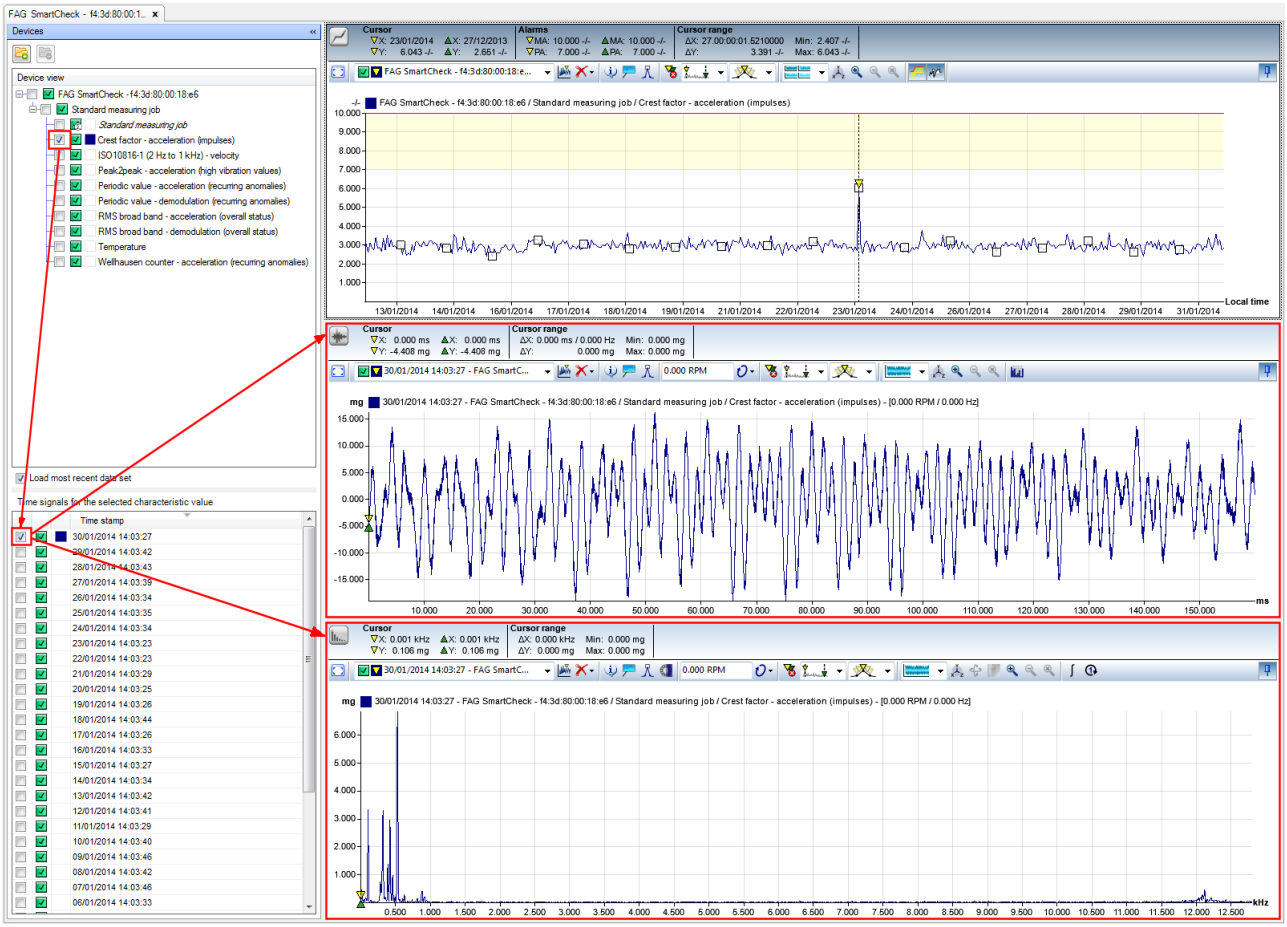


With each additional open trend, the Time Signal and Spectrum Viewer are also adjusted: If the **Load most recent data set** option has been activated, the last available time signal will be loaded in the Time Signal and Spectrum Viewer for each loaded trend.

Automatically open the most recent data set (Time Signal and Spectrum Viewer)

Activate the option **Load most recent data set**. Then activate the characteristic value in the device overview, for which you want to load the most recent data set in the Time Signal and Spectrum Viewer:

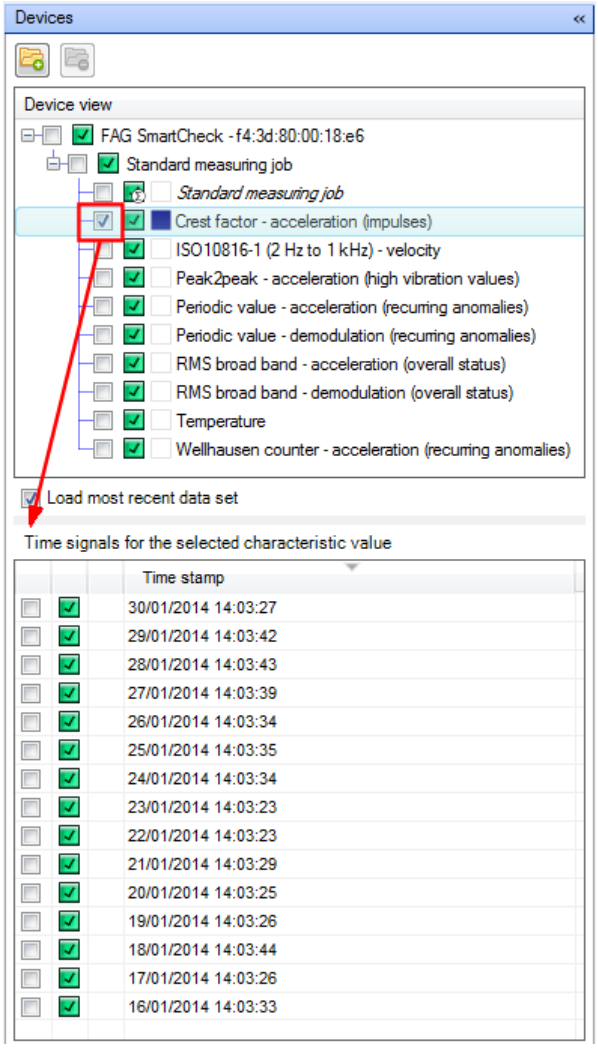
- The latest time signal is automatically activated in the list of **Time signals for the selected characteristic value**.
- The latest time signal is loaded in the Time Signal/Spectrum Viewer.



In addition, the associated trend is loaded in the Trend Viewer and the list of time signals for the characteristic value is displayed.

Open one time signal (Time Signal and Spectrum Viewer)

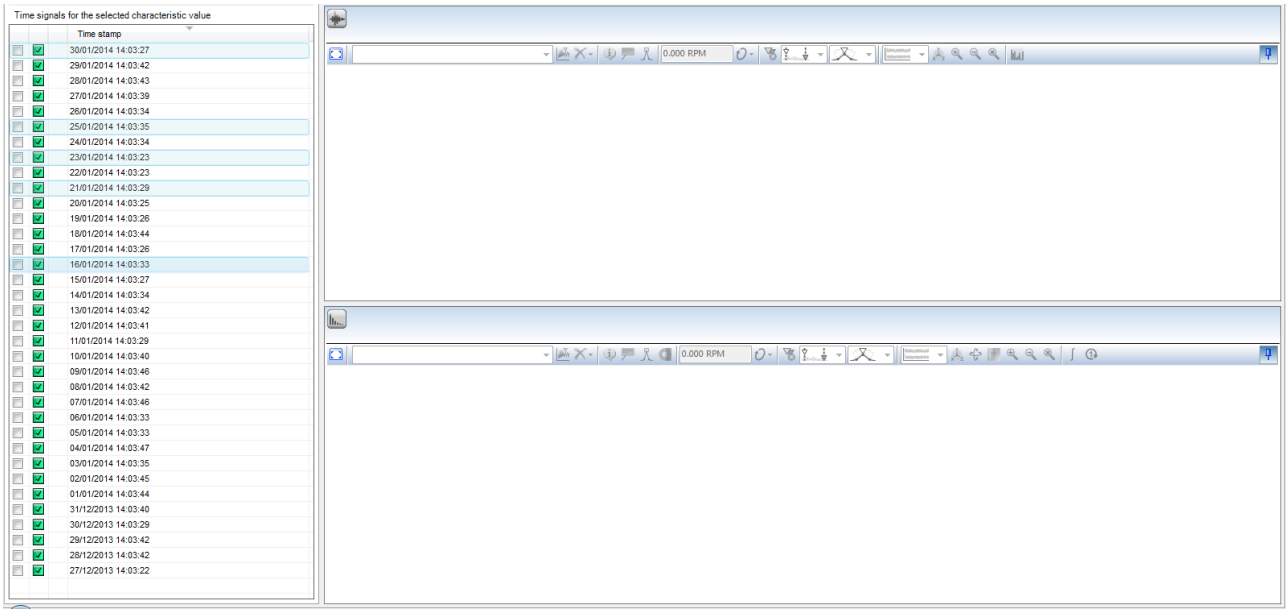
1. Select the characteristic value for which you want to load a time signal. The list of **Time signals for the selected characteristic value** fills up:



2. In this list, activate the time signal you want to load in the Time Signal and Spectrum Viewer. The signal is then displayed directly:

Open several time signals at the same time (Time Signal and Spectrum Viewer)

1. In the list of **Time signals for the selected characteristic value** select those time signals that you want to load into the Time Signal and Spectrum Viewer. Selected time signals are highlighted in colour:



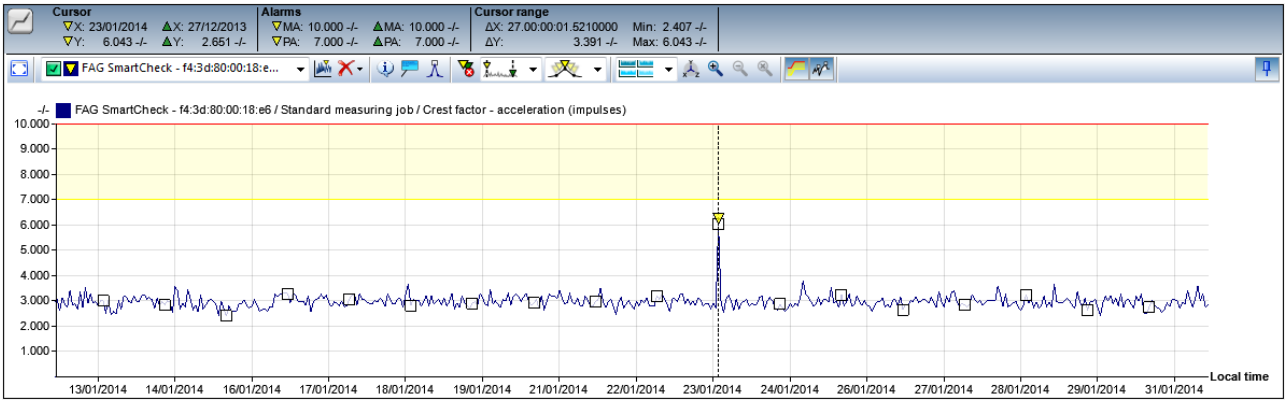
Here you have the following options:

- Shift + click: Selects all the time signals, which are located in the list between the first and second clicks.
- CTRL + click: Adds each clicked time signal to the previous selection.

2. Select one of the selected signals by setting a check box. By doing so, all the other selected signals are also marked automatically, hence receiving a checkmark and colour highlighting. All the marked signals are displayed in the Time Signal and Spectrum Viewer:

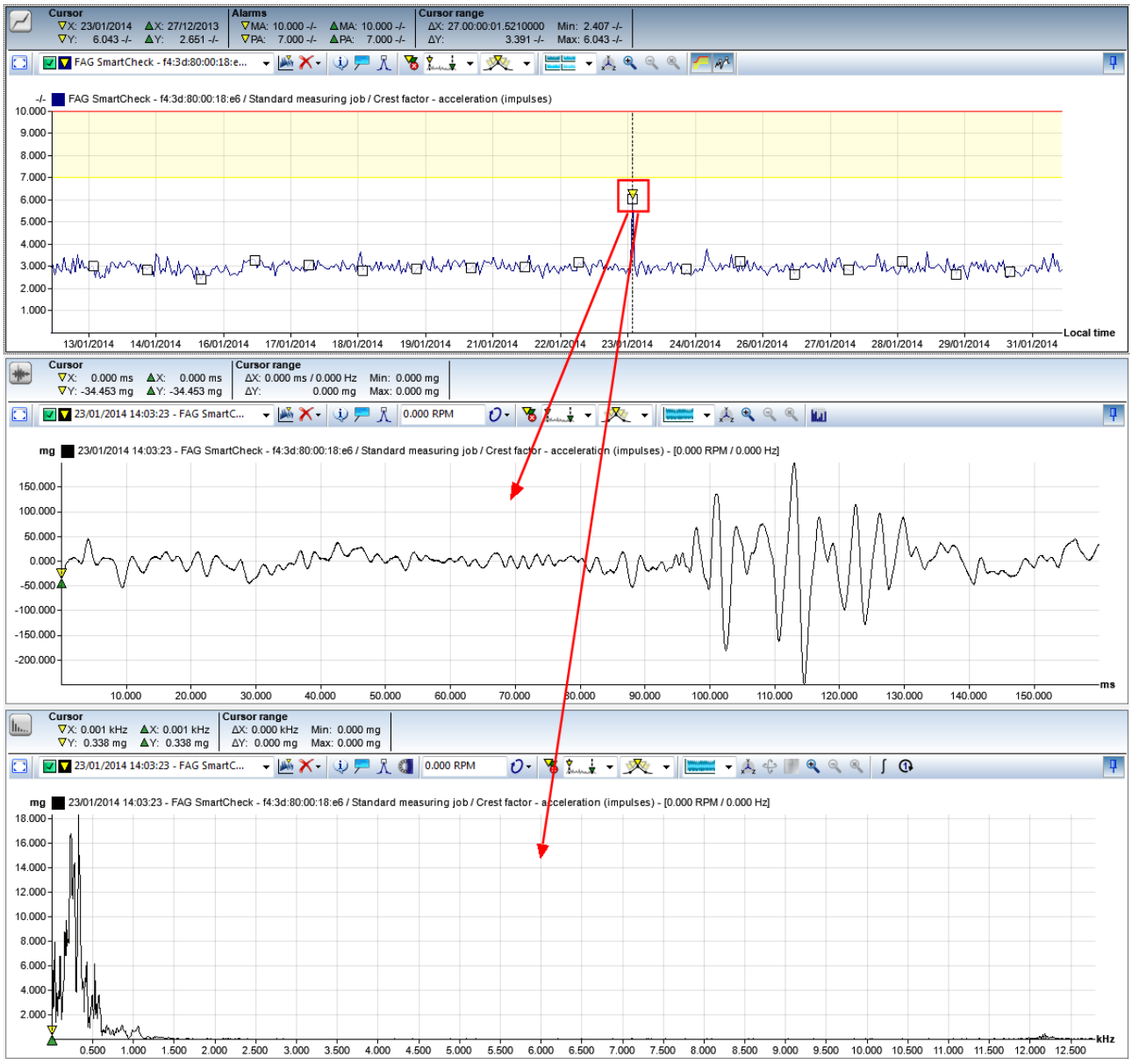
Open time signals from the Trend Viewer (Time Signal and Spectrum Viewer)

1. Show the time signal marker in the Trend Viewer. To do so, use the button in the toolbar:



2. Move the mouse over a time signal marker to see a preview of the diagram and to make it easier to select the required time signal:

3. Double click on the time signal marker to load the associated diagram in the Time Signal and Spectrum Viewer:



- When you double-click to open a time signal, it may take a while depending on the length of the time signal: the more measurement values a time signal contains, the longer it will take to calculate the spectrum.
- The default symbol for the time signal marker is a box; however you can define a different symbol under **Settings > Symbol** 75 in the context menu for the Viewer.

Open spectrum from the Time Signal Viewer (Spectrum Viewer)

You can create a spectrum in the Spectrum Viewer directly from within the Time Signal Viewer and specify your own settings, e.g. for windowing. To do so, use the function **Calculate spectrum** 66 in the toolbar for the Time Signal Viewer.

Remove data from the viewers

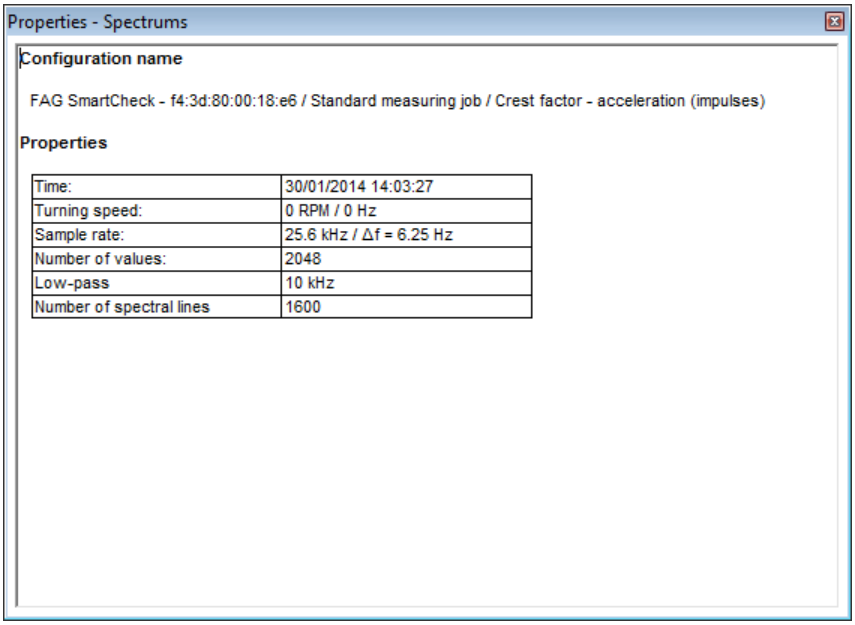
You have several options to remove data from the viewers:

- You can remove the marking for a characteristic value in the device overview. As a result, the associated trend will be removed from the Trend Viewer.

- You can remove the marking for a time signal in the list of time signals for the selected characteristic value. As a result, the associated signals will be removed from the Time Signal Viewer and the Spectrum Viewer.
- In the toolbar for each viewer, using the button you will find functions which you can use to remove certain signals from the diagram. More details on this can be found in the **Viewer and diagram** 31 section.

5.3.4.2 Show signal properties

If you click on in the toolbar, a dialogue box opens with an overview of the most important properties of the active signal:

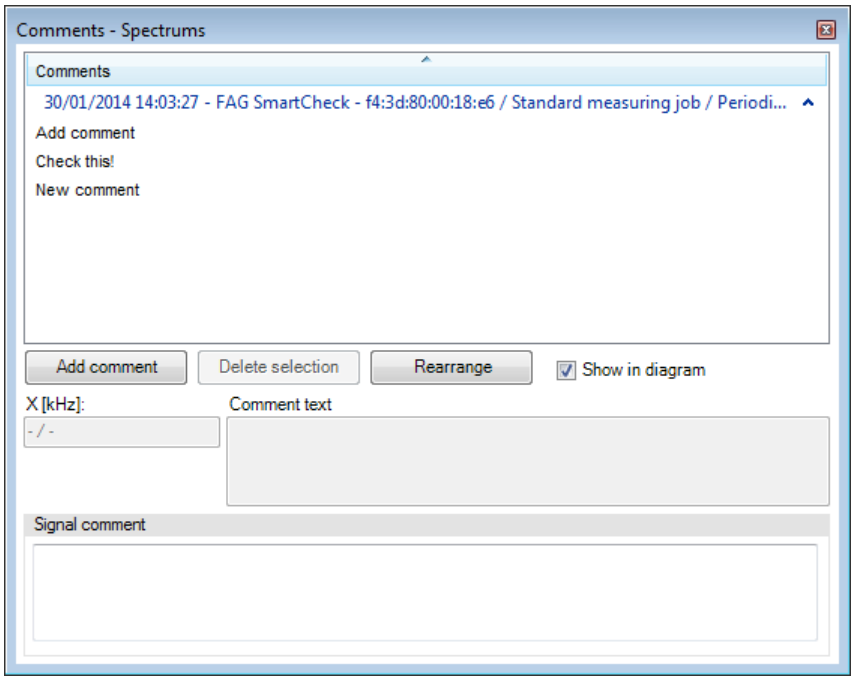


Here you will find, for instance, the name of the configuration and measurement, the time stamp and the sample rate. For Trend, you will also find details of the start and end timepoint for the data set.

You can select these details, use **CTRL+C** to copy them to the clipboard and then use **CTRL+V** to paste them into a Word document, for instance.

5.3.4.3 Adding/editing comments

If you click on in the toolbar, a dialogue box opens with an overview of all the comments that exist for the currently loaded signals in this Viewer. You can also add comments to the active signal and determine whether the comments should be displayed in the diagram:



Here you have the following options:

Comments

This section contains a list of all the comments relating to the currently loaded data in the Viewer. Here you have the following options:

- The data, for which comments already exist, can be found in this list in each case as a blue heading. Below this heading, i.e. below the data name, you will find the appropriate comments.
- You can click on comments to select them, for instance if you want to delete them. Selected comments are highlighted in colour.
- Every new comment has its own line.
- You can select longer comments to read by clicking on the comment. The entire comment text then appears in the field labelled **Comment text** and can be edited there.
- You can remove the comments from the data by clicking on the arrow to the right ^, the comments reappear by clicking again.

Add comment

Click this button to create a new comment for the active data. When the option **Show in diagram** is activated, the comment window in the diagram is automatically attached to the X position of the base cursor. You can then enter your text into the field labelled **Comment text** and, where indicated, change the position of the comment 43 in the diagram.

Delete selection

Click this button to delete the selected comments.

Click on a comment to select it. To select multiple comments, press and hold **CTRL** and click on the comments you want to select.

Rearrange

Click this button to automatically arrange comments so that all of them are visible in the diagram. This can be useful if, for example,

- numerous comments are stacked on top of each other and some of them are not visible;
- if you switch the viewer from full screen mode back to integrated mode; this can cause comments to disappear from the significantly smaller area.

Show in diagram

Activate this option to display all of the comments in the diagram.

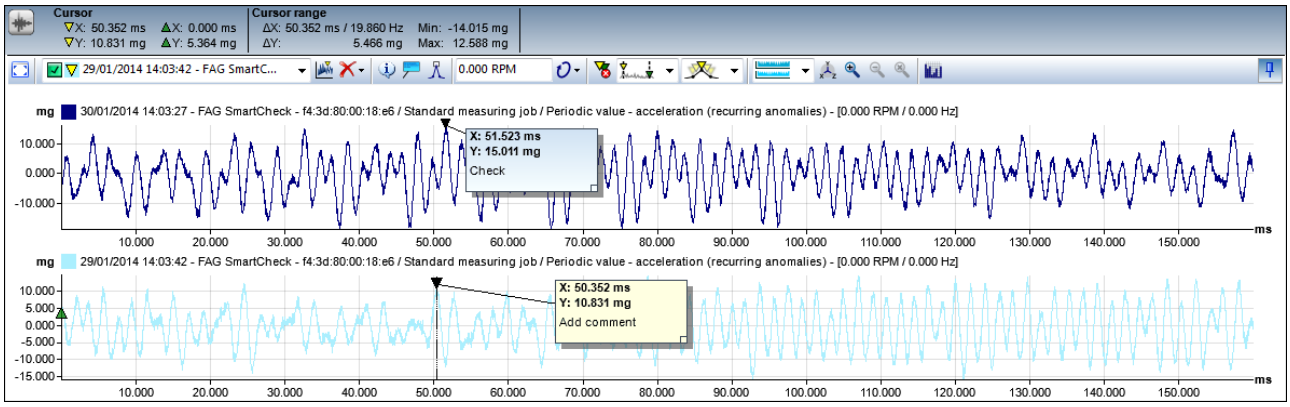
X value

You can edit this field as soon as you have selected a comment or have clicked on the **New comment** button.

You can directly input the X position for the new/selected comment here. The peak formed by the data at the X position automatically becomes the Y position of the comment.

Comment text	You can edit this field as soon as you have selected a comment or have clicked on the **New comment** button. Enter the new comment here or edit the text of an existing comment.
Signal comment	The signal comment refers to the entire signal/spectrum or trend, and is used to provide an overview of how far your analysis has progressed or whether it is complete, for instance. This comment is not shown in the diagram; you can, however, export it by copying or saving the diagram in RTF format 68.

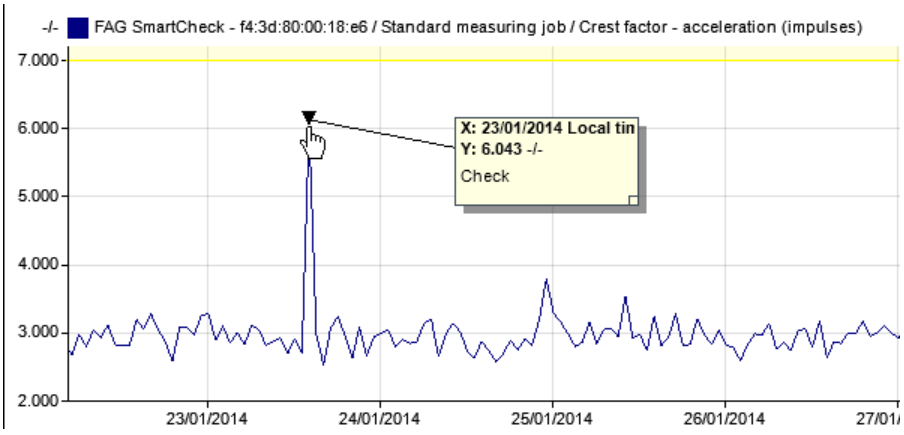
Once you have entered and positioned your comments, the Viewer will look something like the following illustration; you will find the X and Y coordinates where the comment is located adjacent to the text you entered into the **Comment text** box in the comment box:



Positioning comments

You have various options to influence the position of a comment:

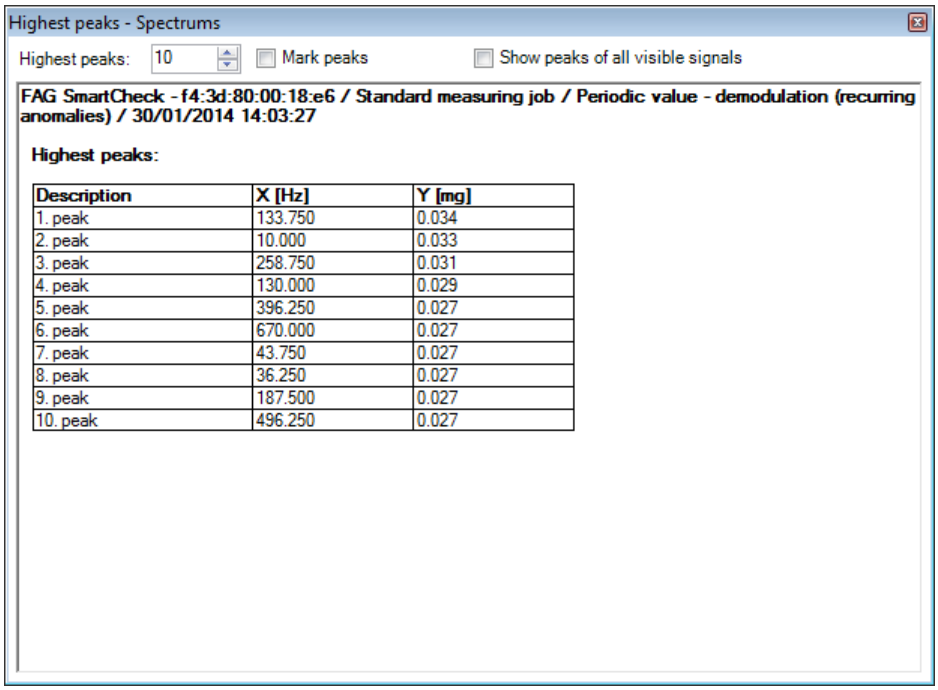
- Before creating a comment, position the base cursor at the X coordinate where you want to pin the comment.
- Enter the X coordinates where you want to pin the comment into the field **X[s]**.
- Move the mouse over the black fixation triangle of an existing comment; when the pointer changes into a hand symbol, press and hold the left mouse button and drag the fixation point to the desired position:



Arranging comments automatically via the **Rearrange** function only changes the position of the comment box; it does not change the fixation point.

5.3.4.4 Show highest peaks

If you click on  in the toolbar, a dialogue box opens. Here you will find a table with the highest peaks for the active signal:



Here you have the following options:

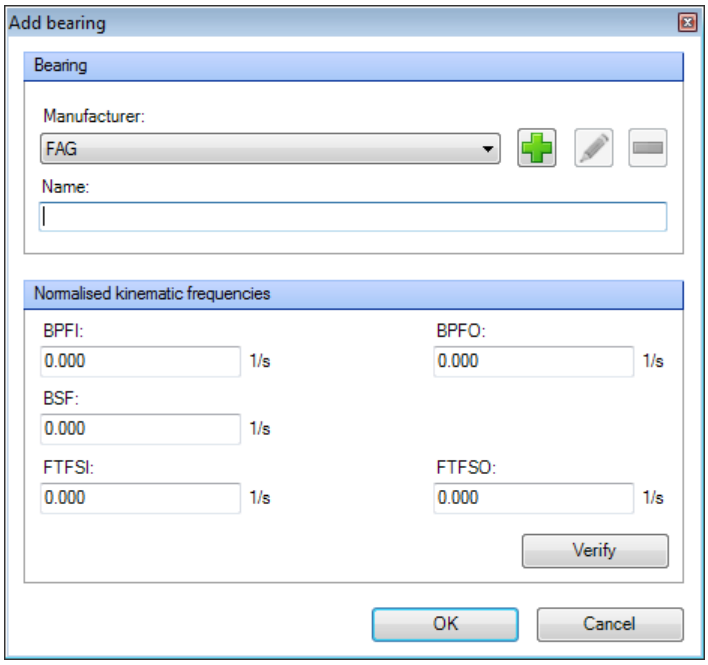
- **Highest peaks:** Determine how many peaks in total should be calculated.
- **Mark peaks:** Activate this option to show the peaks in the diagram. If you do not activate this checkbox, the peaks will only be listed in tabular form.
- **Determine peaks from all visible signals:** Activate this option to determine the peaks not only for the active signal/spectrum, but for all of the signals/spectrums that are marked in the **Time signals for the selected characteristic value** 26 list. The peaks are then listed in tables; the name of the corresponding signal or spectrum is shown in the header of each table. If you have activated the **Mark peaks** option, you will also see all of the peaks displayed in the diagram.

You can select the table(s), use **CTRL+C** to copy them to the clipboard and then use **CTRL+V** to paste them into a Word document, for instance.

- You can define the symbol by which the peaks are denoted in the diagram under Symbol settings 75. To do so, open the context menu for the Viewer 21 and select the **Settings** command. In the **Settings** dialogue box, under **Cursor** you can use the **Other** option to change the shape and colour. Whatever you create here will appear as the symbol for peaks in the diagram.

5.3.4.5 Show frequency bands (Spectrum only)

The function **Show frequency bands**  is only available in the toolbar for the spectrum viewer. It opens a dialogue box, in which you will find information about frequency bands and bearings of the loaded spectrums:



Here you have the following options:

Tabs

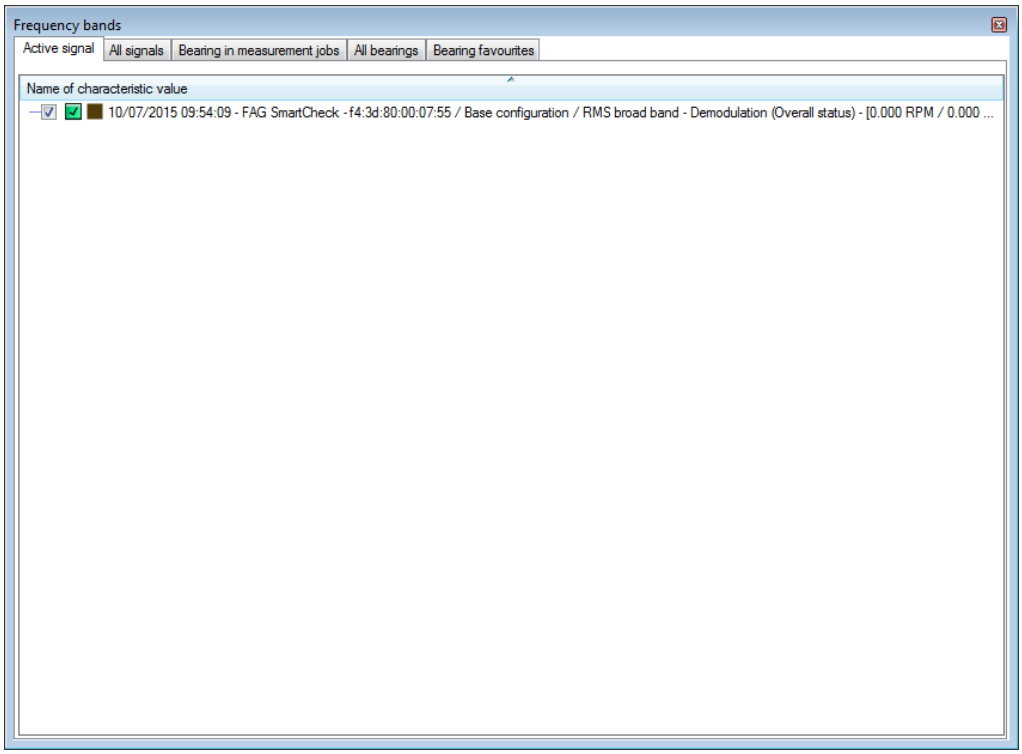
Use these tabs to select which frequency bands you want to have in the selection.

For details on the different display options, see **Active signal** 45, **All signals** 45, **Bearings in measurement jobs** 46, **All bearings** 47 and **Favourite bearings** 48.

For details on the functions for each display option, see **Display functions** 50.

Active signal (tab)

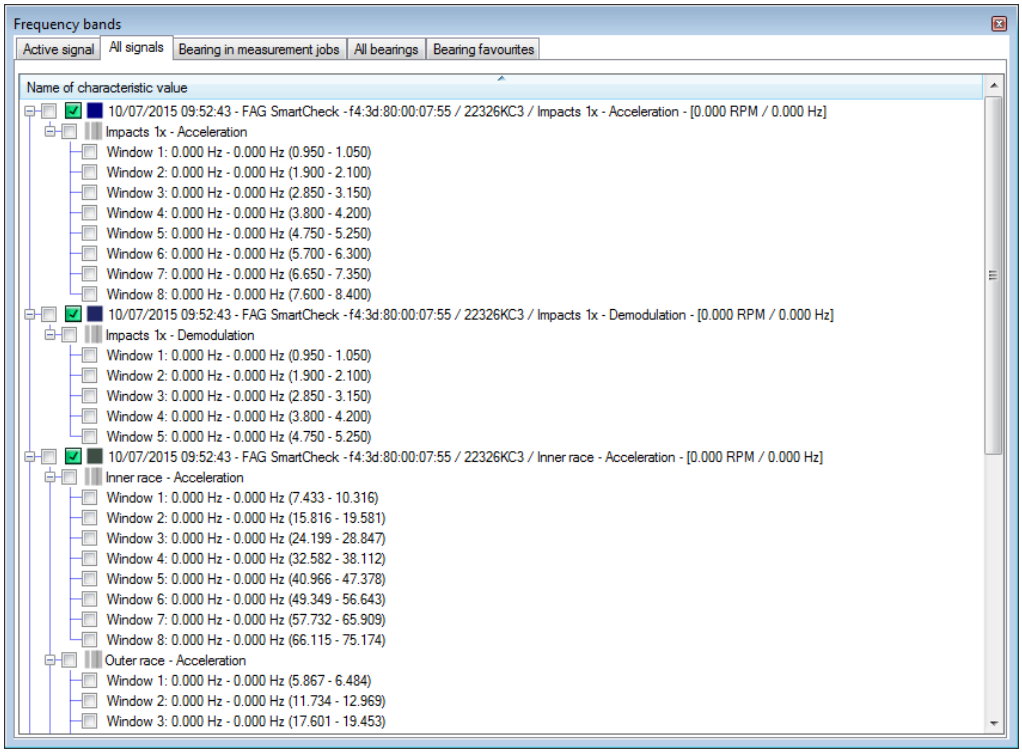
With this view option you can select from the frequency bands for the active spectrum. On the upper level you will find information about the characteristic value, in the level below, the available frequency bands are listed. If the characteristic value was configured with a bearing, you will also find the appropriate information here:



All signals (tab)

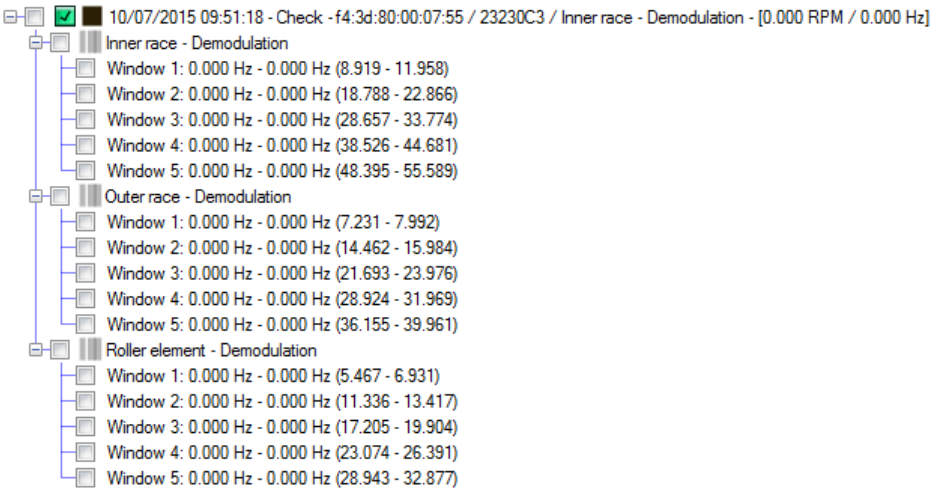
If you have loaded several spectrums in the Spectrum Viewer, you can use this option to select from the frequency bands for all the loaded spectrums. As in the **Active signal** view, you will

see the characteristic values on the upper level, and on the level below you will find the appropriate available frequency bands. You will also find bearing information here, if one of the characteristic values was configured with a bearing:



In the **Active signal** and **All signals** tabs, all characteristic values calculated from the selected signal are displayed. This also applies if you have opened the signal for a particular characteristic value.

For example, if you open the trend for one of the envelope characteristic values (e.g. inner ring) for a bearing, then all 3 envelope characteristic values are displayed here, i.e. roller element, inner ring, and outer ring:



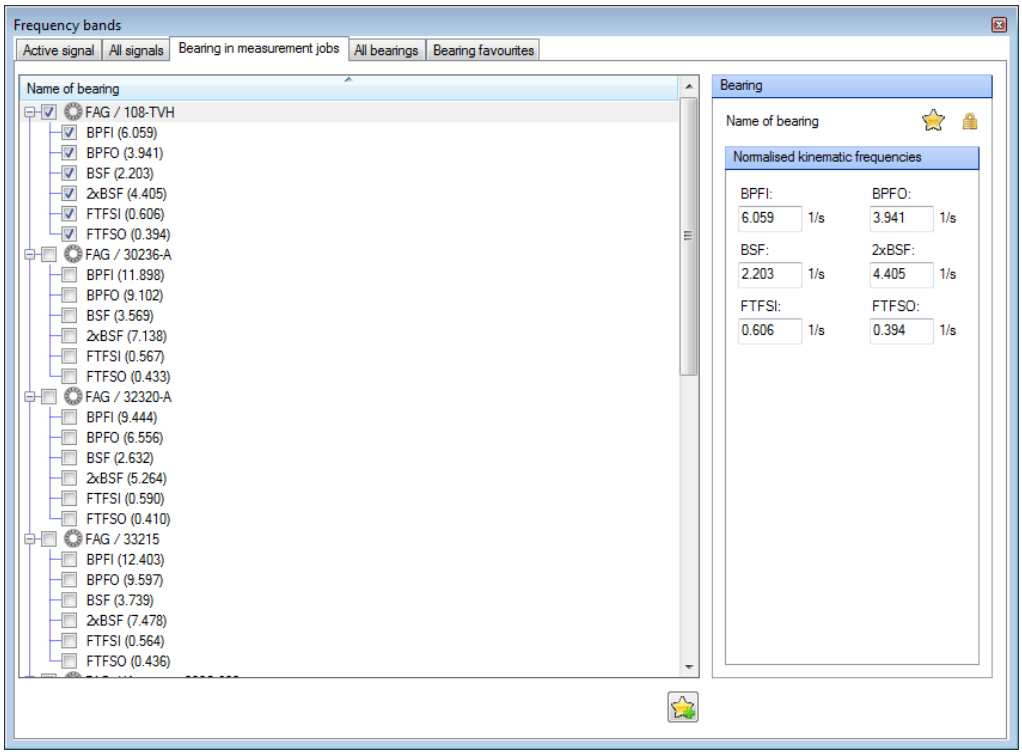
Bearings in measurement jobs (tab)

This view option is independent from the currently active and loaded spectrums. It is always available if one of the loaded FAG SmartCheck devices was configured with a bearing from the bearing database.

If the speed is known, you have the following additional option: The standardised cycling frequencies of the bearing are multiplied by the speed frequency. This results in cycling frequencies in Hertz, which are displayed in the spectrum.

If you select this option and bearings from the database have been configured, you will see the

following list:



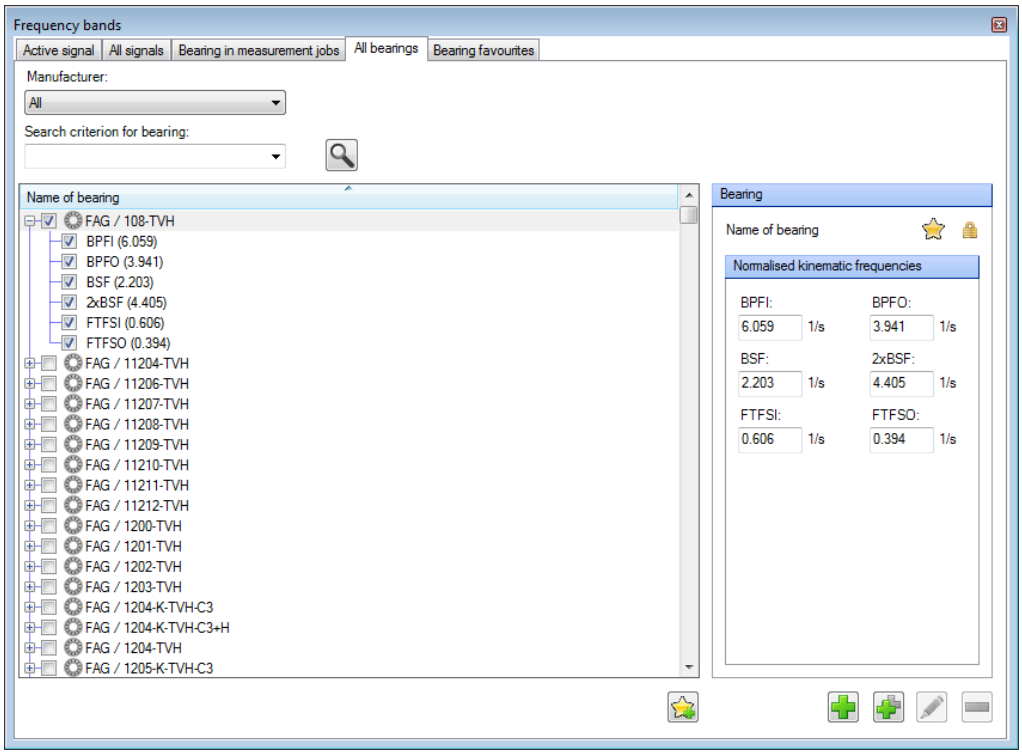
You have the following options:

- In the line with the bearing symbol , you will find the name of the relevant bearing.
- Below the line with the bearing symbol , you will find the bearing frequencies, which you can also display in the diagram. You will find the following bearing frequencies here, in each case with the calculated frequency in Hertz and the standardised frequency (value in brackets):
  - **BPFO:** the abbreviation stands for **Ball Pass Frequency Outer race**, in other words the cycling frequency of the outer race.
  - **BPFI:** the abbreviation stands for **Ball Pass Frequency Inner race**, in other words the cycling frequency of the inner race.
  - **BSF:** the abbreviation stands for **Ball Spin Frequency**, in other words the rotation frequency of the roller element.
  - **FTF:** the abbreviation stands for **Fundamental Train Frequency**.
  - **FTFSO:** the abbreviation stands for **Fundamental Train Frequency Standing Outer race**, in other words the fundamental train frequency where the outer race is stationary.
  - **FTFSI:** the abbreviation stands for **Fundamental Train Frequency Standing Inner race**, in other words the fundamental train frequency where the inner race is stationary.
- To the right of the list of bearings, an overview of the currently selected bearings is displayed. The overview contains all details of the **normalised kinematic frequencies**. The following symbols are also displayed:
  - : This symbol indicates that the selected bearing belongs to your favourite bearings.
  - : This symbol indicates that the selected bearing is write-protected and cannot be edited. This applies for all bearings contained in the bearing database on delivery.
  - : This symbol indicates that the selected bearing can be edited. This applies for all bearing copies and for bearings that you have created yourself.
- : Click on this button below the list of bearings to add the currently selected bearing to the **Favourite bearings**.

All bearings (Tab)

In this view option, you will find a list of all bearings in the database. You can display the frequencies for all bearings in the database. For example, if you have a measurement job for a specific bearing, but a different bearing is actually installed, you can search for this bearing

here and select it.

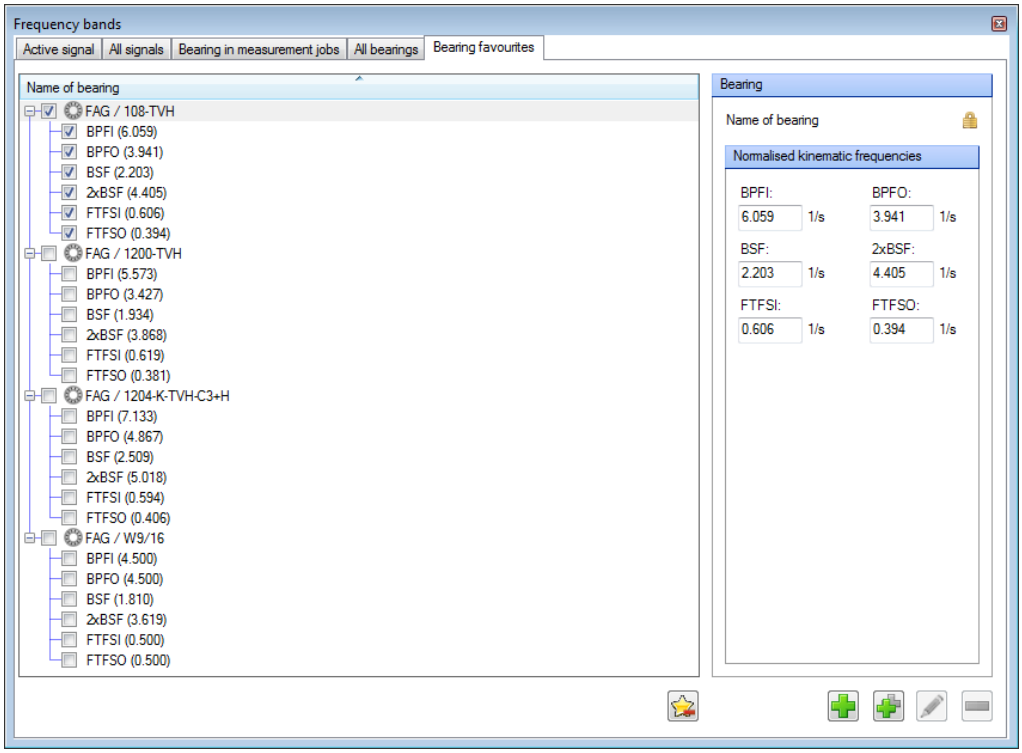


You have the following options:

- **Manufacturer**: Select the manufacturer of the required bearing to filter the list of all bearings by manufacturer.
- **Search criteria for bearings**: You can search for bearings by name in the database. To do so, enter the name in this field. Here you have the following options:
  - Search criteria that you have previously used are available in the selection box.
  - The placeholder * stands for any character string.
  - The placeholder ? stands for any single character.
  - Upper/lower case must be observed.
-  : Click on this button or use the input key to filter by **Manufacturer** and **Search criteria for warehouse** .
- As with the **Bearings in measurement jobs** view, the list of bearings contains the names of the relevant bearings on the top level, and the level below shows the bearing frequencies that you can display in the diagram. Select the frequencies that you want to display with a check box.
-  : Click on this button below the list of bearings to add the currently selected bearing to the **favourite bearings**.
- As with the view **Bearings in measurement jobs** , to the right of the list, an overview of currently selected bearings 44 is displayed.
- Use the buttons below the bearing overview for your bearing management 49.

Bearing favourites (Tab)

In this view option, you can see all bearings that you have marked as favourites using the  button. This list provides you with quick access to frequently used bearings.



You have the following options:

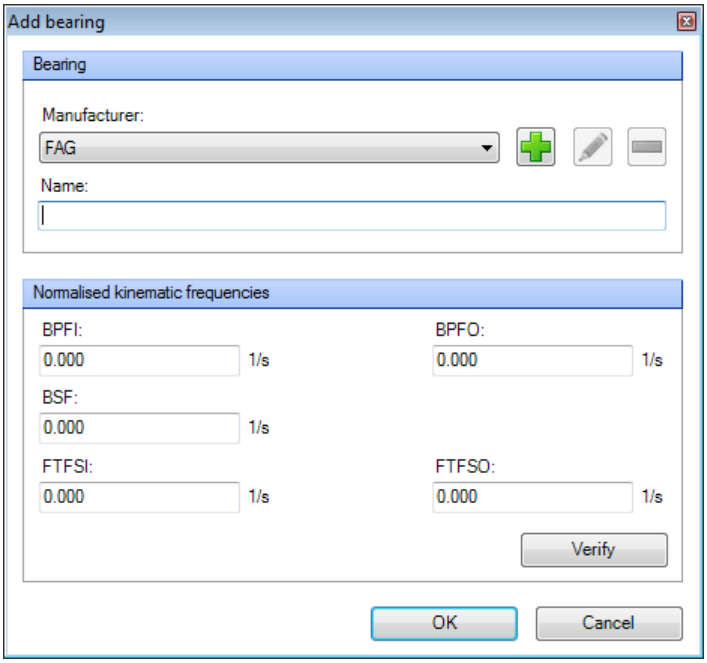
- **Display bearing frequencies**: As with the **Bearings in measurement jobs** view, this list contains the names of the relevant bearings on the top level, and the level below shows the bearing frequencies that you can display in the diagram. Select the frequencies that you want to display with a check box.
-  : Click on this button below the list of bearings to delete the currently selected bearing from the list of favourites.
- As with the view **Bearings in measurement jobs** , to the right of the list, an overview of currently selected bearings 47 is displayed.
- Use the buttons below the bearing overview for your bearing management 49.

Bearing management

On the **All bearings** and **Favourite bearings** tabs, the functions that you can use to manage your bearings are shown below the bearing overview:

Add bearing

Click on  to add a new bearing for analysis in the Viewer software. The **Add bearing** dialogue is displayed:



In this dialog, enter the **Manufacturer** and **Name** of the bearing. You can change the list of manufacturers with the following buttons:

- : Click on this button to add a new manufacturer name to the list.
- : Click on this button to edit the currently selected manufacturer name. You can only edit manufacturer names that you have added yourself, and for which you have not yet created a bearing.
- : Click on this button to delete the selected manufacturer name. You can only delete manufacturer names that you have added yourself and for which you have not yet created any bearings.

The specifications for the **normalised kinematic frequencies** are important for the correct calculation of bearing damage frequencies and thus for the reliable monitoring of this component. You can find the corresponding information on **BPFI**, **BPFO**, **BSF** and **FTF** in the technical data of the bearing. Click on the **Check** button to check your specifications against the minimum requirements.

Copy bearing

Click on to create a copy of the selected bearing. You can edit and delete copies of bearings.

Edit bearing

Click on to edit the currently selected bearing. The **Edit bearing** dialogue is displayed. The same options are available here as in the **Add bearing** dialogue.

Delete bearing

Click on to delete the selected bearing. You can only delete bearings that you have added or created as a copy yourself.

Display functions

Irrespective of the current display option, you have the following editing options:

Sort list:

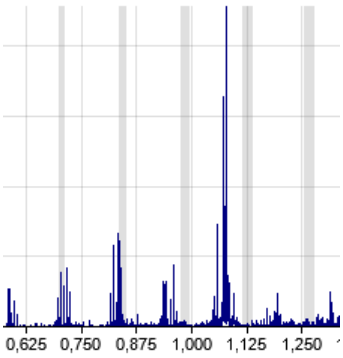
Click on the column header of the list, for example, **Name of bearing** to reverse the sort order.

Show preview:

If you move the mouse over an entry in the list, the associated frequency band or the bearing frequency is displayed as a preview in the diagram.

Show frequency band/bearing frequency:

- Activate a characteristic value or a bearing, in order to show all the associated frequency bands or bearing frequencies.
- You can also add or remove a checkmark against individual frequency bands or bearing frequencies to show them or remove them again.
- Frequency bands or bearing frequencies are highlighted in colour in the diagram:



- A maximum of 10 bearing frequencies can be displayed.
- If you remove a device with bearing configuration in the Viewer software from the device overview, the associated bearing information remains in the software. It is therefore possible that bearing frequencies are displayed for a device that is no longer loaded.
- Expert knowledge is required to use the **Bearings** view correctly.

5.3.4.6 Set speed/frequency

If you click in the toolbar, a menu opens with the following options:

- **Apply current value to selected signals:** The speed value of the active signal (identified with yellow triangle) is applied to all the signals currently loaded in the Viewer.
- **Reset value:** This command resets the speed/frequency for the active signal to the original value.
- **Reset value for all signals:** This command resets the speed/frequency for all signals loaded in the Viewer to the original value.
- **RPM:** Activate this option if you want to specify the speed in **rpm**.
- **Hz:** Activate this option if you want to specify the rotational frequency in **Hz**.

5.3.4.7 Cursor setting

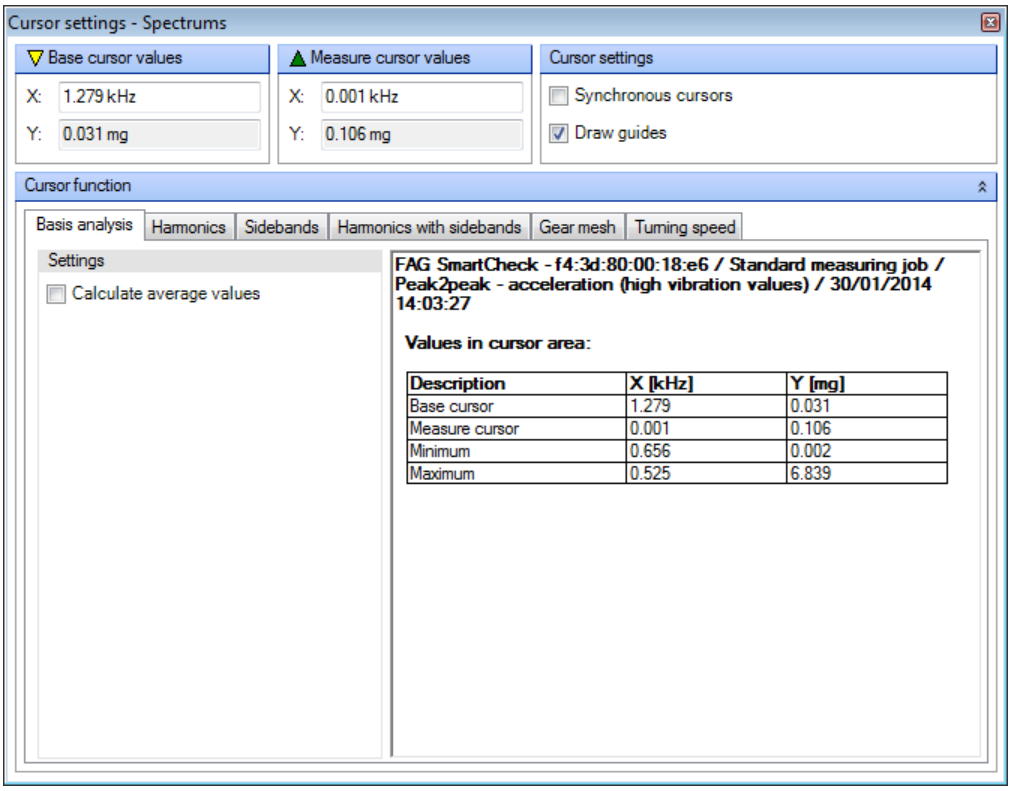
If you click on in the toolbar, a menu opens, in which you can selected the required cursor function 52. In addition, you can open the **Cursor settings** dialogue box from this menu. The **Cursor settings** dialogue box has many options designed to help you use the base and measure cursors and the associated cursor functions 52.

Use the **base cursor** to define the base value for your analysis. In the Spectrum Viewer, for instance, this would be the base frequency you want to determine the harmonics for; you can set the base cursor to a specific time point in the time signal and apply the desired cursor function from then on.

The **measure cursor** is used together with the base cursor to measure and define areas in which the specific cursor functions are then carried out.

The dialogue box is split into two areas:

- In the upper section you will find general settings, which apply for each selected cursor function.
- In the lower section **Select cursor function** you will find the tabs for the individual cursor functions and can make additional settings for each function. By default, you will see the cursor function that is currently selected here:



You have the following options in the area for general settings:

Base cursor or measure cursor values

These fields show the current position of each cursor on the X and Y axis. Click inside a field to enter a value and directly change the cursor's position.

Cursor settings

Synchronous cursors — Activate this option to carry out your selected cursor options - for instance, positioning the base cursor or applying cursor functions - for all loaded signals.

If the checkbox is deactivated, your cursor functions will only be executed for the active signal 26.

Draw guides — Activate this option to show vertical guides at the cursor position in the diagram in addition to the cursor symbols. The guides for base cursor and measure cursor are dashed; those for cursor functions are solid.

If the checkbox is deactivated, you will only see the cursor symbols in the diagram.

In the **Select cursor function** area, your options depend on which function is currently selected. You can specify a cursor function by selecting a tab in the area for **Select cursor function**; you can then adjust further settings for this cursor function. The availability of tabs depends on which Viewer is active. You will find detailed information on individual sub-tabs in the corresponding section:

- **Basis analysis** 53
- **Harmonics** 55
- **Sidebands (Spectrum only)** 56
- **Harmonics with sidebands (Spectrum only)** 56
- **Gear mesh (Spectrum only)** 54
- **Speed** 57

The only cursor function available for the Trend Viewer is **basis analysis**.

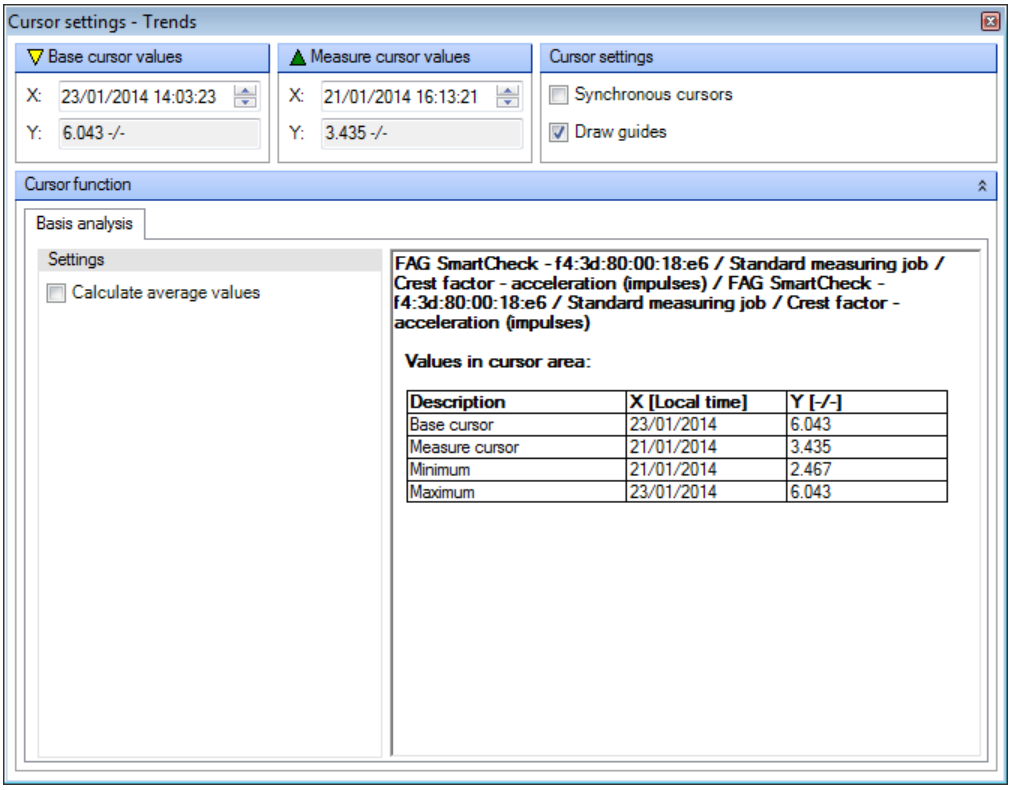
5.3.4.7.1 Basis analysis

The basis analysis is the basic cursor function set by default when the program starts. The scope of this function allows you to view the position values for the base cursor and measure cursor, as well as the minimum and maximum measurement values of the difference range. In the Trend Viewer, for instance, the delta of the Y values and the scatter of minimum and maximum values are used as an initial interference diagnosis.

You can also activate the calculation of average values; this is of particular interest with regard to the manual calculation of characteristic values in the Spectrum Viewer.

Basis analysis tab

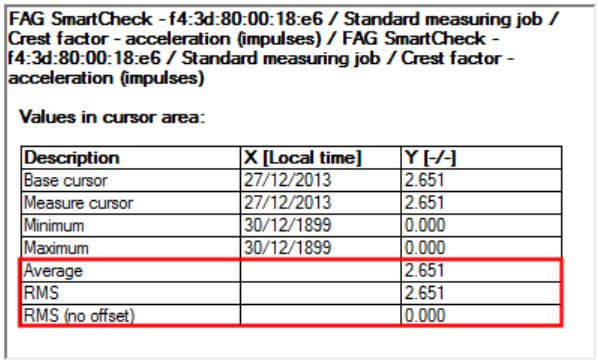
You will find the **Basis analysis** tab in the **Cursor settings** dialogue box under **Select cursor function**:



Here you have the following options:

Calculate average values

If you activate the **Calculate average values**option, the table will be adjusted automatically and the values calculated for the **average value**, **RMS** (root mean square) and **RMS (no offset)** will be added to the table:



This option is deactivated by default, as calculating average values can slow down the display in conjunction with extensive measurements.

Table with values

You can find all the results of the basis analysis in the table. They include:

- Name of the configuration and signal
- Position value of the base cursor
- Position value of the measure cursor

- Minimum and maximum measurement values on the X and Y-axis
- Average values, RMS (root mean square) and RMS (no offset); you will only be able to find these values if you have activated the **Calculate average values**option.

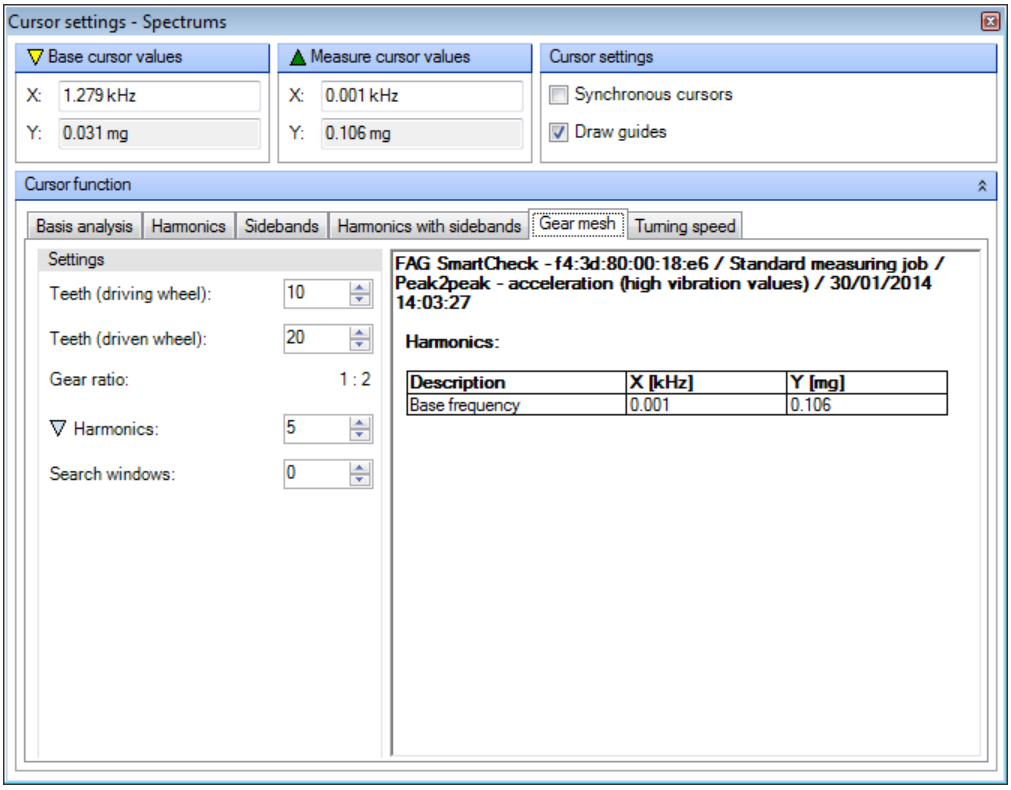
You can select the table - including headers and signal names -, copy it to the clipboard and paste it into a document.

5.3.4.7.2 Gear mesh (Spectrum only)

The **Gear mesh** cursor function is available for gear systems with multiple gears: It allows you to search for gear mesh frequencies depending on the speed.

Gear mesh tab

You can define cursor function details and view the results of the calculation in the **Gear mesh** tab. If you are working with the Spectrum Viewer, you will find the **Gear mesh** tab in the **Cursor settings** dialogue box under **Select cursor function**:



Here you have the following options:

Teeth (driving wheel) Enter the number of teeth for the pinion.

Teeth (driven wheel) Enter the number of teeth for the pinion.

Transmission ratio This value is calculated automatically from the information entered for **Teeth (driving wheel)** and **Teeth (driven wheel)**.

Harmonics Enter the maximum number of harmonics, i.e. the integral multiple of the base cursor, to be displayed in the diagram.

Search windows The search window you define here refers to the number of measurement values surrounding the calculated value; the search window is used to search for peaks within the number of measurement values. The table on the right will adjust automatically to reflect any changes to values you make here.

Table with values Here you can find the X and Y values for all of the harmonics and sidebands displayed in the diagram.

You can select the table - including headers and signal names -, copy it to the clipboard

and paste it into a document.

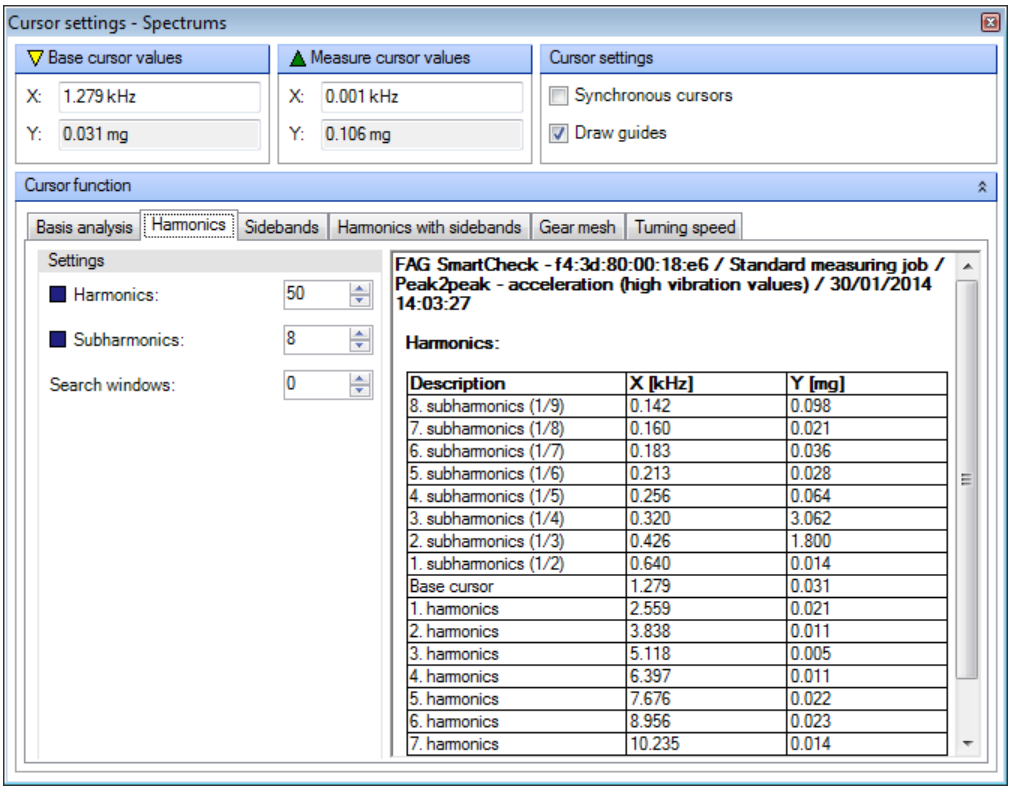
5.3.4.7.3 Harmonics

You can use the **Harmonics** cursor function to establish if and where harmonics exist in the diagram; harmonics are the integral multiple of a vibration. The function is particularly suitable for analyses in the Spectrum Viewer, as damage can manifest itself as patterns in the spectrum.

For example, position the base cursor on the suitable frequency in the Spectrum Viewer; the corresponding harmonics appear automatically as a solid line accompanied by the harmonics symbol. You can define the harmonics symbol in the symbol settings 75; you can access these settings by opening the context menu for the Viewer 21 and then selecting **Settings**.

Harmonics tab

You can define cursor function details and view the results of the calculation in the **Harmonics** tab. You will find the **Harmonics** tab in the **Cursor settings** dialogue box under **Select cursor function**:



Here you have the following options:

Harmonics — Enter the maximum number of harmonics, i.e. the integral multiple of the base cursor, to be displayed in the diagram.

Subharmonics — Enter the maximum number of subharmonics, i.e. the integral multiple of the base cursor, to be displayed in the diagram.

Search windows — The search window you define here refers to the number of measurement values surrounding the calculated value; the search window is used to search for peaks within the number of measurement values. The table on the right will adjust automatically to reflect any changes to values you make here.

Table with values — Here you can find the X and Y values for all of the subharmonics and harmonics displayed in the diagram.

You can select the table - including headers and signal names -, copy it to the clipboard and paste it into a document.

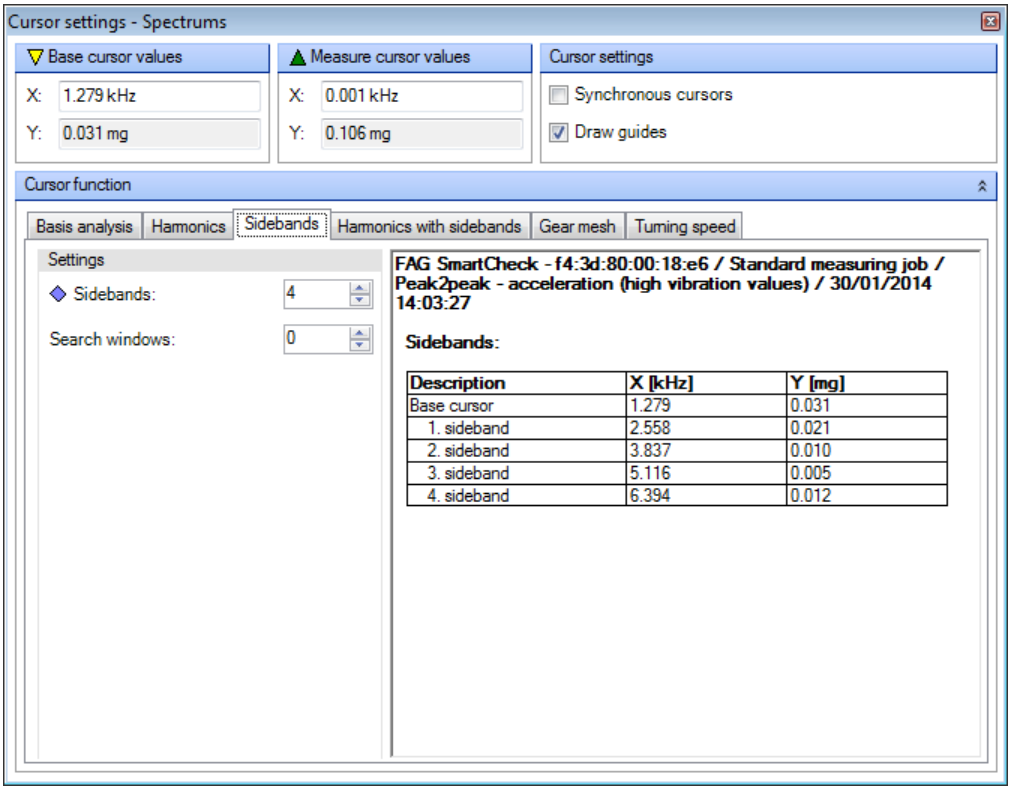
5.3.4.7.4 Sidebands (Spectrum only)

You can use the **Sidebands** cursor function to ascertain other measurement values in sidebands. Sidebands are used, in particular, to detect any damage to the outer race.

Set the base cursor at the desired position in the diagram; the corresponding sidebands are automatically displayed as sideband symbols. You can define the sideband symbol in the symbol settings 75; you can access these settings by opening the context menu for the Viewer 21 and then selecting **Settings**.

Sidebands tab

You can define cursor function details and view the results of the calculation in the **Sidebands** tab. If you are working with the Spectrum Viewer, you will find the **Sidebands** tab in the **Cursor settings** dialogue box under **Select cursor function**:



Here you have the following options:

Sidebands	Determine the number of sidebands to be calculated for the current base cursor position. The table on the right will adjust automatically to reflect any changes to values you make here.
Search windows	The search window you define here refers to the number of measurement values surrounding the calculated value; the search window is used to search for peaks within the number of measurement values. The table on the right will adjust automatically to reflect any changes to values you make here.
Table with values	Here you can find the X and Y values for all of the sidebands displayed in the diagram. You can select the table - including headers and signal names -, copy it to the clipboard and paste it into a document.

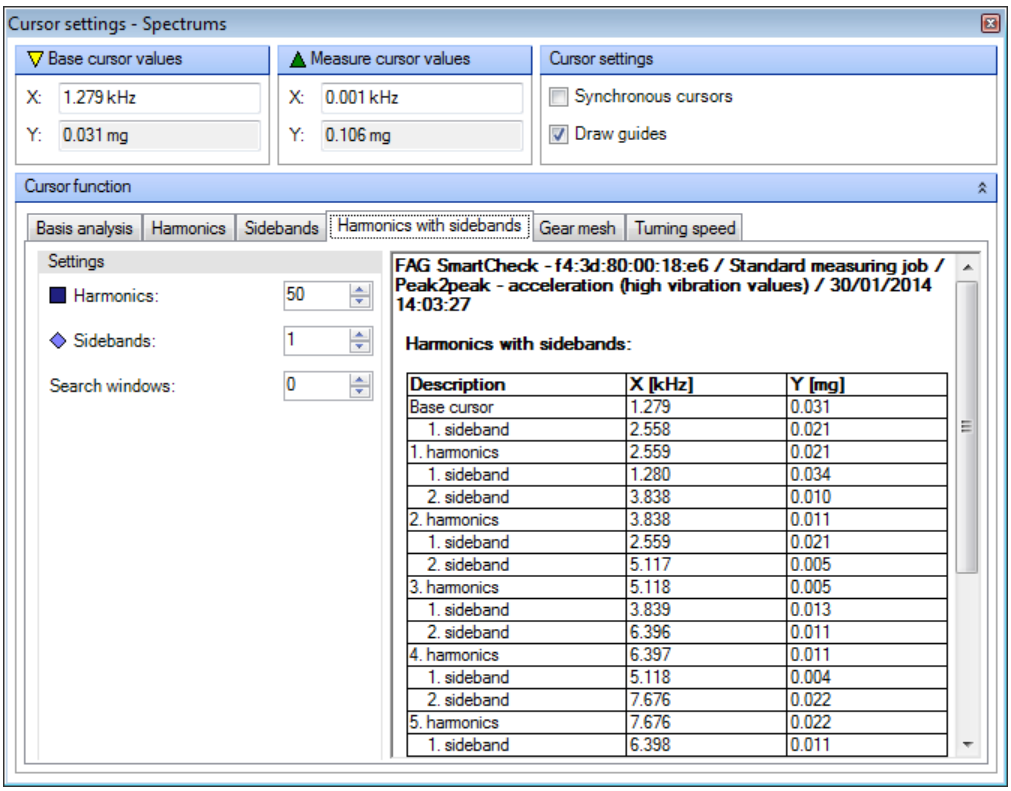
5.3.4.7.5 Harmonics with sidebands (Spectrum only)

The cursor function **Harmonics with sidebands** combines the two cursor functions **Harmonics** and **Sidebands**. You can therefore ascertain the harmonics and sidebands simultaneously for the position of the base cursor.

To do so, position the base cursor at the desired location in the diagram; the corresponding harmonics and sidebands are automatically displayed as a solid line and appear together with the corresponding cursor symbols. You can define the cursor symbols in symbol settings 75; to access these settings, open the context menu for the Viewer 21 and select **Settings**.

Harmonics with sidebands tab

You can define cursor function details and view the results of the calculation in the **Harmonics with sidebands** tab. If you are working with the Spectrum Viewer, you will find the **Harmonics with sidebands** tab in the **Cursor settings** dialogue box under **Select cursor function**:



Here you have the following options:

Harmonics — Enter the maximum number of harmonics, i.e. the integral multiple of the base cursor, to be displayed in the diagram.

Sidebands — Determine the number of sidebands to be calculated for the current base cursor position. The table on the right will adjust automatically to reflect any changes to values you make here.

Search windows — The search window you define here refers to the number of measurement values surrounding the calculated value; the search window is used to search for peaks within the number of measurement values. The table on the right will adjust automatically to reflect any changes to values you make here.

Table with values — Here you can find the X and Y values for all of the harmonics and sidebands displayed in the diagram.

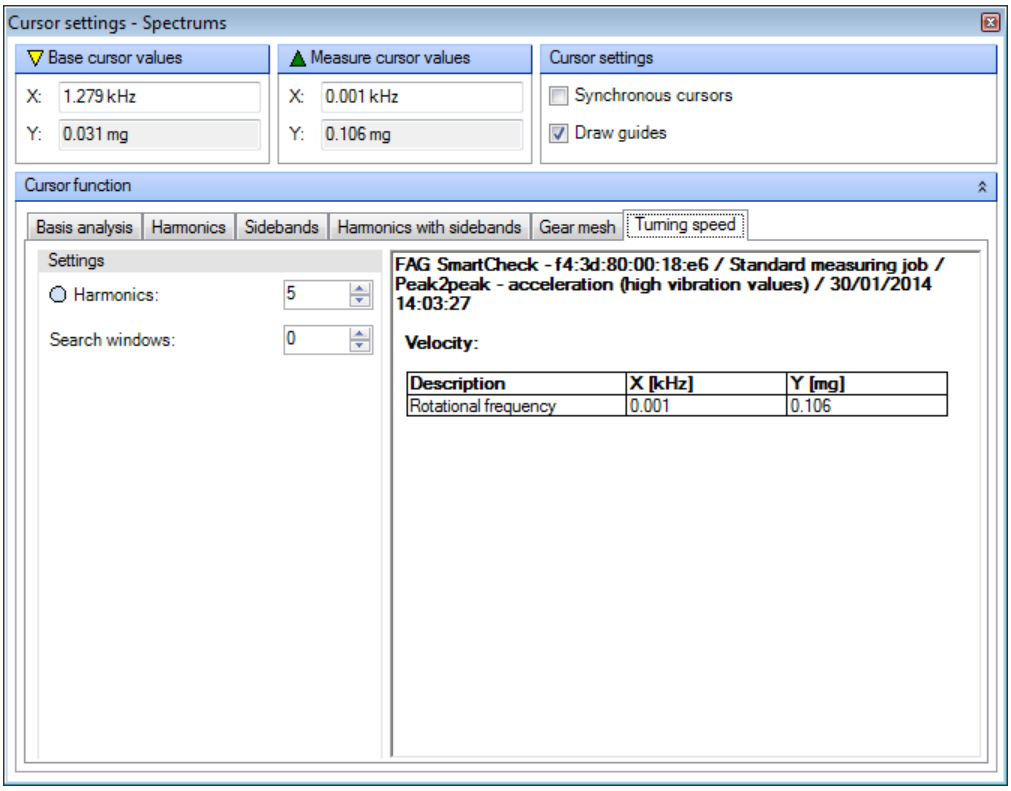
You can select the table - including headers and signal names -, copy it to the clipboard and paste it into a document.

5.3.4.7.6 Speed

You can establish harmonics depending on speed using the cursor function **Speed**. The speed is automatically used as a basis when working with the Spectrum Viewer. When working with the Time Signal Viewer, you will need to select a speed range using the base and measure cursors.

Speed tab

You can define cursor function details and view the results of the calculation in the **Speed** tab. You will find the **Speed** tab in the **Cursor settings** dialogue box under **Select cursor function**:



Here you have the following options:

Harmonics	Enter the maximum number of harmonics, i.e. the integral multiple of the base cursor, to be displayed in the diagram.
Search windows	The search window you define here refers to the number of measurement values surrounding the calculated value; the search window is used to search for peaks within the number of measurement values. The table on the right will adjust automatically to reflect any changes to values you make here.
Table with values	Here you can find the X and Y values for all of the harmonics displayed in the diagram. You can select the table - including headers and signal names -, copy it to the clipboard and paste it into a document.

5.3.4.8 Position cursor

If you click on  in the toolbar, a list will open, from which you can select a positioning option for the cursor. These options assist you in positioning the base cursor in the diagram precisely. Here you have the following options:

- **Free:** The cursor position is determined based on pixel values and is freely selectable, even between two measurement values.
- **Next sample:** The base cursor is set to the next measurement value.
- **Nearest peak:** The base cursor is set to the nearest peak.
- **Tenths:** The area between two measurement values on the X-axis is divided into tenths; the base cursor is set to the nearest tenth**.**
- **Hundredths:** The area between two measurement values on the X-axis is divided into hundredths; the base cursor is set to the nearest hundredth**.**

- If you click in the diagram, the cursor always moves to the nearest peak, irrespective of the setting you make here. Precise positioning, in line with the options in this list, occurs by dragging the base cursor. Move the mouse over the base cursor until you see the double-headed arrow:
  ↔
  Click and drag the base cursor to the required position. The interim steps when pulling are defined by the selected positioning option.
- You can show/hide the cursor by clicking on the toolbar.

5.3.4.9 Select diagram view

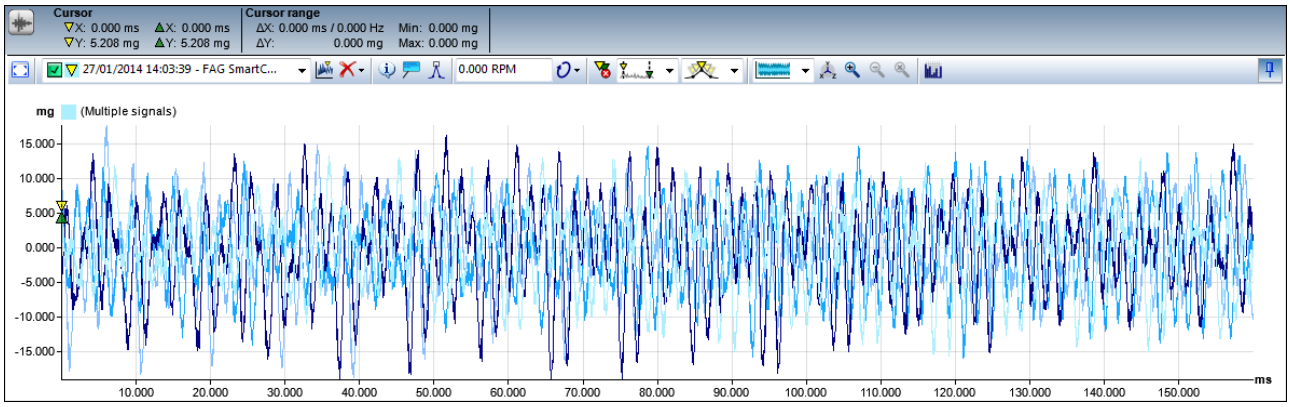
If you click in the toolbar, a list opens in which you can select how the diagram should be displayed in the active Viewer. To do so, select the appropriate option from the selection list. The available options depend on the active viewer. You will find detailed information on individual options in the corresponding section:

- Overlapping 59
- Stack 59
- Matrix 60
- Wireframe (Spectrum only) 60
- Waterfall (Spectrum only) 61
- 2D spectrogram (Spectrum only) 61
- 3D spectrogram (Spectrum only) 61

Zoom functions are available in every view.

5.3.4.9.1 Overlapping

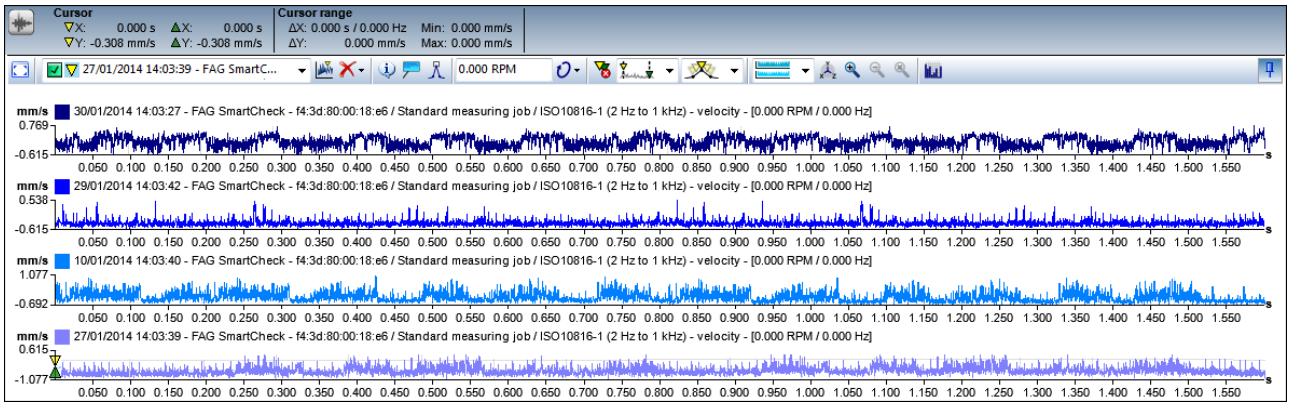
The **Overlapping** view is available for every Viewer. If you select this option, all data that is open in the active Viewer will be displayed together in one co-ordinate system:



This will provide you with an initial rough overview where you can view the minimum and maximum values for all of the displayed data directly on the axes.

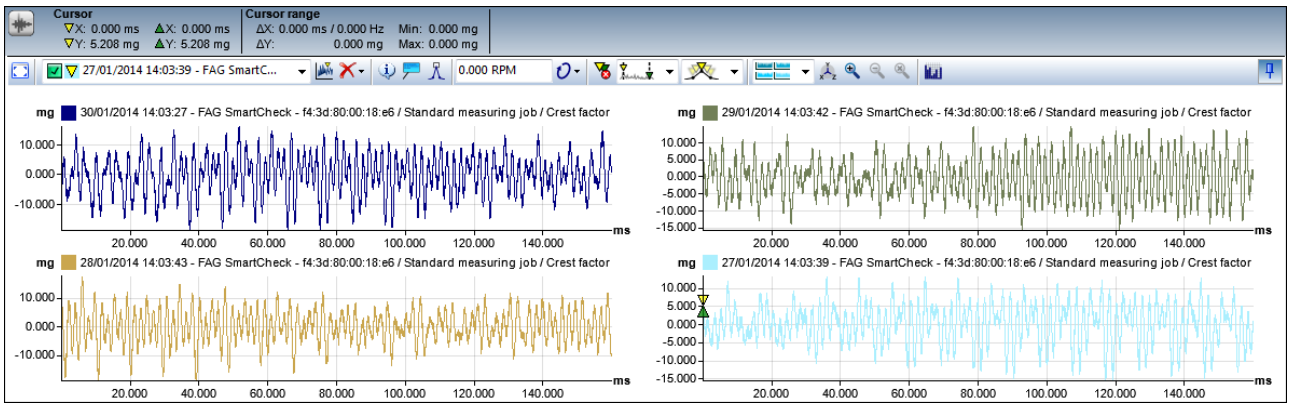
5.3.4.9.2 Stack

The **List** view is available for every Viewer. If you select this option, each dataset open in the active Viewer will receive its own co-ordinate system. All of the co-ordinate systems appear as a stack in the viewer:



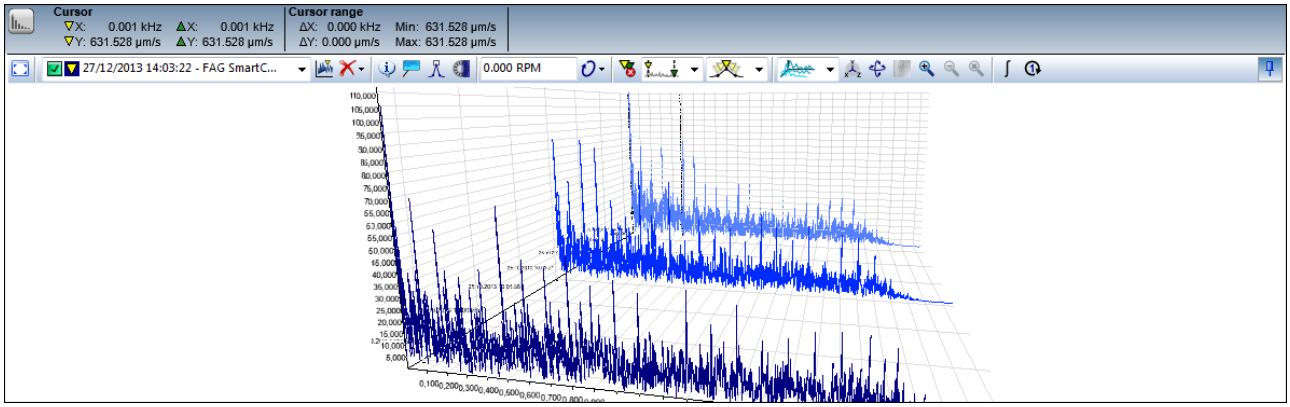
5.3.4.9.3 Matrix

The **Matrix** view  is available for every Viewer. If you select this option, each dataset open in the Viewer will receive its own co-ordinate system. Up to three co-ordinate systems are displayed as a stack; four or more are displayed in the viewer as a multi-column matrix:



5.3.4.9.4 Wireframe (Spectrum only)

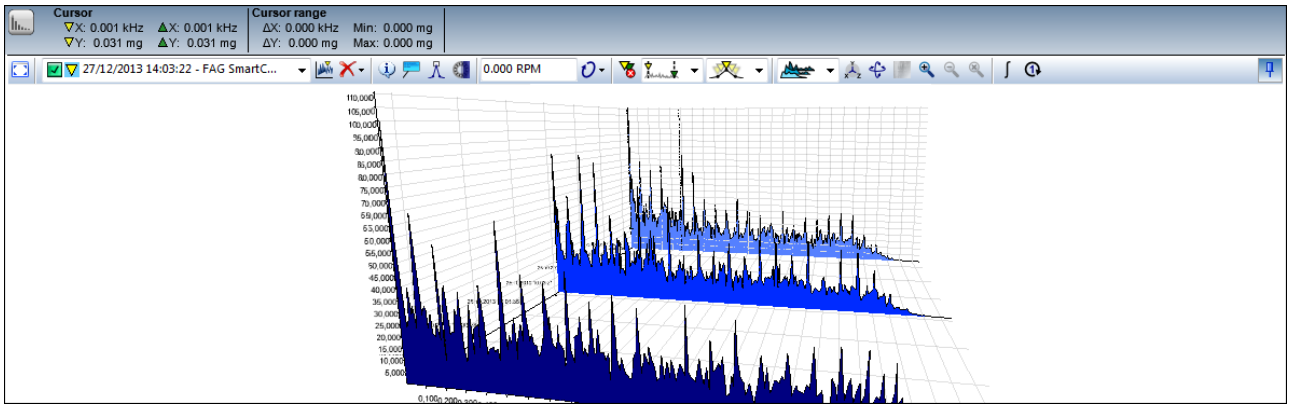
The **Wireframe**  view is only available in the Spectrum Viewer. If you select this option, all of the spectrums that are open in the active Viewer will be displayed on one wireframe:



You can amend the view options for your application by modifying the camera settings 63 and spectrogram settings 63.

5.3.4.9.5 Waterfall (Spectrum only)

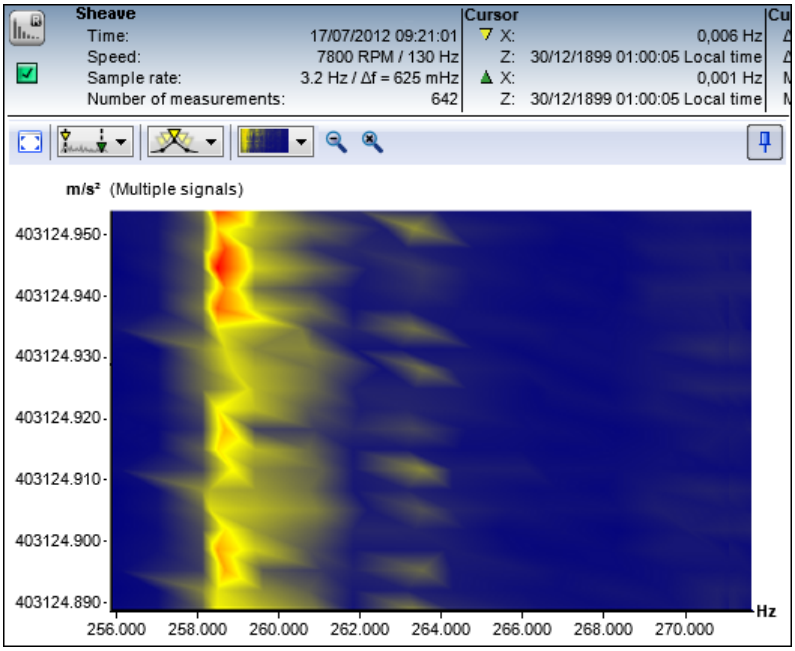
The **Waterfall** view  is only available in the Spectrum Viewer. If you select and set this option, all of the spectrums that are open in the active Viewer will be displayed as a waterfall:



You can amend the view options for your application by modifying the camera settings 63 and spectrogram settings 63.

5.3.4.9.6 2D spectrogram (Spectrum only)

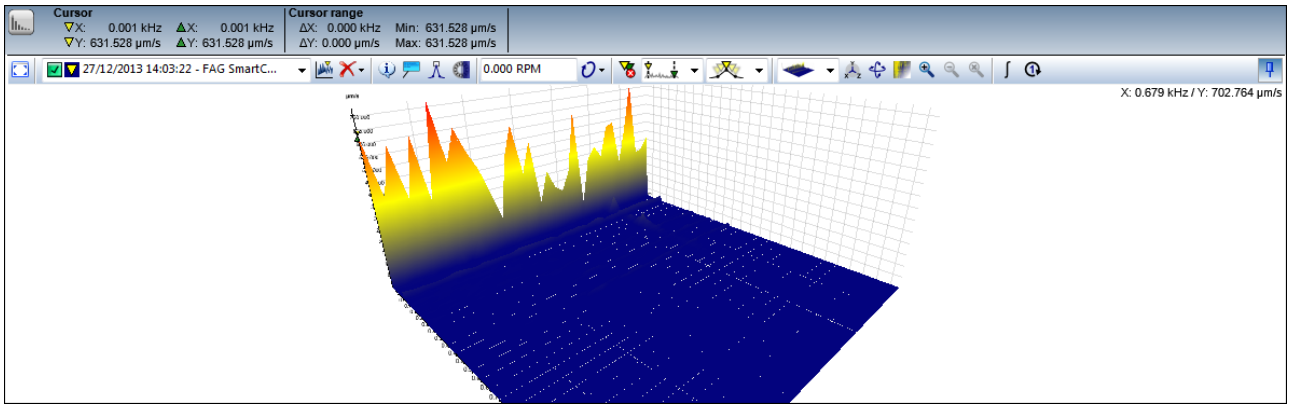
The **2D spectrogram** view  is only available in the Spectrum Viewer. If you select and set this option, all of the spectrums that are open in the active Viewer will be displayed as a two-dimensional spectrogram:



You can amend the view options for your application by modifying the camera settings 63 and spectrogram settings 63.

5.3.4.9.7 3D spectrogram (Spectrum only)

The **3D spectrogram** view  is only available in the Spectrum Viewer. If you select this option, all of the spectrums that are open in the active Viewer will be displayed as a three-dimensional spectrogram:

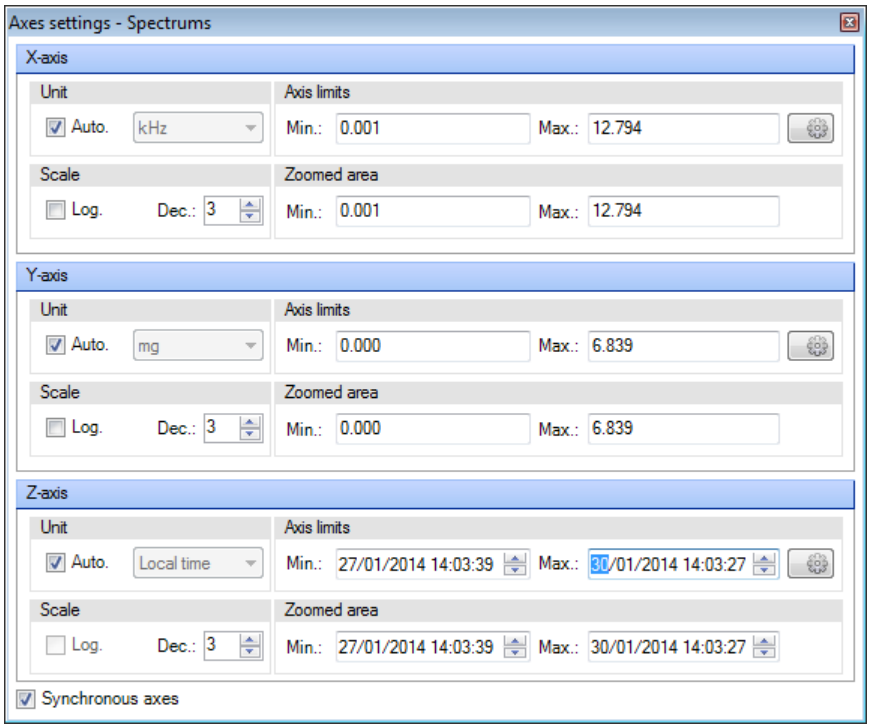


You can amend the view options for your application by modifying the camera settings 63 and spectrogram settings 63.

5.3.4.10 Edit axes settings

If you click  in the toolbar, a dialogue box opens in which you can specify details for the X and Y axes. Your settings always apply to the axes of the active viewer.

You can specify and change standard settings, e.g. in the **Unit** or **Scaling** area under Viewer settings 73; to do so, open the context menu for the Viewer 21 and go to **Settings**.



You can specify the following settings for the axes:

Unit

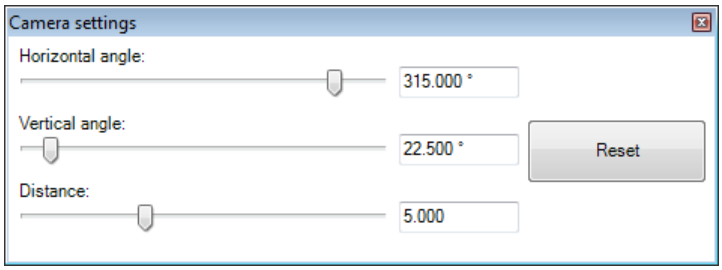
Here you can determine the unit used to display the axis. You have the following options:

- **Auto:** Activate this field to prompt the FAG SmartUtility Viewer software to use the unit best suited to displaying the current signal types. The FAG SmartUtility Viewer software automatically decides which unit prefix provides the best, i.e. the shortest possible, representation in the diagram.
- **Selection list:** You can choose a unit for the axis display from the list. The availability of units depends on the unit profile 76. To change or view unit profiles, open the context menu for the Viewer 21 and select the **Settings** command.

Scaling	This is where you set the scale for the axes: • **Log**: This option activates logarithmic scaling. (Spectrum Viewer only) • **Dec.**: Use this option to define how many spaces should follow the decimal point.
Axes limits	Use this option to determine the unit range to be shown in the diagram. You have the following options: • Use **Min.** and **Max.** to define the value range you want to show in the diagram. •  : Click this symbol to access the following functions: ▪ **Standardise:** The axis range covers the minimum and maximum for all displayed signals. ▪ **Reset:** This command resets the original axis limits.
Zoomed area	Use this function to define the area you want to zoom within the axis limits. Zooming out again does not change the defined axis limits.
Synchronous axes	If you activate this option, your axis settings will apply to all loaded data. Uncheck the box if you want to apply your settings to the X and Y axes of the active data only.

5.3.4.11 Modify camera settings (Spectrum only)

The function **Modify camera settings**  is only available in the toolbar for the Spectrum Viewer and only for the view options **Wireframe**, **Waterfall**, **2D spectrogram** and **3D spectrogram**. The function opens a dialogue box, in which you can edit the camera settings for these display options:

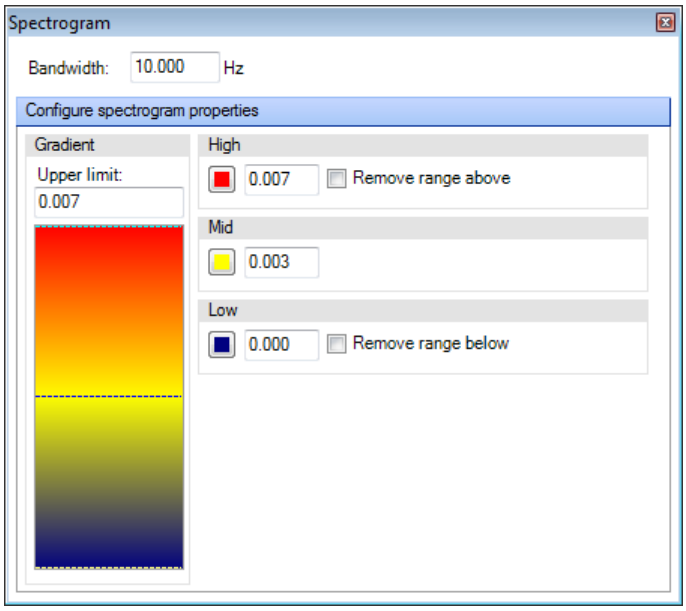


Use these settings to define the perspective from which you view the diagram. Here you have the following options:

- Use the **Horizontal angle** to determine the rotation of the diagram, i.e. which side you will view it from.
- Use the **Vertical angle** to determine whether you will view the diagram more at eye level (small value) or more so from above (greater value).
- The **Distance** refers to the distance between the camera and the centre of the diagram. The maximum value is 10.
- Click on **Reset value** to reset the camera settings to the default values.

5.3.4.12 Change spectrogram settings (Spectrum only)

The function **Change spectrogram settings**  is only available in the toolbar for the Spectrum Viewer. The function opens up a dialogue box, in which you can define the basic spectrogram properties:



Here you have the following options:

Bandwidth

Your graphics board would need a lot of disk space if you were to display every dot when working with many signals. This is avoided by dividing the spectrogram into sections and displaying the maximum value for each one.

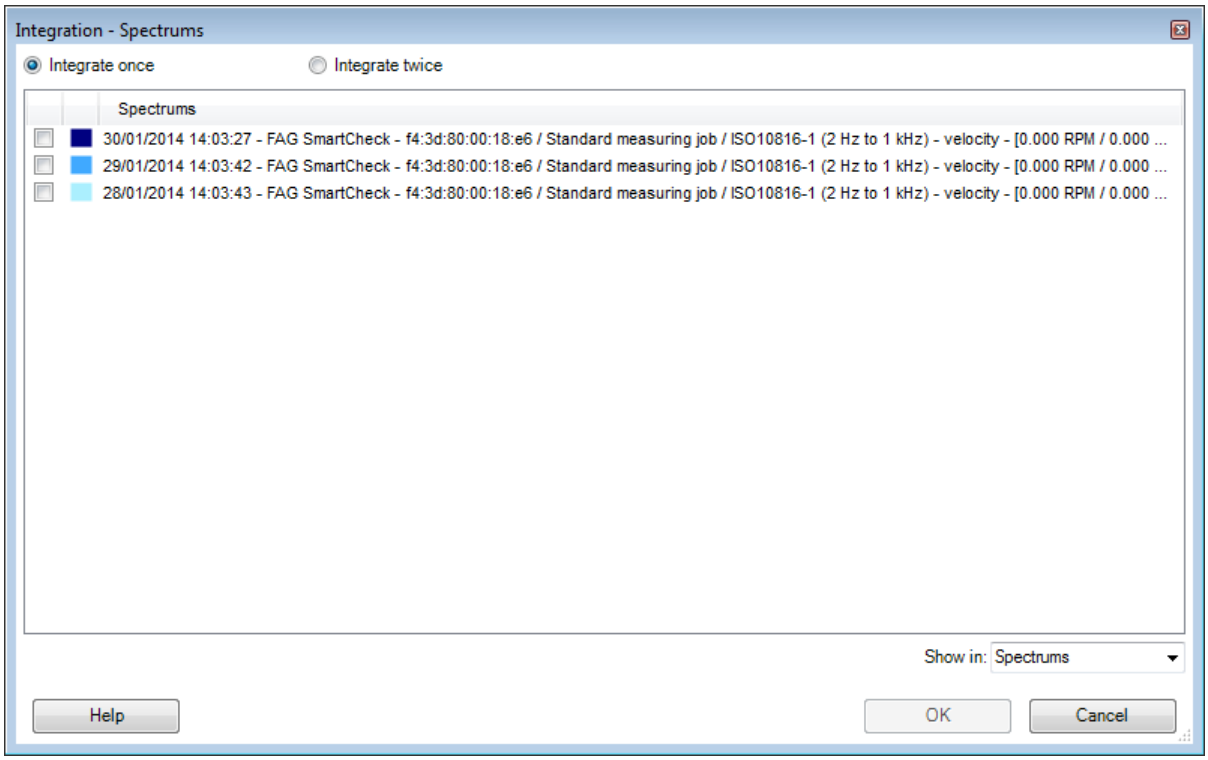
If you decrease the value for **bandwidth**, you increase the number of sections; this in turn increases the number of displayed values and improves the accuracy of the display. With regard to weaker graphics boards, you can raise the bandwidth to achieve a balance between accurate display and required disk space.

Spectrogram properties

- **Gradient:** You can use the value for the **upper limit** to define the maximum value at which the displayed range of the Y-axis should end. Additionally, the dashed line tells you at a glance where your values for **Upper**, **Middle** and **Lower** are positioned on the Y-axis.
- For instance, you can set the values for **Upper** and **Lower** at your alarm limits.
- The options **Remove upper range** and **Remove lower range** allow you to remove these ranges from the diagram and concentrate on analysing the middle range.
- Clicking on the colour symbols for **Upper**, **Middle** and **Lower**opens the standard colour dialogue box. You can amend the colour of the relevant area here, for example to achieve a better contrast.

5.3.4.13 Integrate signals (spectrum)

The function **Integrate signals** is available in the toolbars for the Time Signal Viewer and the Spectrum Viewer. The function opens a dialogue box, in which you can integrate time signals:



Here you have the following options:

Integrate once Activate this option to view all the time signals in the list that can be integrated once.

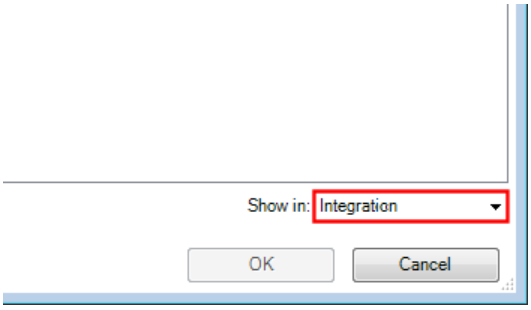
Integrate twice Activate this option to view all the time signals in the list that can be integrated twice.

Time signals In this list you will see the time signals that are available for the integration option selected above. Use the checkbox to activate the required time signals for the integration.

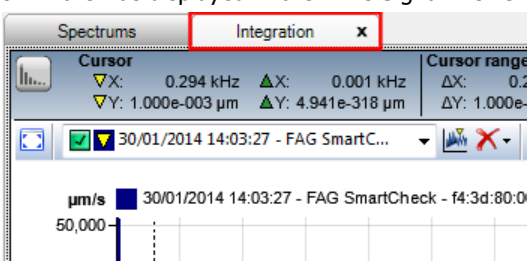
Show in You have the following options to show the integrated time signals:

Time signals: Use this option to show the integrated time signals with all the other loaded time signals in the Time Signal Viewer.

Own tab: You can activate the option **Time signals** and overwrite it with its own name:



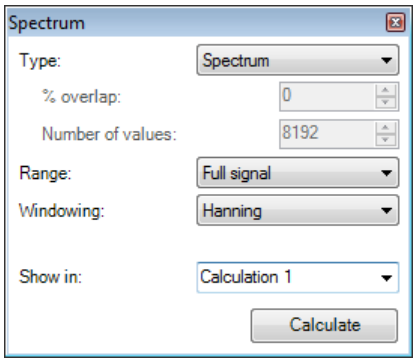
The integrated time signals will then be displayed in the Time Signal Viewer on their own tab:



Click on **OK** to confirm your entries and integrate the selected time signals.

5.3.4.14 Calculate Spectrum (time signal only)

The function **Calculate Spectrum** is only available in the toolbar for the Time Signal Viewer. Use this function to produce a spectrum or spectrogram from the time signals in the Time Signal Viewer.



Here you have the following options:

Type

Here you can define whether the loaded time signals should be used to generate a **spectrum** or a **spectrogram**.

% overlap (spectrogram only)

Enter the percentage by which the individual spectrums of the spectrogram should overlap each other. The ideal value also depends on the choice of **windowing**; this is where you ensure that all important measurement values are taken into account during spectrum calculation.

Number of measurement values (spectrogram only)

Enter the number of measurement values after which the next spectrum begins.

Area

Here you can define whether the spectrogram/spectrum should be based on the complete time signal or only on the cursor range. The cursor range is defined by the base cursor and the measure cursor.

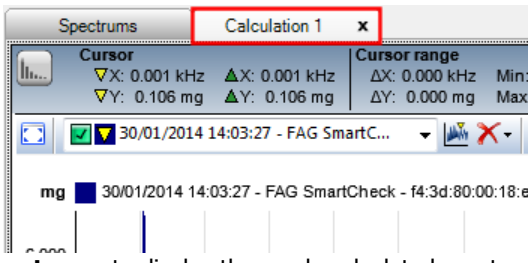
Windowing

Defines the windowing function to be used when the spectrogram/spectrum is generated. You can choose between **No windowing**, **Hanning** and **Hamming**.

Show in

Here you can define where the spectrogram/spectrum is displayed. You have two options:

- You can enter a name in the empty field. The newly calculated spectrogram is then displayed on its own tab in the Spectrum Viewer. The tab is given the name you enter here:



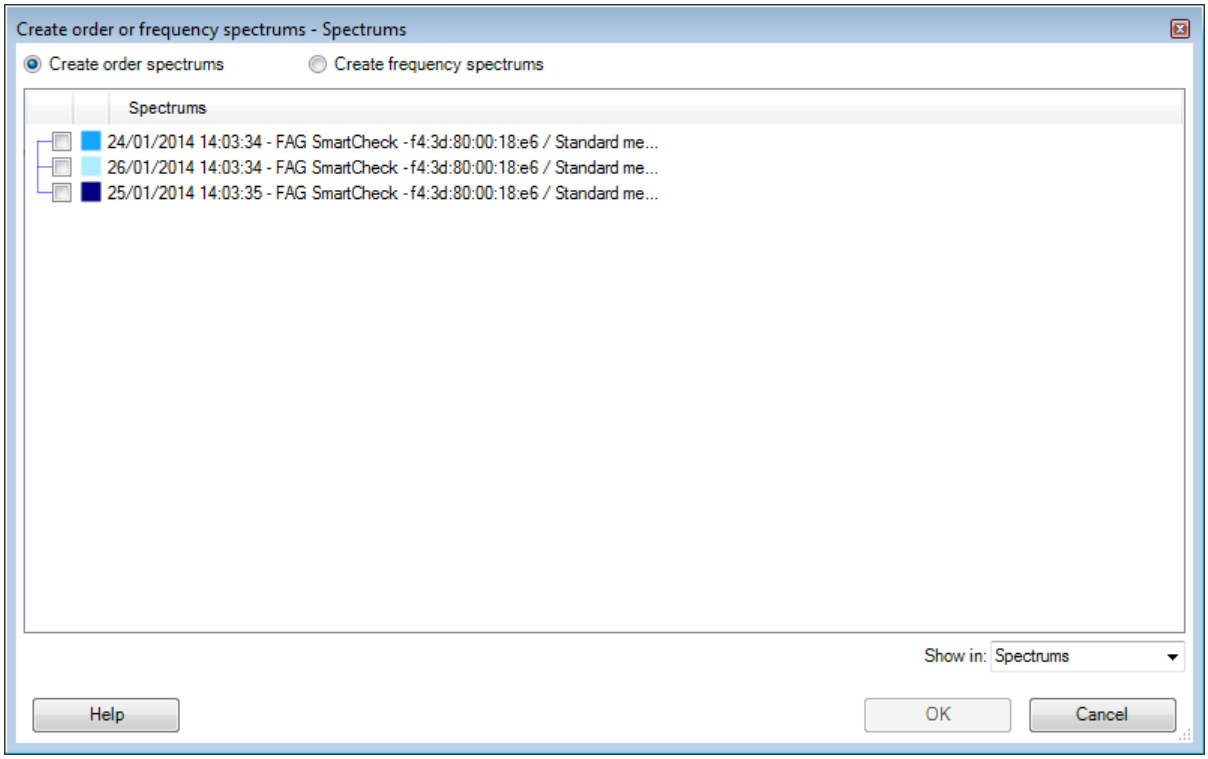
- You can select **Spectrums** to display the newly calculated spectrogram together with all the other open data in the Spectrum Viewer.

Calculate

Click here to create the new spectrogram/spectrum. Depending on the settings in Show in, it will then be displayed in the Spectrum Viewer or on its own tab in the Spectrum Viewer

5.3.4.15 Calculate order spectrum (Spectrum only)

The function **Calculate order spectrum** is only available in the toolbar for the Spectrum Viewer. The function opens a dialogue box, in which you can calculate the order spectrums and the frequency spectrums:



Here you have the following options:

Generate order spectrums — Activate this option to see all the spectrums in the list from which an order spectrum can be generated.

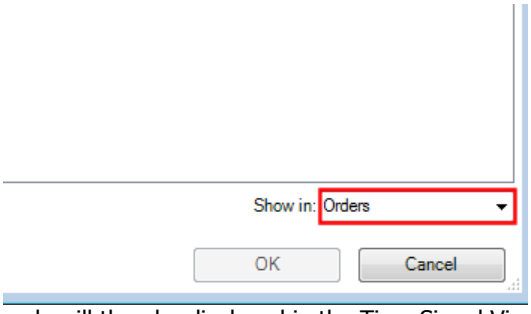
Generate frequency spectrums — Activate this option to see all the spectrums in the list from which a frequency spectrum can be generated.

Spectrums — In this list you will see the spectrums that are available for the option selected above. Use the checkbox to activate the required spectrums for the operation.

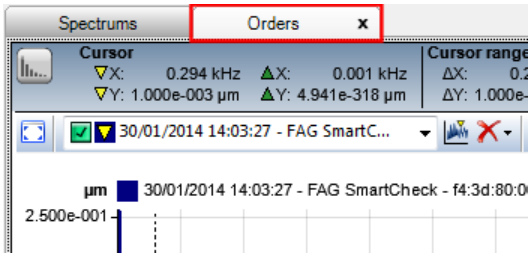
Show in — You have the following options to display the generated spectrums:

Spectrums: Use this option to display the generated spectrums with all other loaded spectrums in the Spectrum Viewer.

Own tab: You can activate the option **Spectrums** and overwrite it with its own name:



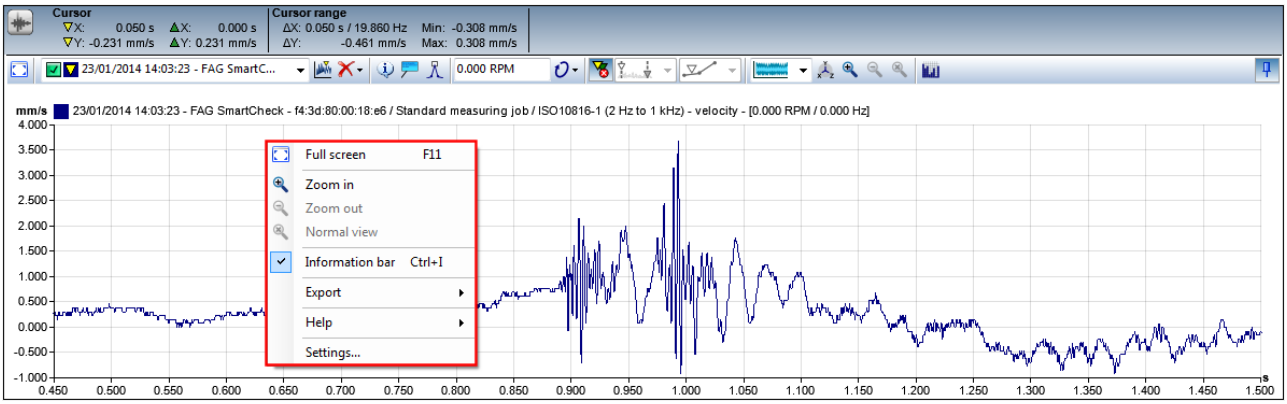
The integrated time signals will then be displayed in the Time Signal Viewer on their own tab:



Click on **OK** to confirm your entries and generate the required spectrums.

5.3.4.16 Exporting diagrams

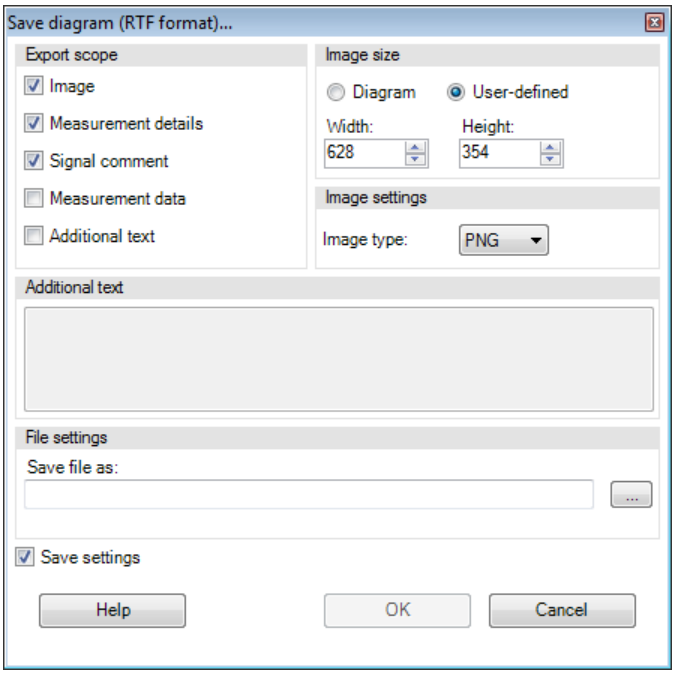
Each viewer allows you to export the diagram and its associated information. You can export the diagram in RTF format, image format and CSV format; you can also choose to save it as a file or copy it to the clipboard. The corresponding commands are called up by right-clicking in the context menu for the viewer:



You will find detailed information on exporting different formats in the following sections. We have described the Save options in each case to give the explanations a clearer layout. Copying will remove the **file settings** from the dialogue.

Export diagram in RTF format

When you save or copy a diagram in RTF format, you can subsequently open it in Office applications or paste it into an Office file. To export, select **Export > Save diagram (RTF format)** or **Export > Copy diagram (RTF format)** from the context menu for the Viewer. This opens the corresponding dialogue box:



Here you have the following options:

Export scope

Use these options to determine exactly what the saved or copied diagram should contain:

- **Image**: Exports the diagram together with the information bar.
- **Measurement details**: Exports additional measurement information such as the sample rate.
- **Signal comment**: Exports the signal comment provided you have defined one in the **Comments** 41 tab; the signal comment usually describes the result after an investigation of a list of signals and acts primarily as an overview.
- **Measurement data**: Exports all of the values for the X and Y coordinates and, where necessary, for the Z coordinates (e.g. for 3D or 2D spectrogram) in tabular form.
- **Additional text**: Activating this option allows you to enter a comment on the diagram in the **Additional text** field. Your comment will then also appear in the exported diagram.

Image size

This is where you define the size of the exported diagram:

- **Diagram**: Sets the size to the Viewer's current dimensions. The diagram may be exported in full screen size in certain circumstances.
- **User-defined**: Allows you to determine the diagram's **width** and **height** yourself. If you select this option, the width and height are optimised as standard for DIN A4 portrait, but can be changed.
  If you increase only the value for the width, the result will be that more values will be displayed on the X-axis and the resolution will be improved.

Image settings

Here you can define whether the image exported as a part of the RTF export should be in PNG or WMF format.

Additional text

You can add further comments on the diagram here if you have activated the **Additional text** option under **Export scope**.

File settings (Save only)

You can save files in RTF format.

Enter the name under which you want to save the diagram.

Click ... to select the directory to which you want to save the file containing the diagram.

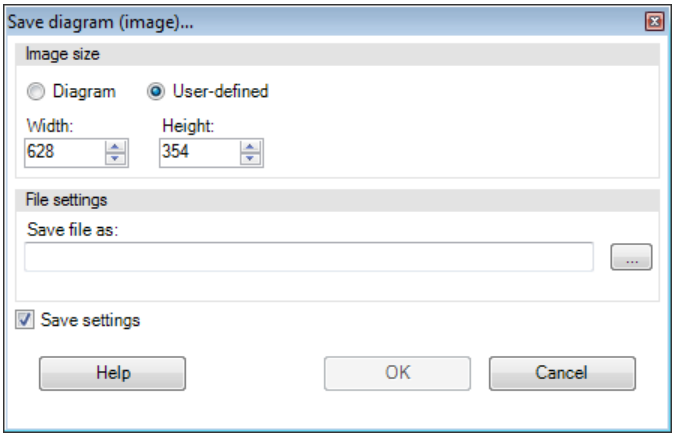
Save settings

If you select this option, the settings you made above will be saved as a default. These settings will be applied automatically every time you call up this export function.

Click **OK** to confirm your settings and export the diagram. Depending on the selected function, you can open the saved file in an Office program or paste the clipboard contents into a Word file, for instance.

Export diagram as an image

If you save or copy the diagram as an image, you will be able to open it in an image editing program or paste it into files that support images. To export, select **Export > Save diagram (image)** or **Export > Copy diagram (image)** from the context menu for the Viewer. This opens the corresponding dialogue box:



Here you have the following options:

Image size

This is where you define the size of the exported diagram:

- **Diagram**: Sets the size to the Viewer's current dimensions. The diagram may be exported in full screen size in certain circumstances.
- **User-defined**: Allows you to determine the diagram's **width** and **height** yourself. If you select this option, the width and height are optimised as standard for DIN A4 portrait, but can be changed.
  If you increase only the value for the width, the result will be that more values will be displayed on the X-axis and the resolution will be improved.

File settings

You can save files in PNG format.

Enter the name under which you want to save the diagram.

Click ... to select the directory to which you want to save the file containing the diagram.

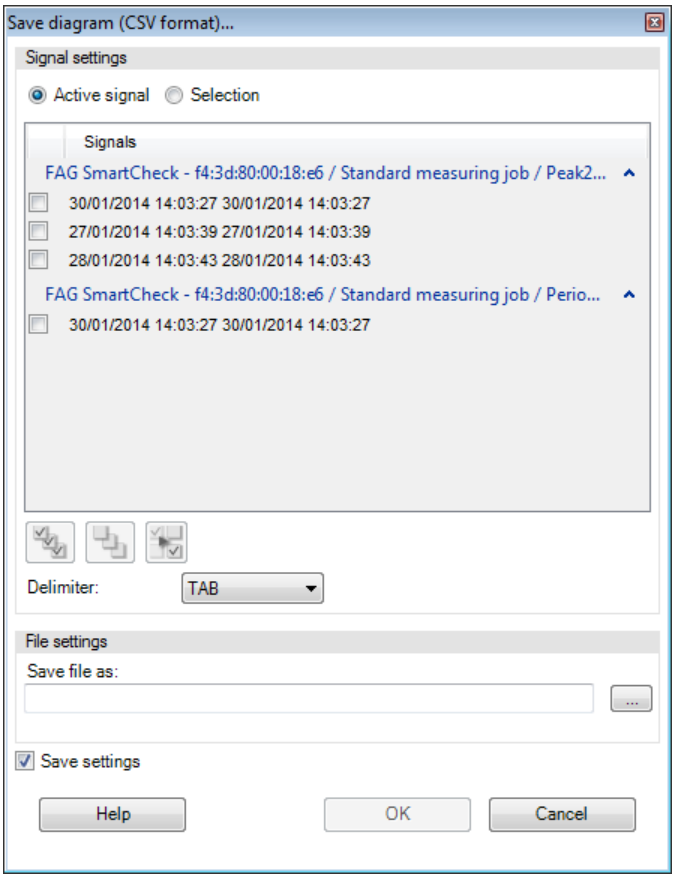
Save settings

If you select this option, the settings you made above will be saved as a default. These settings will be applied automatically every time you call up this export function.

Click **OK** to confirm your settings and export the diagram. Depending on the selected function, you can open the saved file in an image editing program or paste the clipboard contents into a Word file, for instance.

Export diagram in CSV format

Saving or copying the diagram in CSV format will export all of the X and Y coordinates and, where indicated, the Z coordinates. You can then open them as a table, for instance in MS Excel, or paste them into an appropriate file. To export, select **Export > Save diagram (CSV format)** or **Export > Copy diagram (CSV format)** from the context menu for the Viewer. This opens the corresponding dialogue box:



Here you have the following options:

Signal settings

Here you can determine the signal types you want to export the diagram for:

- **Active signal** 30: You will find the active signal in the signal selection list for the Viewer where it is marked with a yellow triangle.
- **Selection**: Activate this option if you want to select specific signals for export. You can choose from all of the currently loaded signals. Make your selection by activating the checkbox for each signal you want to export. You can also make use of the quick selection options offered by the buttons below the list:

  : Selects all of the signals for export.

  : Selects none of the signals for export.

  : Inverts the current selection, i.e. checks unchecked boxes and unchecks checked boxes.

Use the **Delimiter** option to determine how coordinate values should be separated in the exported format. You can choose between the options **TAB**, comma **,** and semicolon **;**.

File settings

You can save files in CSV format.

Enter the name under which you want to save the diagram.

Click ... to select the directory to which you want to save the file containing the diagram.

Save settings

If you select this option, the settings you made above will be saved as a default. These settings will be applied automatically every time you call up this export function.

For the CSV export of the trend, the data for the X axis is transferred as numbers in a time stamp column.

Example: **41884,4173678241**

These values correspond to an EXCEL-specific date and time format:

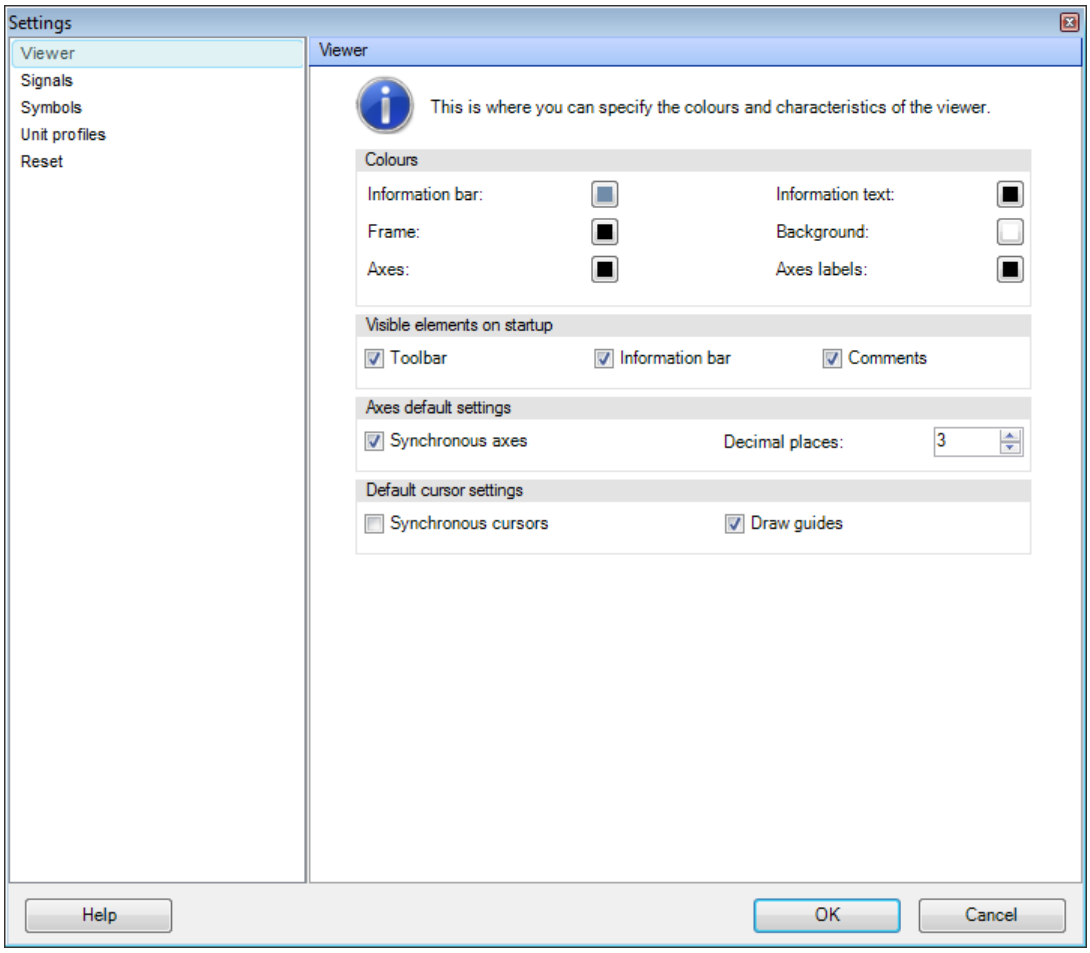
- **Number before the comma**: number of days since 01/01/1900
- **Number after the comma**: time

To convert this format to the normal date and time format, proceed as follows:

1. Highlight the time stamp column.
2. Right-click and select **Format cells**.
3. On the **Numbers** tab, select the **User-defined** category and then on the right-hand side select the required **Type**, i.e. **DD/MM/YYYY hh:mm:ss**. The example above would then be converted as follows: **09/02/2014 10:01:01**.

5.3.5 Change program settings

You can find the program settings for the FAG SmartUtility Viewer software in the context menu, which you can access by right-clicking in any Viewer. In the context menu, select the **Settings** command to open the following dialogue box:



You can select the area for which you want to view or change settings from the list on the left. You will then see the settings you can change for the area in question on the right. You can change the settings for the following areas:

- Viewer 73
- Signals 74
- Symbols 75
- Unit profiles 76
- Reset 77

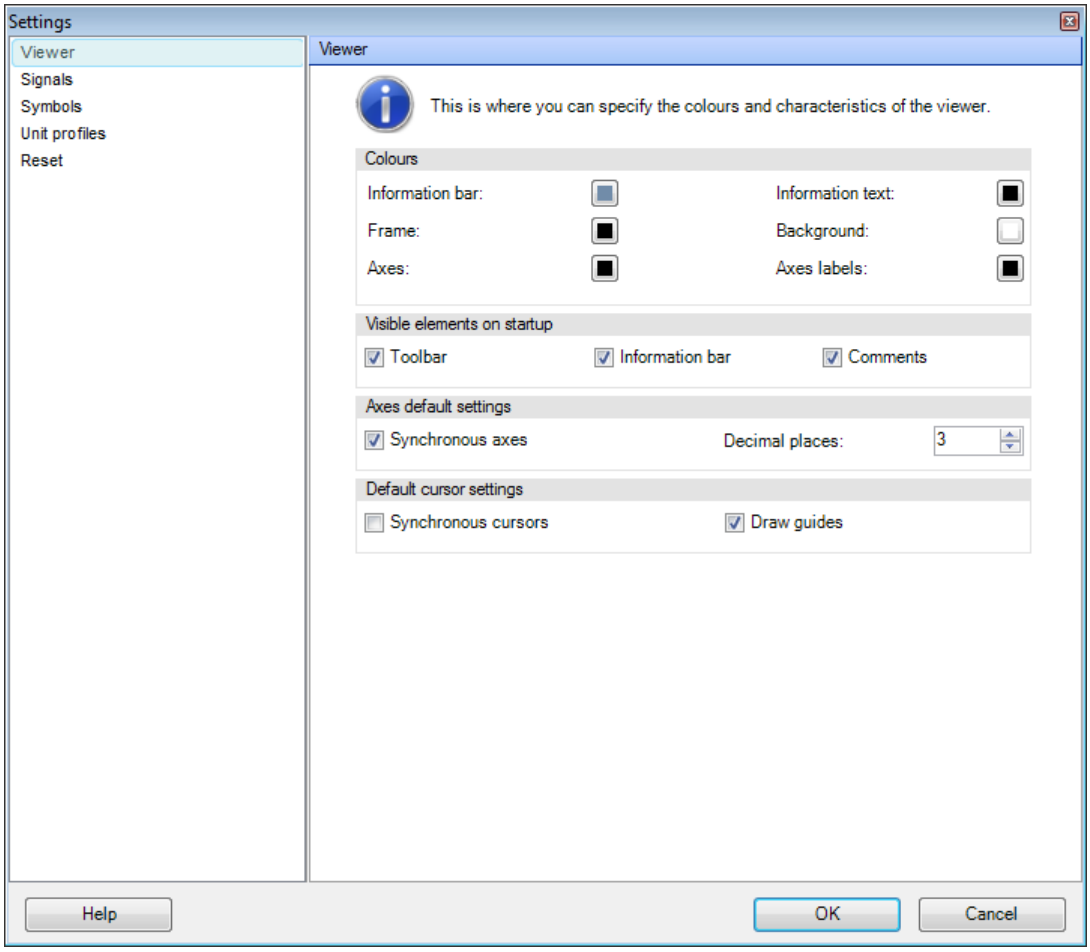
You will find detailed information on settings options in the following sections.

Some of these settings relate to the colours used in the FAG SmartUtility Viewer software, e.g. for cursor symbols and signal displays. Follow these steps to change a colour:

1. Click the colour symbol you want to change. This opens the standard colour dialogue box.
2. Choose the colour you want to assign to the cursor symbol or signal display. Here you have the following options:
   - Click one of the **basic colours** to select it.
   - Click inside the colour chart to select a colour.
   - Use the slide on the far right to change the colour shade.
   - Directly enter the required values for **red**, **green** and **blue** or for **colour shade**, **saturation** and **brightness**.
3. Confirm your changes with **OK**.

5.3.5.1 Viewer settings

You can determine how the Viewers should look on start-up and their standard behaviour in the **Viewers** area.



You have the following options:

Colours

This is where you determine the colours to be used for specific areas of the Viewer, e.g. the background colour of the **information bar** or the colour of the **axes labels**. Click on the corresponding colour symbol to open the standard colour dialogue box 73, where you can specify your settings.

Visible elements on startup

Here you can define which elements should be visible on startup.

The **toolbar** and **information bar** elements are activated as standard; **comments** are not visible.

Axes default settings

Here you can define the appearance of the X and Y axes on startup:

- **Synchronous axes:** When this option is activated, any changes made in the **Axes** 62 tab will always apply to the axes of every diagram in the active Viewer. If you deactivate the checkbox, any changes made in the **Axes** 62 tab will only apply to the diagram for the active signal or spectrum.
- **Decimal places:** Here you can define how many decimal places the values for the X and Y axes should have.

The option **Synchronous axes** is activated as standard; 3 decimal places are displayed.

Default cursor settings

Here you can define basic cursor behaviour:

- **Synchronous cursors**: If you select this option, the measure cursor and base cursor will be displayed simultaneously for all data on the display. Any changes to the cursor position for the active data will apply to the cursor positions for all of the data.

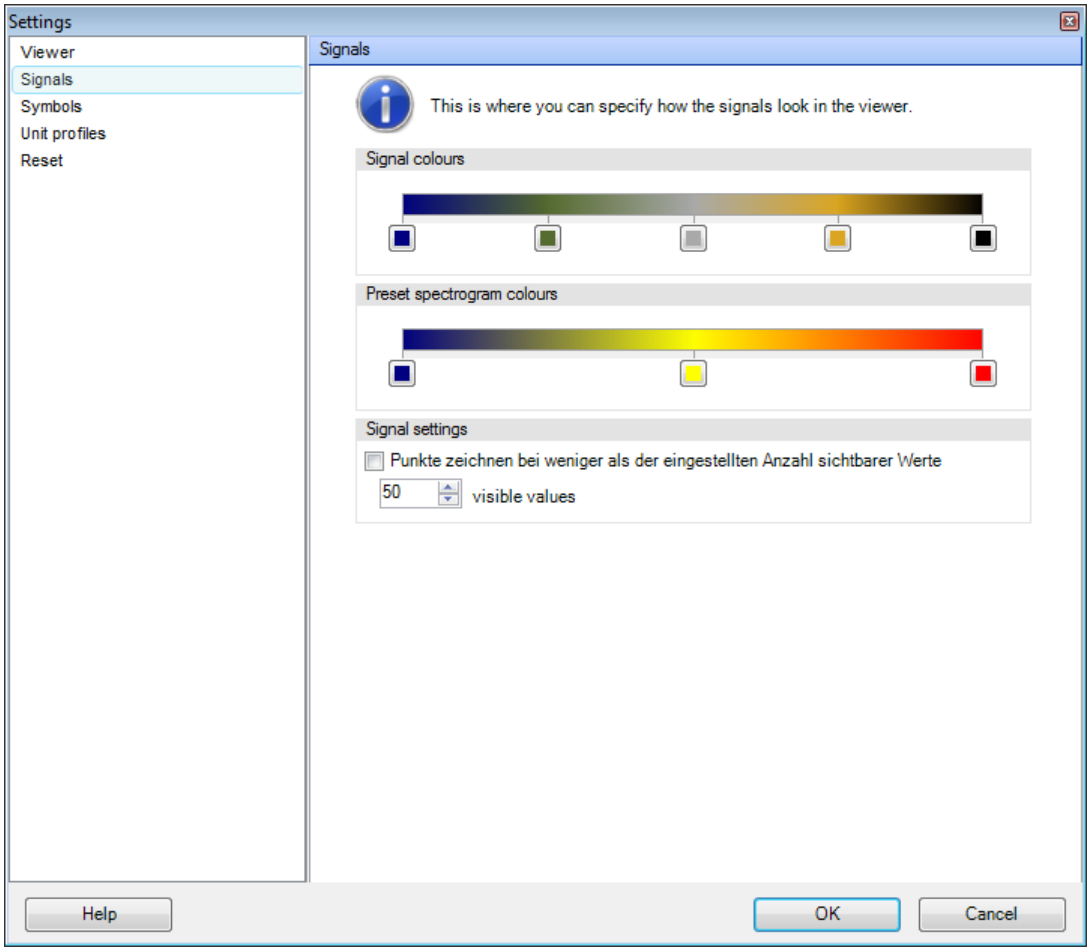
  If the checkbox is deactivated, any cursor action will apply to the active data only.
- **Draw guides**: If you activate this option, the program will not only display the cursor symbols in the diagram, but will also draw a line at the corresponding position.

  If the checkbox is deactivated, you will only see the cursor symbols without any guides.

Both options are deactivated as standard.

5.3.5.2 Signal settings

You can define the colours to be used to display signals, spectrums and spectrograms in the **Signals** area.



You have the following options:

Signal colours

Here you can define the colours to be used to display signals, spectrums and spectrograms in the viewers. Click on the corresponding colour symbol to open the

standard colour dialogue box 73, where you can specify your settings.

Preset spectrogram colours

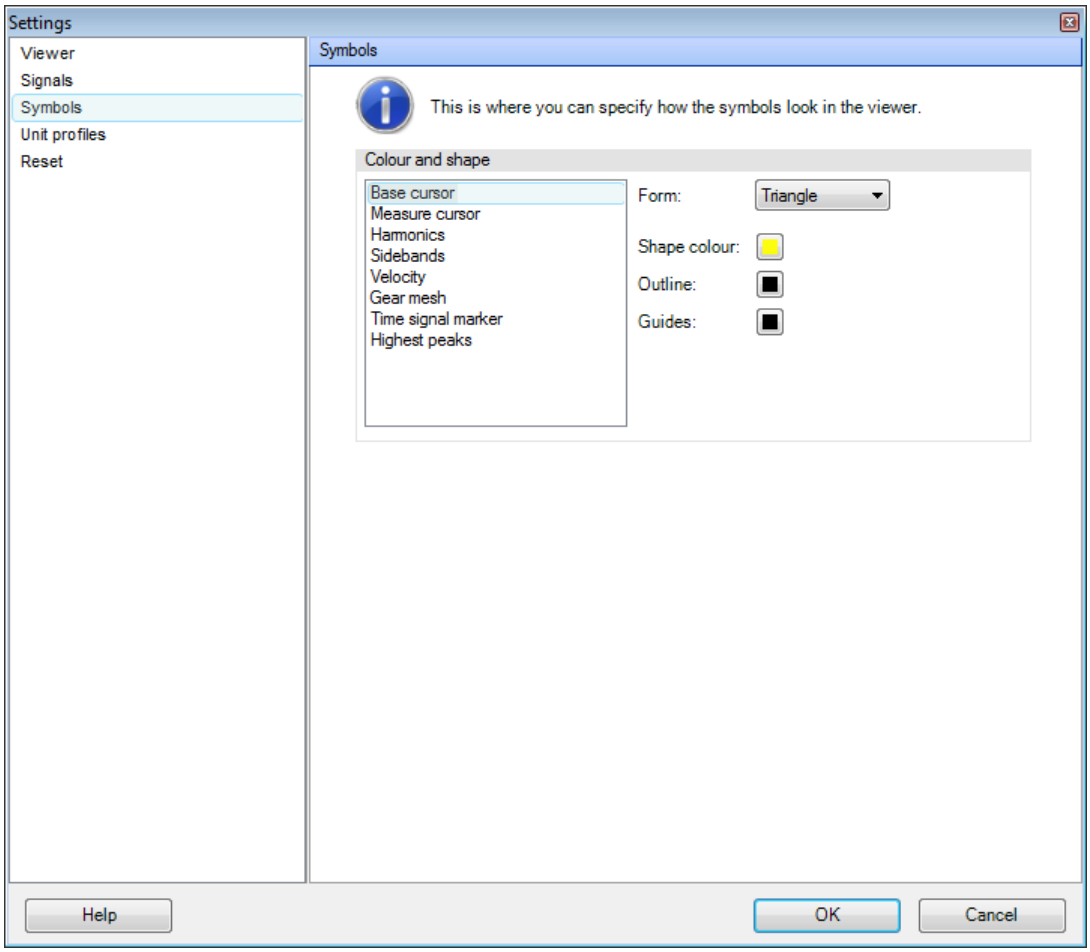
Here you can define the colours to be used for the **2D spectrogram** 61 and **3D spectrogram** 61 view options. Click on the corresponding colour symbol to open the standard colour dialogue box, where you can specify your settings.

Signal settings

- **Draw dots at less than the set number of visible values**: If this option is activated, signals are shown as dots, as soon as less than 50 values lie in the visible area of the diagram. If you deactivate the checkbox, the program will also display 50 values and fewer as a solid line. This option is activated by default.
- **Visible values**: Here you can define how many values lie in the visible area of the diagram. This are 50 values by default.

5.3.5.3 Symbol settings

You can determine how the cursor and the symbols for cursor functions should looks, as well as their standard behaviour, in the **Symbols** area.



You have the following options:

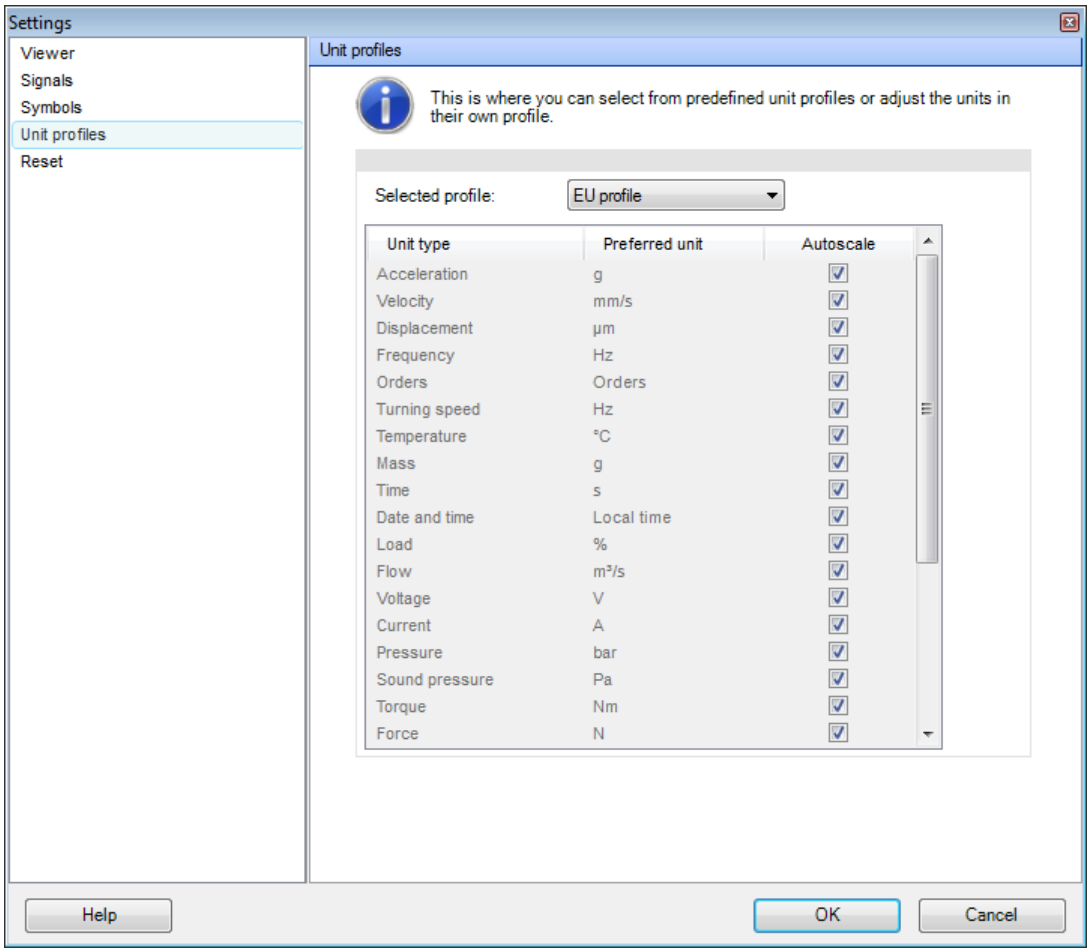
Colour and shape

You can define the shapes and colours to be used for the measure and base cursors and for cursor functions. To do this, proceed as follows:

1. Click on the symbol you want to change in the list on the left. This automatically updates the fields on the right; they show you the current settings for the symbol.
2. In the selection list labelled **Shape**, select the shape you want to use to symbolise the cursor or cursor function in the diagram, e.g. **box** or **diamond**.
3. Define the **shape colour** for the shape, as well as the colours for its **outline** and **guides**. Click on the corresponding colour symbol to open the standard colour dialogue box 73, where you can specify your settings.

5.3.5.4 Unit profiles

You can define the unit profile to be used to display the X and Y axes in the diagrams under **Unit profiles**. Unit profiles determine the unit types, units and scaling for X and Y axes.



You have the following options:

Selected profile

You have the following choice:

- **EU profile:** This profile defines the units commonly used in the European region as the **preferred unit**; **auto-scaling** 76 is activated for all unit types. You cannot edit this profile.
- **US profile:** This profile defines the units commonly used in the US region as the **preferred unit**; **auto-scaling** 76 is activated for all unit types. You cannot edit this profile.
- **Own profile:** This profile allows you to specify your own settings for the **preferred unit** as well as for **auto-scaling** 76. When you open the profile for the first time, you will be prompted to select one of the two other profiles based on the language of your operating system. You can, however, change all of the values via the profile table.

Profile table

If you have selected **EU profile** or **US profile**, this table will be used for informational purposes only, i.e. it will show you which unit is preferred for each unit type and whether auto-scaling is activated.

If you have selected **Own profile**, you will have the following options:

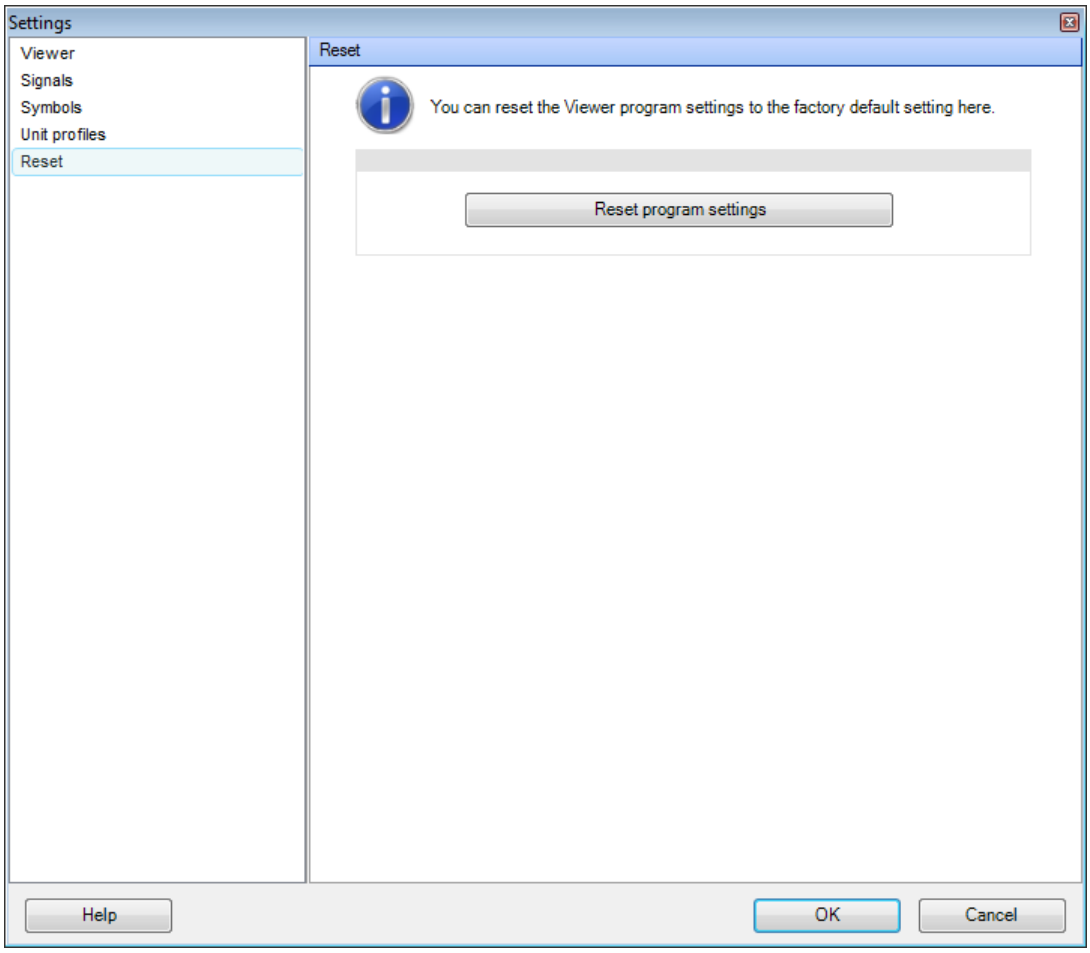
- **Preferred unit**: This option allows you to determine the unit to be used as standard for each unit type in every diagram. This unit will always be used to display the axes for this unit type.
- **Auto-scaling:** If this option is activated, the FAG SmartUtility Viewer software will automatically decide which unit is best for the diagram display, i.e. provides a clear and short display. This unit may differ from the **preferred unit** under certain

circumstances.

A list of the underlying base units for the **EU profile** and the **US profile** can be found in **Appendix II: Base units** 81.

5.3.5.5 Reset

You can reset the Viewer program settings to the factory default setting in the **Reset** area. To do so, click on the **Reset program settings** button.



5.3.6 Appendix I: working with the keyboard and mouse

You can execute many of the FAG SmartUtility Viewer software functions using hotkeys and mouse movements. You will find these functions mainly in the following areas:

- **Zoom functions** 78: Zoom in and out of diagrams in increments simply by using the keyboard or mouse.
- **Scrolling and moving** 79: Cursor positioning and movements along axes are also possible with the keyboard and mouse.
- Hotkeys and the mouse also offer support in various areas, e.g. camera control 80 in 2D and 3D view or when adjusting the Viewer 80.

You can find details on hotkeys and mouse movements in the following sections.

5.3.6.1 Zooming in a diagram

Zooming with hotkeys

+ / **-**	Zoom in on X-axis / zoom out on X-axis
SHIFT + / **SHIFT -**	Zoom in on Y-axis / zoom out on Y-axis
CTRL + / **CTRL -**	Zoom in on Z-axis / zoom out on Z-axis (3D view)
SPACE BAR	Undo all zoom steps
BACK KEY	Undo the last zoom step

Zooming with the mouse, or hotkeys and mouse

Press and hold left mouse button and drag	Zoom along the X-axis: as soon as you release the mouse button, the program zooms into the defined area*)
SHIFT + Press and hold left mouse button and drag	Zoom along the Y-axis: as soon as you release the mouse button, the program zooms into the defined area*)
CTRL + Press and hold left mouse button and drag	Zoom along the X and Y-axes: as soon as you release the mouse button, the program zooms into the defined area*)
Rotate mouse wheel forwards	Zoom into the X-axis by 10%
SHIFT + rotate mouse wheel forwards	Zoom into the Y-axis by 10%
CTRL + rotate mouse wheel forwards	Zoom into the Z-axis by 10%
SHIFT + **ALT** + left-click	Undo all zoom steps
ALT + left click	Undo the last zoom step
Rotate mouse wheel backwards	Undo the last zoom step on the X-axis
SHIFT + rotate mouse wheel backwards	Undo the last zoom step on the Y-axis
CTRL + rotate mouse wheel backwards	Undo the last zoom step on the Z-axis

*) The axis the program will zoom in on depends on the orientation of a 3D diagram:
If you are viewing the diagram from above, **SHIFT +** / **SHIFT -** will zoom in on the Z axis, not the Y axis. In general, the following applies:

- Not using the**SHIFT** key zooms the horizontal axis
- Using the **SHIFT** key zooms the vertical axis
- Using the **CTRL** key zooms a combination of the two

5.3.6.2 Scrolling and moving in a diagram

Scrolling in the diagram using hotkeys

A / **D**	Scroll the X-axis
W / **S**	Scroll the Y-axis
Q / **E**	Scroll the Z-axis

Scrolling in the diagram using the mouse

Press and hold the centre mouse button and drag the mouse in the desired direction

Using hot keys to move the cursor

ARROW LEFT / **ARROW RIGHT**	Moves the base cursor
ARROW UP / **ARROW DOWN**	Moves the measure cursor
CTRL + **ARROW LEFT** / **ARROW RIGHT**	Moves base and measure cursor simultaneously whilst maintaining the space between them
CTRL + **ARROW UP** / **ARROW DOWN**	Moves base and measure cursor simultaneously whilst maintaining the space between them
SHIFT + **ARROW LEFT** / **ARROW RIGHT**	Moves the base cursor at a faster speed
SHIFT + **ARROW UP** / **ARROW DOWN**	Moves the measure cursor at a faster speed
HOME	Positions the base cursor at the start of the signal
END	Positions the base cursor at the end of the signal
SHIFT + **HOME**	Positions the measure cursor at the start of the signal
SHIFT + **END**	Positions the measure cursor at the end of the signal

Moving the cursor with the mouse

Left-click	Sets base cursor
SHIFT + left-click	Sets measure cursor
Left-click on cursor and hold	Grabs base or measure cursor, e.g. to drag it to a different position
CTRL + left-click	Sets base cursor and moves measure cursor in relation whilst maintaining the same distance between them
CTRL + **SHIFT** + left-click	Sets measure cursor and moves base cursor in relation whilst maintaining the same distance between them
CTRL + left-click on cursor and hold	Grabs base and measure cursor simultaneously to drag them to a different position

5.3.6.3 Other functions

Viewer and diagram

F11	Switch full screen mode on/off You can also switch off full screen mode with **ESC**.
CTRL + **C**	Copy diagram
CTRL + **F**	Save diagram
CTRL + **I**	Show/hide the viewer's information bar

Signal display

INPUT KEY	Change between showing the active signals and all signals
IMAGE UP/IMAGE DOWN	Change between the loaded signals

Select cursor functions

F2	Select cursor function **Basis analysis**
F3	Select cursor function **Gear mesh**
F5	Select cursor function **Harmonics**
F6	Select cursor function **Sidebands**
F7	Select cursor function **Harmonics with sidebands**
F8	Select cursor function **Speed**
F10	Open **Cursor settings** dialogue box

Select option for cursor positioning

SHIFT + **F2**	Select **Free** positioning option
SHIFT + **F3**	Select **Next value** positioning option
SHIFT + **F4**	Select **Nearest peak** positioning option
SHIFT + **F5**	Select **Tenths** positioning option
SHIFT + **F6**	Select **Hundredths** positioning option

Scaling

CTRL + **Z**	Reset range scaling 21

Camera functions in 3D spectrogram, waterfall and wireframe views

CTRL + **ALT** + press and hold middle mouse button	Rotate the diagram around the X and Y-axis
CTRL + **ALT** + turn mouse wheel	Zoom in/out of the diagram

5.3.7 Appendix II: basic units

Basic units in the EU profile and US profile

Unit type	Basic unit EU profile	Basic unit US profile	Autoscale
Acceleration	m/s²	in/s²	Yes
Speed	mm/s	in/s	Yes
Displacement	µm	mil	Yes
Frequency	Hz	Hz	Yes
Speed	Hz	Hz	Yes
Temperature	°C	°F	Yes
Mass	g	oz	Yes
Time	s	s	Yes
Date and time	Local time	Local time	Yes
Load	%	%	Yes
Flow	m³/s	in³/h	Yes
Voltage	V	V	Yes
Current rating	A	A	Yes
Pressure	bar	bar	Yes
Sound pressure	Pa	Pa	Yes
Torque	Nm	lbf in	Yes
Force	N	N	Yes
Power	W	W	Yes
Band speed	m/min	in/s	Yes
Unknown	-	-	Yes
Phase	°	°	Yes
Counter	Quantity	Quantity	Yes
Expansion	µEpsilon	µEpsilon	Yes

5.4 Create report

You use this wizard to create reports from the downloaded SmartCheck device data. To do this, the SmartUtility software combines the selected device data with a selected report template in Rich Text Format (RTF). The RTF report templates are included in the scope of delivery for all program languages. You can Edit templates 96 and under **More actions > Manage report templates**, you can Change settings for reports 94.

In every step of this wizard, you will see the **Finish** button. Click this button to create the report from your current step and go directly to the results list in **step 4**.

If you have started the wizard for the first time, the report will be created with the program's default settings:

- **Selected devices (step 1)**: All available devices in the data directory
- **Report template (step 2)**: The current default report template
- **Report time range (step 3)**: The previous three months up to the current day
- **Save options (step 3)** : Summarise reports for all devices in one report

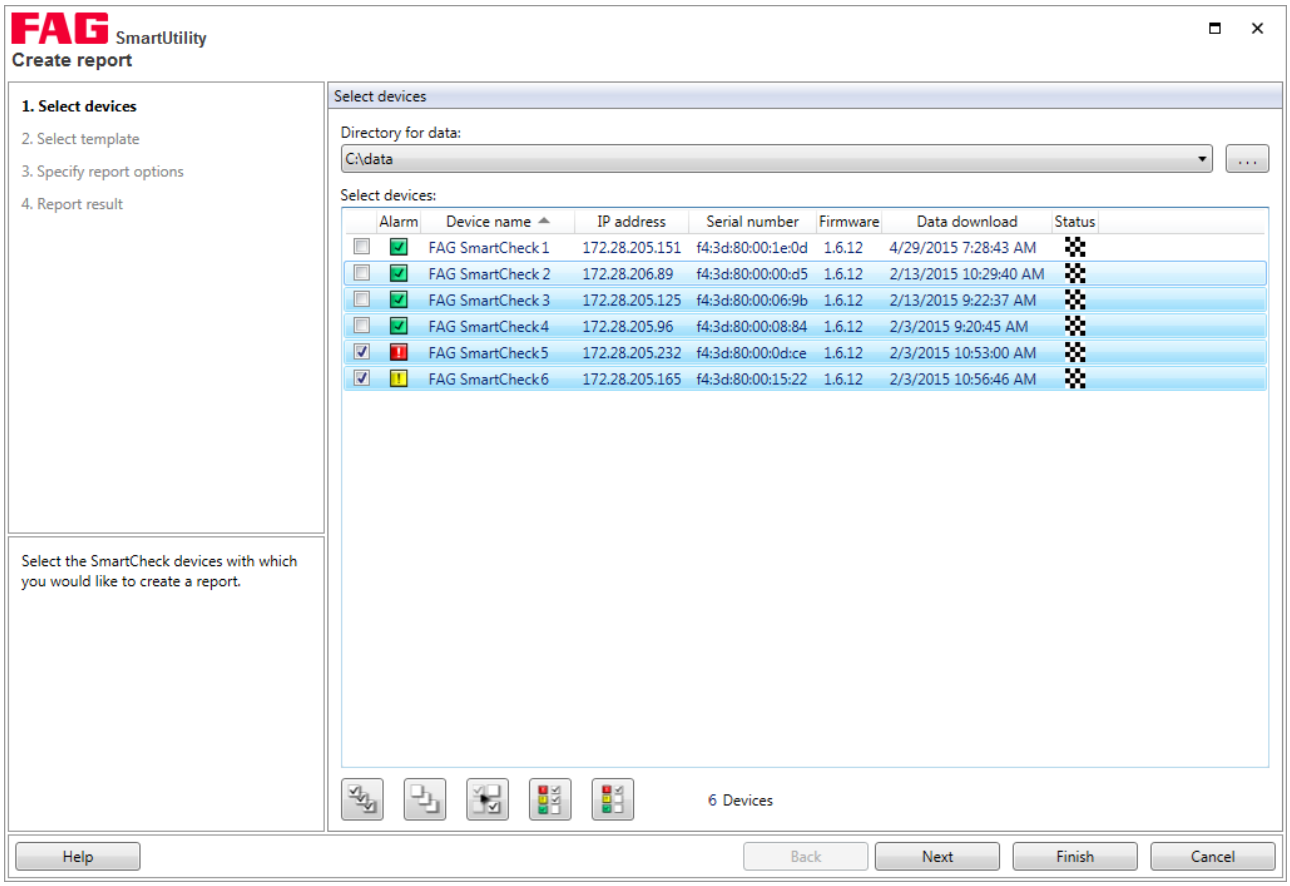
You can also use the **Finish** button as a short cut using your own settings:

- When you start the wizard for the first time, set the individual steps as desired.
- The next time you start the wizard, these settings will be applied automatically when you click **Finish**.

To create a report, proceed as follows:

Step 1:

1. Select the data directory containing the data for the report. This may be the default data directory 108 or a directory you have selected yourself.
2. The SmartCheck devices for which data is available will then be listed below. Here, you can select one or multiple devices:



In the list of devices, you will find detailed information about each device, such as the alarm status, the device name, or the date the data was downloaded. In the **Status** column, you will see data status symbols. When you move the mouse over it, you will see information about the status of the data. 19

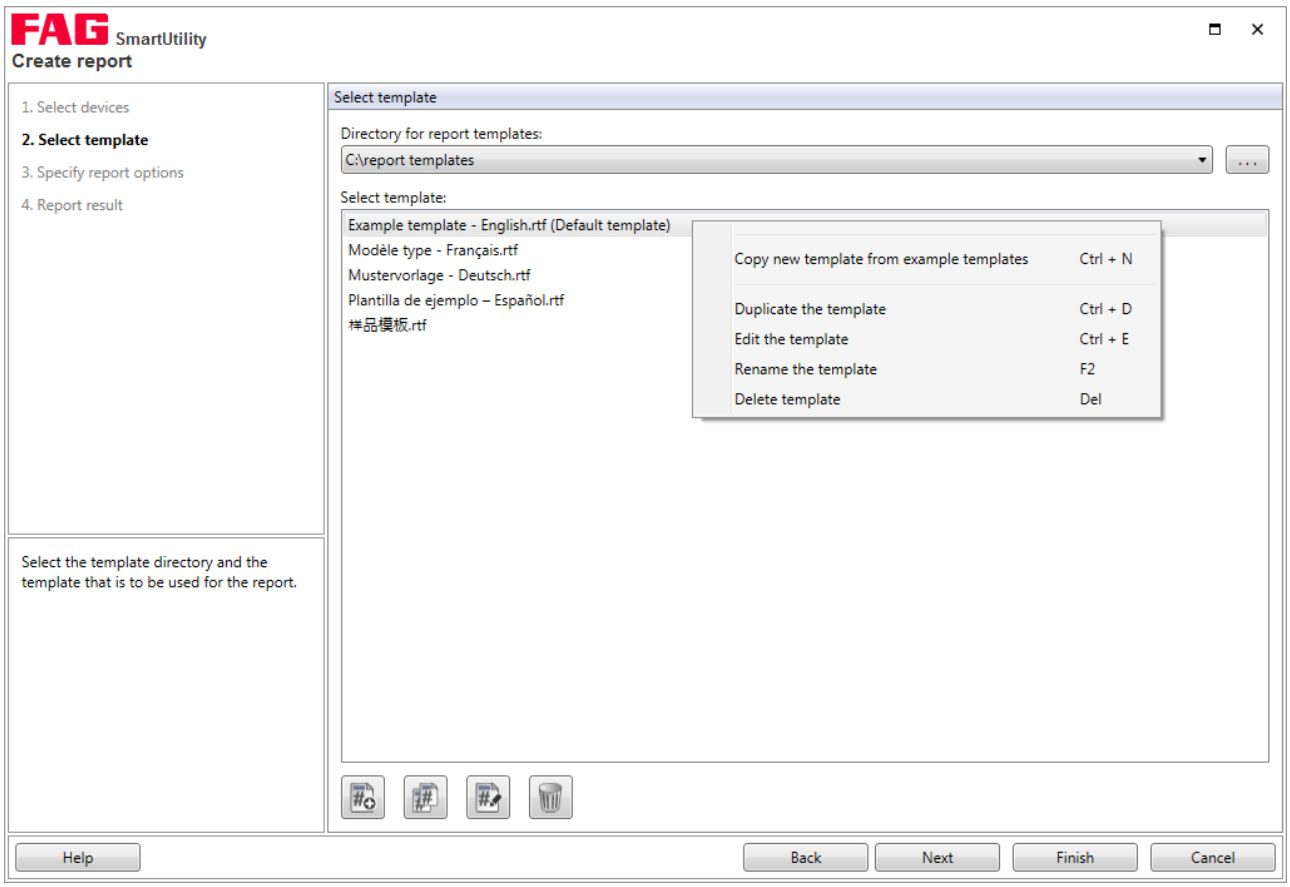
The  symbol indicates that the downloaded data has not been converted. A report cannot be created from data in an out-dated format.

Step 2:

Select the directory containing the report templates. This may be the default directory 95 or a directory you have selected yourself.

The available report templates will then be listed below. The current default template will be preselected. Immediately after installation, this is the sample template in the current program language. You use the function **More actions > Manage report templates** to Change the standard template 100 as well as the template directory.

You can also use the buttons and context menu to create new templates 96, duplicate templates 98, edit templates 97, rename templates 99 or delete templates:

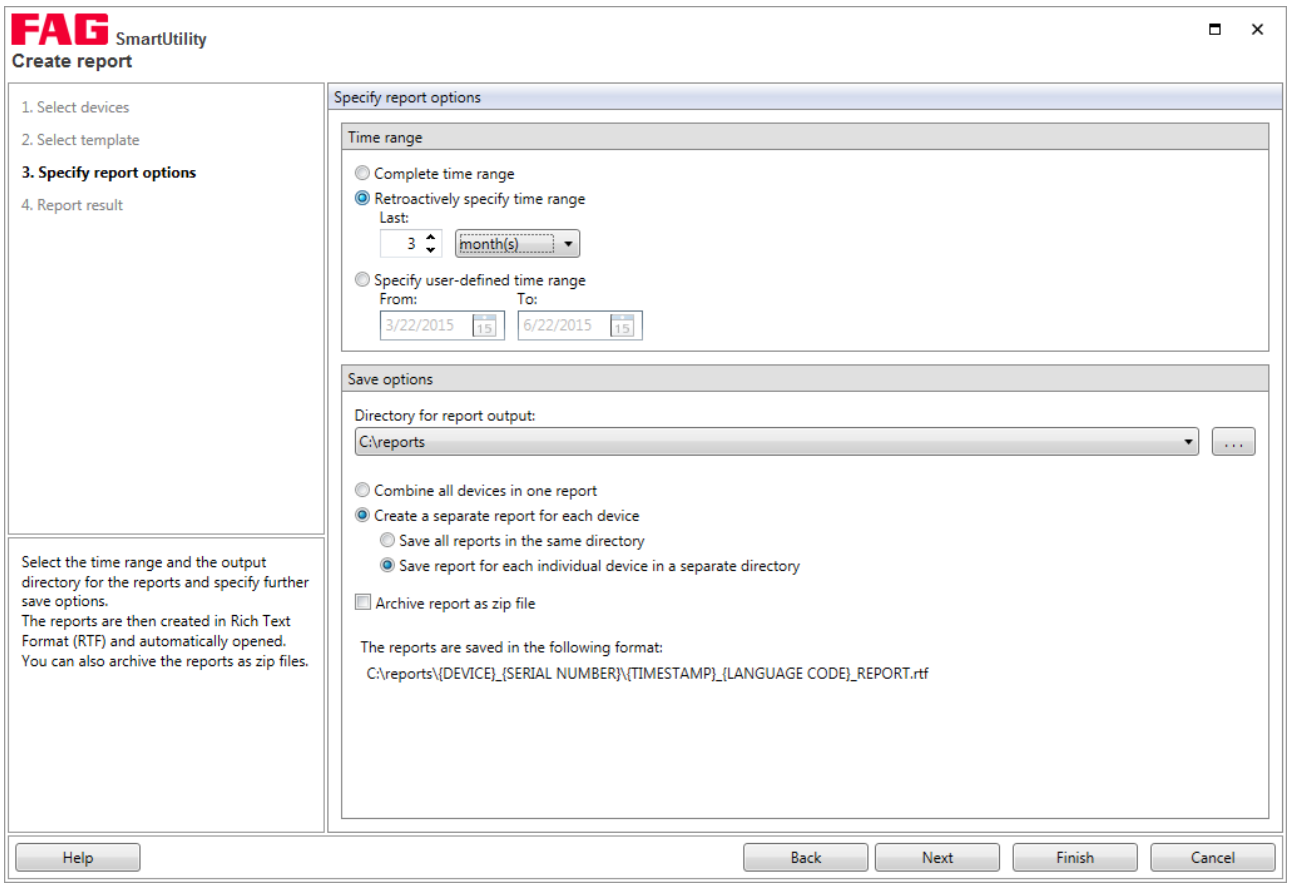


You must install the corresponding Windows language pack for the language in which you want to create a report. If foreign-language entries in the **Logbook** section are incorrect or feature invalid characters, you must also install the Windows language pack for the language of the logbook entries. Note that some language packs are only available for the Professional or Ultimate versions of the Windows operating system.

Future versions of SmartUtility will contain new sample templates if required. If you want to create a new template or edit an existing one, you will be informed about new sample templates. You can stop receiving these messages with the option **Do not show this message again**. After reinstallation, the message will appear again.

Step 3:

In the third step, you must specify the time range for which the report is to be created. You also specify here where and in which format the result is saved:



Here you have the following options:

Time range

This is where you determine the time range for which the report is to be created:

- **Complete time range:** The report is created from all measurement data available in the selected directory.
- **Retroactively specify time range:** If you activate this option, you can set the number of months, weeks or dates up to the current date. Only the measurement data for this time range is then used for the report.
  If 3 months is preset, the report will then include measurement data for the previous three months up to the current date.
- **Set user-defined time range:** If you activate this option, you can enter dates directly in the **From** and **To** fields for the desired time range or select the dates from the calendar. Only the measurement data from the selected time range is used for the report.

Save options

Specify the location and format in which the report is saved. You have the following options:

- **Directory for report output**:
  The list is preset to display the default directory to which the report is saved. You specify this directory when you first open the software, or later under **More actions > Manage report templates** 95.
  Click ... to select a different directory as the storage location.
- **Combine all devices in one report**:
  Select this option to summarise the report information for all selected devices in a single RTF document. Each device will have its own section in this RTF document.
- **Create a separate report for each device:**
  Select this option to create a separate RTF document for each device.
  With both sub-operations, you specify whether the individual documents will be saved in one subdirectory or in separate subdirectories.
- **Archive reports as zip file:**
  Activate this option to add the created reports to a ZIP archive. The compressed reports can then be sent by email for example.

The ZIP archive saves the entire path including all subdirectories as set up on the hard drive.

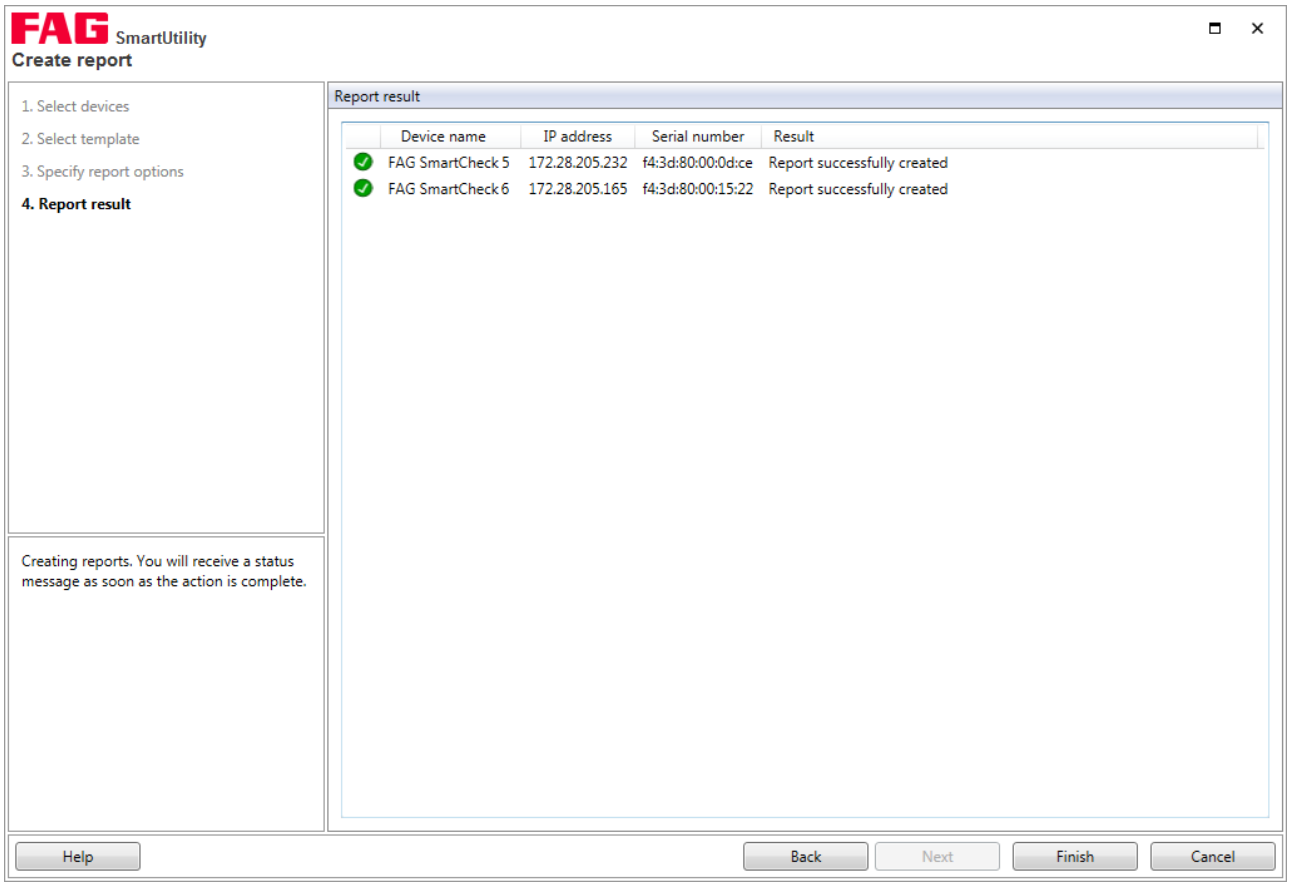
- **Directory path:**
  This path corresponds to your settings and is updated immediately each time the settings are changed.
  The terms time stamp, device (for individual reports) and language are simply used here as placeholders and will be replaced by the actual values.

If you have not changed the default name of your SmartCheck device, all devices have the same name. In this case, the appropriate serial number according to the selected save format will be integrated into the file or subdirectory names.

Devices with the same name that are to be summarised in a single report cannot be differentiated from each other by name in the report. In this case, we recommend giving each device its own name 87.

Step 4:

Once **step 3** is completed, the reports will be created in accordance with your settings. In **step 4**, you will see the created reports:



The symbol at the beginning of each line shows you the status of the report at a glance. In the **Results** column, you will find more detailed information about the creation of the report or problems encountered during creation. You can find the following information here:

Symbol	Result	Explanation
	Report successfully created	There were no problems when creating the report. Click **Finish** to open the report.
	An error occurred when accessing the template. The file may still be open in another program.	If the selected report template is still open in an editor, the report cannot be created. Close the report template and try again.
	You may not have permission to write to the directory <name>! Please check your details!	You need write access for the directory to which the report is to be saved. Check your permissions for the directory selected in **step 3**.
	The file used is not a valid report template. Please repair or replace the report template.	Valid report templates must be in RTF format and can only contain predefined tags 100. The SmartUtility software provides you with a sample template for each program language. You can use these as a basis to create and edit your own report templates 96.
	The report template does not contain any tags that can be expanded with data. Please repair or replace the report template.	Report templates can only be populated with data if they contain predefined tags. The SmartUtility software offers a wide selection of different tags 100 that can be used to populate the report with device data, measurement time ranges and creation data.
	SQLite database file not found	This message indicates that the device data is not in an up-to-date format. A report cannot be created from data in an out-dated format. Use the function **More actions > Rescan data directory** 104 to convert the data.

Result:

Click **Finish** to view the created reports. This step depends on your settings in **step 3**:

- **All reports in one file**: The file is immediately opened in the default program for RTF files, such as MS Word.
- **An separate report file for each SmartCheck device**: The directory containing the individual files is opened.
- **A separate subdirectory for each individual report**: The parent directory containing all the report subdirectories is opened.
- **ZIP archive**: The content of the ZIP archive is displayed.

When you open the completed report in MS Word, please note the following:

- Confirm where necessary that the file opens in RTF format.
- The table of contents must be updated manually. To do this, right-click in the table of contents and select **Update Field > Update entire table**.

5.5 Edit device settings

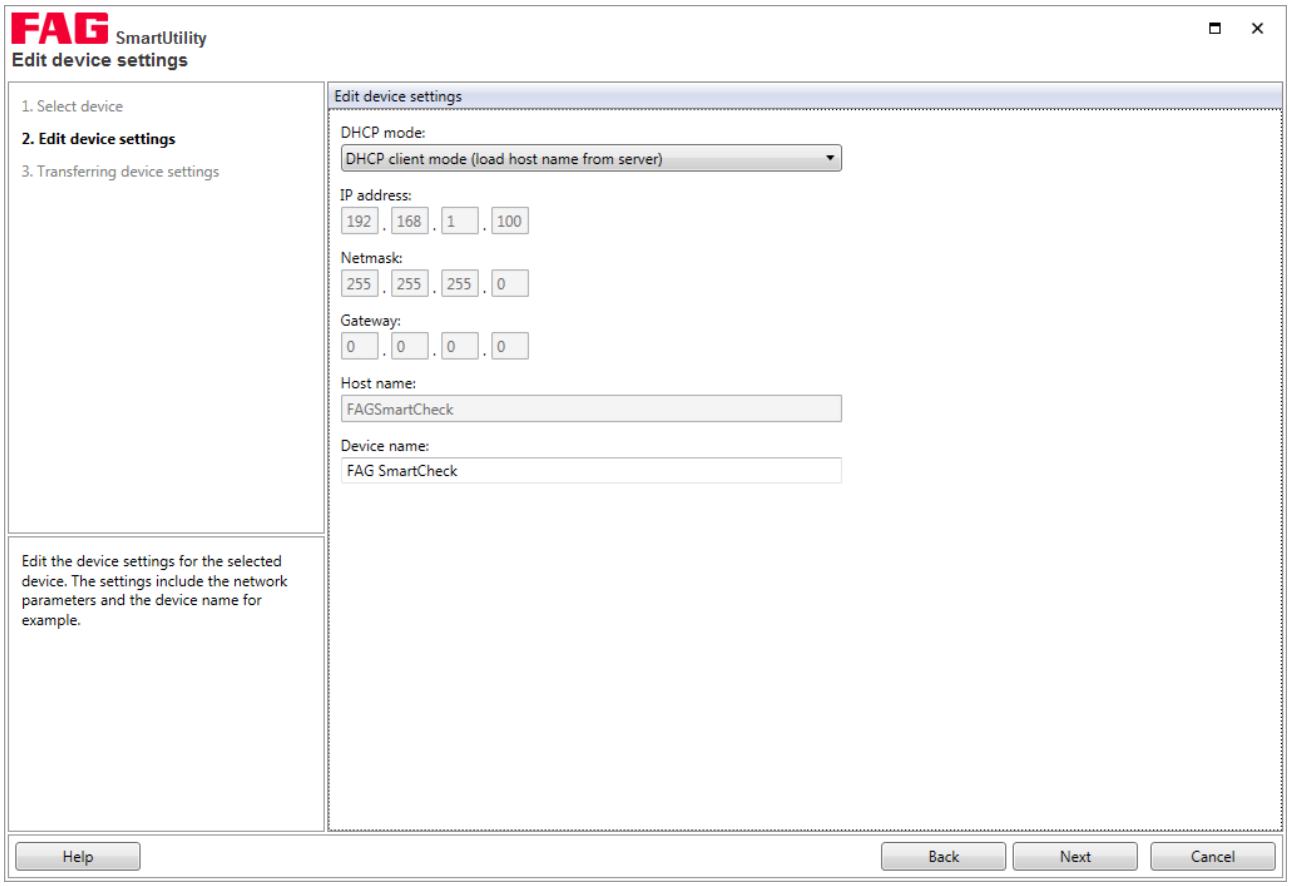
With this wizard, you can specify the DHCP mode and associated settings for the SmartCheck device. Proceed as follows:

Step 1:

Select the required SmartCheck device for which you want to specify the settings.

Step 2:

In the second step, you can specify the required settings and send them to the SmartCheck device.



Here you have the following options:

No DHCP	With this option, you can specify the IP address or continue to use the SmartCheck device's default IP address. If **No DHCP** is activated, you must also specify the other settings in this step, e.g. **IP address**, **netmask** or **host name**.
DHCP client mode (load host name from server)	With this option, the SmartCheck device automatically receives an IP address within your network. The device name is specified via the network's DNS server (reverse DNS).
DHCP client mode (send host name to server)	With this option, the SmartCheck device automatically receives an IP address within your network. The device name is registered by the SmartCheck device in the network's DNS server.
Device name	Here you can alter the **device name** of the SmartCheck device. Give each device a unique name, so that you can find it again straight away in the list boxes.

- If the IP address is allocated automatically via DHCP, the SmartCheck device can only be accessed via the automatically allocated IP address. You can no longer use the default IP address.
- The name of every SmartCheck device is **FAG** SmartCheck by default. If you want to integrate multiple SmartCheck devices into your system, it is important that you give each device a unique name. Otherwise, you can only identify the devices in the wizard lists via the IP address or the serial number.
- If user management is activated on the SmartCheck device, you must also enter a username and password in SmartUtility. If these are not entered, the device settings will not be transferred.

Result:

In the third step, you can check the result of the action.

5.6 Downloading the configuration

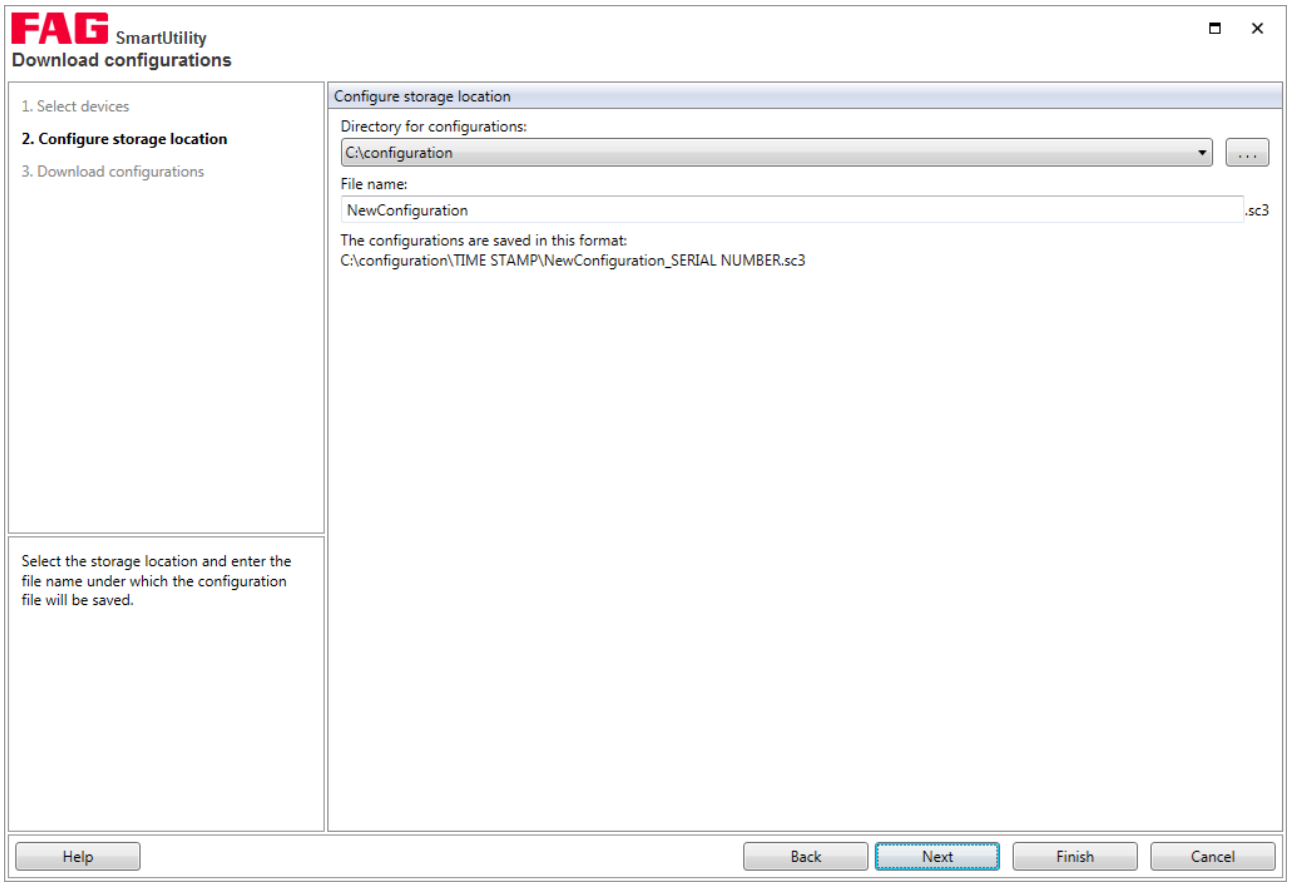
With this wizard, you can download the configuration of the measurement jobs of one or multiple SmartCheck devices as files, for example to send them to other SmartCheck devices or to back them up before a firmware update. Proceed as follows:

Step 1:

Select the required SmartCheck device or devices.

Step 2:

The second step is to specify the directory and file name under which the configuration files, i.e. the files with the measurement jobs, are saved:



Here you have the following options:

Directory for configurations	Here you determine the directory in which the configurations are to be saved. You have the following options: • The list shows the default directory for configurations by default. This directory is specified either when you first launch the software or subsequently under **More actions > Settings > General** 108. • Click … to select another directory and specify this as the save location.
File name	Using valid characters, you can determine the file name here, which is to receive the downloaded configurations. The names of the individual files will then be supplemented with the serial number of the appropriate device. Invalid characters for the file name are: < > ? " : | / \ * If you use these characters, they will be replaced by an underscore.

The path generated from your settings for **Directory** and **File name** is indicated to you beneath the settings as an example. This is automatically amended with each additional change.

Step 3:

The file or, in the case of several selected devices, the files, is/are downloaded. Wait until the process is completed.

Result:

The downloaded configurations are located in the directory you specified in **Step 2**. The file name specified by you is supplemented by the serial number of the appropriate SmartCheck devices.

If you download configurations from an SmartCheck device with firmware version 1.4 (or older), the configurations are automatically converted into the new file format for SmartUtility software 1.6. You can then no longer send these converted configurations to SmartCheck devices with the old firmware version!

5.7 Sending the configuration

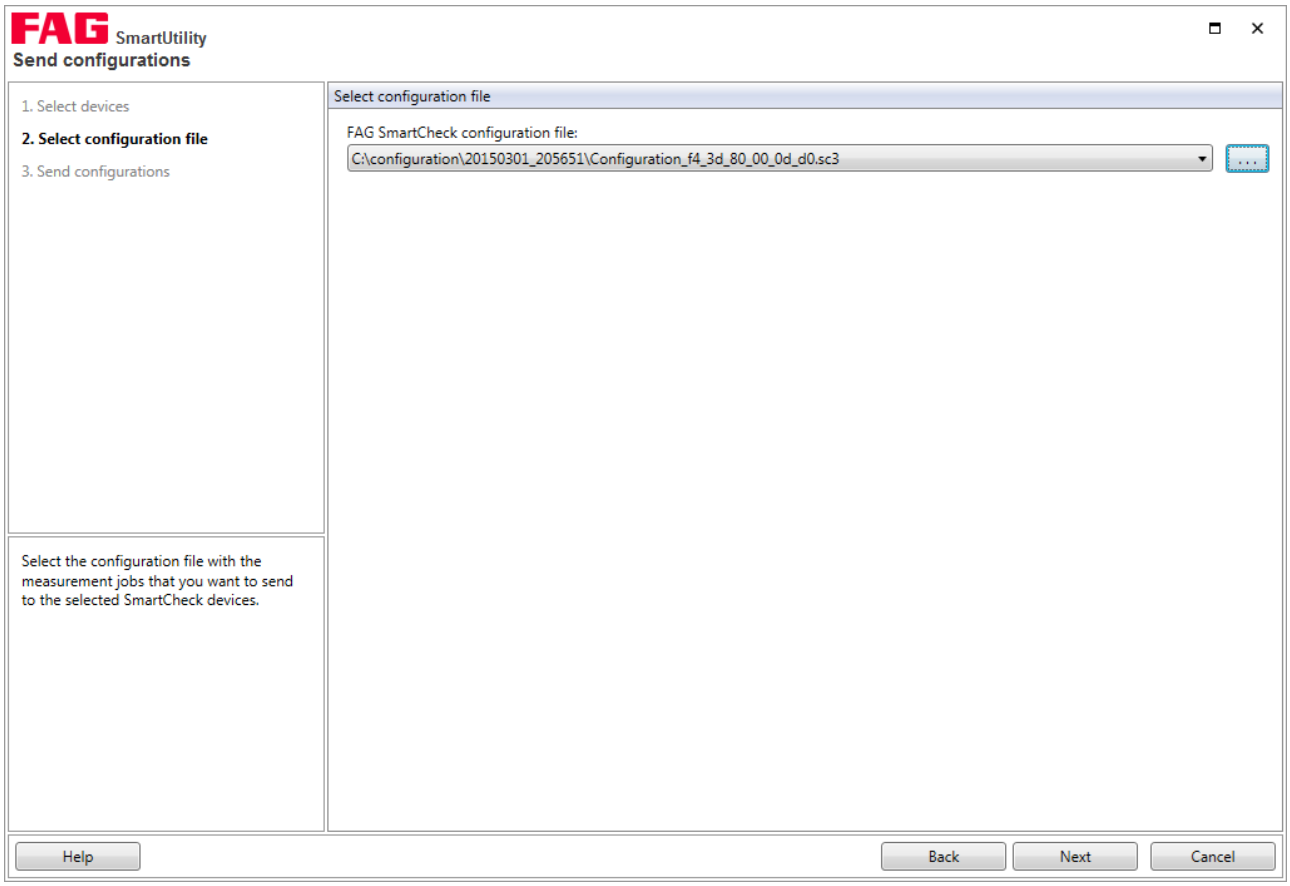
With this wizard, you can send the configuration of the measurement jobs as a file to one SmartCheck device or several devices. Proceed as follows:

Step 1:

Select the required SmartCheck device or devices to which you want to send the configuration file with the measurement jobs.

Step 2:

The second step is to select the file (file ending SC1, SC2 or SC3) you want to send.



Step 3:

The file with the measurement jobs is sent to the selected SmartCheck devices. Wait until the process is completed.

Result:

The measurement jobs saved in the selected file are now on all SmartCheck devices to which you sent the file.

- After you have sent the new measurement job configuration to the SmartCheck device, it takes a while before the new alarm status for the devices is displayed in the device overview.
- If you had already downloaded data for analysis for the old configuration, the new measurement job configuration also has an effect on the analysis in the Viewer: As soon as you download data from the new configuration and open it for analysis, FAG SmartUtility Viewer creates a new measurement job. The old and new measurement jobs are displayed one underneath the other for the respective SmartCheck device.
- If a downloaded configuration contains an alarm status configuration for an external device (e.g. a controller), this alarm status configuration is by default not included in the transfer. This prevents multiple FAG SmartCheck devices from writing to the same registers for the external controller. Thus, the alarm status configuration can be sent, please activate the check mark in front of "Send external controller outputs to SmartCheck device".
- If you are using an older version of SmartWeb, you should first back up its configurations with the **Download configuration** wizard. You can then update SmartWeb to the SmartUtility version. The backed-up configurations can subsequently be installed again using the **Send configuration** wizard.
- If you want to send a configuration (from version 1.6.10 onwards) to an SmartCheck device with an older or the same version (e.g. 1.6.6), you may need to change the basic measurement job used in the newer version to a standard measurement job so that the configuration can be sent.
- If a downloaded configuration contains external sensors that are connected via both of the analogue inputs or via the digital input, the name of the input channel is extended with the code "_ext" and a number if necessary. This naming format ensures that the external input channels are clearly marked.

5.8 Updating the firmware

With this wizard, you can send a file with a firmware update to one or more SmartCheck devices. More detailed information on firmware updates and the corresponding notifications is set out in your service or maintenance contract.

CAUTION

Measurement data and configurations may be irretrievably deleted!

If you update the firmware of a FAG SmartCheck device, depending on the update version, you may lose all the measurement data and configurations saved on the device. Note the following:

- When changing from version 1.2 to a higher version, all measurement data and configurations are lost.
- When changing from version 1.4 or 1.6 to a higher version, all measurement data is lost.
- From versions 1.4.27 and 1.6.6, configurations are usually retained.
- Measurement data and configurations are not retained until version 1.10

Before updating firmware, download the measurement data with the SmartUtility software if required. In addition, you can download the configuration (with the taught alarm limits) for the SmartCheck device via the SmartUtility software and install it again after the firmware update.

If the configurations and alarm limits must not be deleted during a firmware update, use the SmartUtility software.

Values in alarm maps with a completed learning mode may be lost!

If you have started the learning mode for the SmartCheck device in conjunction with one or two other signals, the associated alarm maps are populated gradually during the learning process. A firmware update has the following effects:

- All alarm maps are reset to the **Use learning mode** status, regardless of whether the learning mode was already completed in a map.
- Learning mode is deactivated and the outstanding maps remain unchanged.
- If you re-activate learning mode, it restarts for all maps. You also lose the values that have already been taught-in.

To back up the values in the alarm maps with a completed learning mode, proceed as follows:

1. Manually deactivate the **Use learning mode** option for each completed alarm map. You can find this option in the configuration wizards for the SmartWeb software.
2. Activate the learning mode only at this point.

For more information about the learning mode, please see the section entitled **Learning mode and alarm maps** in the manual for the SmartWeb software.

The update to version 1.10 is also associated with a migration. Due to the migration, you cannot downgrade from 1.10 to a previous version. All future firmware versions will also be based on this update with migration. This means, for example, that you can no longer update from version 1.6.20 directly to a future version 1.12. You must always install version 1.10 first.

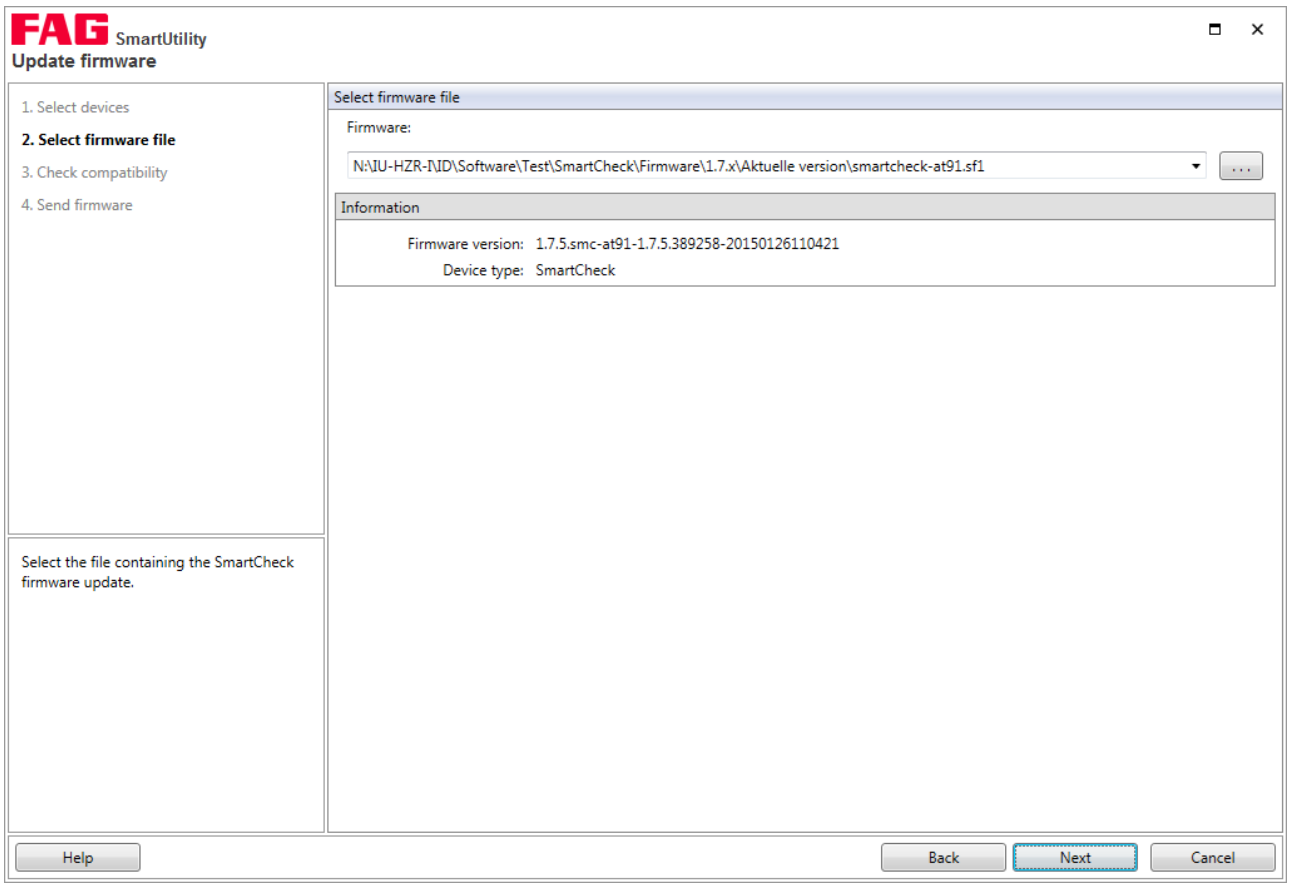
To send a firmware update, proceed as follows:

Step 1:

Select the required SmartCheck device or devices whose firmware you want to update.

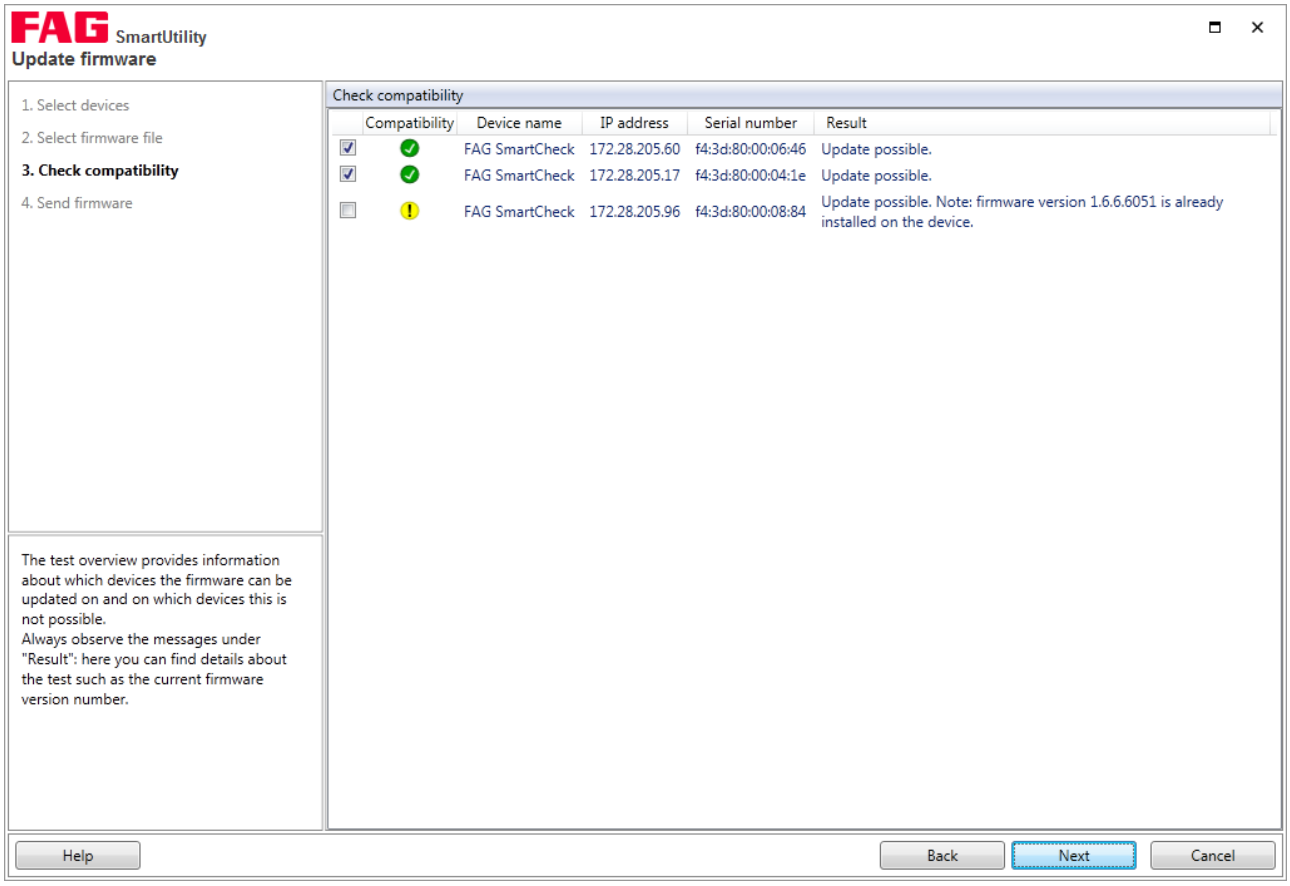
Step 2:

Specify the file with the firmware update; the file has the ending **SF1**. Once you have selected a file, you will find the version number and the device type for this firmware file in the **Information** section.



Step 3:

This step checks which firmware is present on the selected devices. You are then presented with an overview of the verification result:



You can find the following information here:

	This symbol indicates devices whose firmware can be updated with the selected firmware version. These devices are already ticked. The following requirement is needed for this status: • The firmware version of the device is lower than the selected firmware version. Details on this can be found in the **Result** column.
	This symbol indicates devices whose firmware can be updated with the selected firmware version. The checkmark is not yet activated for this status as the update could have undesired effects under certain circumstances. For this status you will receive a detailed explanation of the verification result. You should not activate the checkmark until you have read this explanation and have been notified of the possible effects of the update. • The firmware version of the device is identical to the selected firmware version. • The firmware version of the device could not be read and it is not certain whether it is compatible with the selected firmware version. If you perform the update, it is possible that the device will reset to the factory firmware version. • For updates prior to version 1.10: The firmware version of the device could not be read but the device version is compatible with the selected firmware version. It is possible that you may not be able to re-use the device configurations following the update. • The firmware version of the device is higher than the selected firmware version. The update will downgrade the device firmware version to the lower version. The configurations will be lost in this case, as only upwards compatibility is guaranteed. Details on this can be found in the **Result** column.
	This symbol indicates devices whose firmware cannot be updated with the selected firmware version. You cannot tick these devices. The update may be blocked for the following reasons: • The selected firmware version is lower than the factory firmware for the device. • The selected firmware version is not compatible with the device version. • The selected firmware version is unknown. • The version of SmartUtility is too low. • The firmware version of the device is too low and cannot be updated in a single step. You must first update the device with a lower firmware version. • The firmware version of the device is version 1.10 or above. In this case, it is no longer possible to import a lower version. • A communication error occurred. • Device login failed. You may have activated user management in SmartWeb. Details on this can be found in the **Result** column.
Device name, IP address, Serial number	This information identifies the SmartCheck device for the line in question.
Result	Here you can find detailed information on the verification result. This information tells you why the firmware can - or cannot - be updated.

Step 4:

Click **Next** to send the firmware to the selected SmartCheck devices and wait until the process is fully completed.

The factory firmware is the firmware originally supplied with the device. The version of the new firmware cannot be lower than the factory firmware version; updating in this case will not be possible. The system reverts to the factory firmware if required, e.g. if the update fails.

Result:

After the firmware is updated, the FAG SmartCheck device is unavailable for a time, as a safety mechanism is being run. This mechanism ensures that your device is functioning properly again after the firmware update. The duration of

unavailability depends on the result of this check:

- If your device with the new firmware is functioning properly, the SmartCheck device will be available again after approx. 6–7 minutes.
- If this is not the case, the old firmware will automatically be reinstalled. It will then take approx. 20 minutes until the device is available again.

For updates to firmware versions before version 1.10, please empty the browser cache after the update. This is necessary to ensure that the latest version of the FAG SmartWeb software is loaded in your browser.

Checking the firmware version on the SmartCheck device

You can see which version of the firmware is set up on your FAG SmartWeb device in the SmartCheck software. To do this, proceed as follows:

1. With the **Open devices** 15 wizard, open the required SmartCheck device or the corresponding SmartWeb software. Alternatively, you can also enter the IP address of the SmartCheck device in a browser.
2. In the SmartWeb software menu bar, click on **Help**.
3. Select **Version**.

Here you can find detailed information on the version of the SmartCheck device, including the device ID and serial number.

5.9 Opening the log file directory

Click on **More actions > Open log file directory** to open the directory in which SmartUtility deposits the log files. Depending on the Windows version installed on your system, this directory is located by default under:

- Windows 7:

C:\Users\[User name]\AppData\Roaming\Condition Monitoring

All software system messages and information on processes are recorded in the log files. You can open a log file with a text editor or word processor.

5.10 Opening the default directories

Click on **More actions > Open default directory for data**/**Open default directory for configurations**/**Open default directory for report output** to open the directory in which the SmartUtility software saves/searches for downloaded data/configuration files. Depending on the Windows version installed on your system, these directories are located by default under:

- Windows 7:
  C:\data
  C:\configuration
  C:\reports

You can change the default directories in **Settings > General** 108 or **Settings > Report** 112**.**

Make sure that the downloaded data is always saved to the default directory for data. If you want to specify your own directory, make sure that all the data is available in this directory. The analysis of downloaded data in the Viewer software is only useful when a trend for all the collected data can be created. To this end, it is necessary that all data is collected in the same directory.

5.11 Manage report templates

You can use the SmartUtility software to create reports in RTF format 81 from the downloaded SmartCheck device data. The templates for these reports can be managed in the following locations within the SmartUtility software:

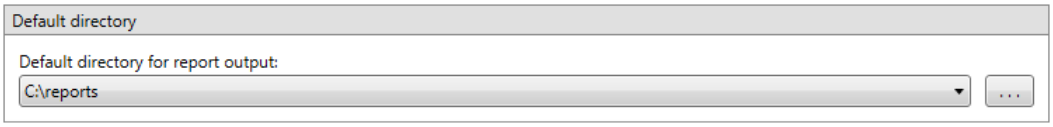
- In the menu under **More actions > Manage templates**
- In the menu under **More actions > Settings > Report**

Default directory

Here, you can change the directory to which the finished reports are saved by default. The directory is automatically created at the following path at installation:

C:\reports

Click ... to search for a new directory and set it as the default directory for reports. You can now find directories that you have previously selected in the list box.



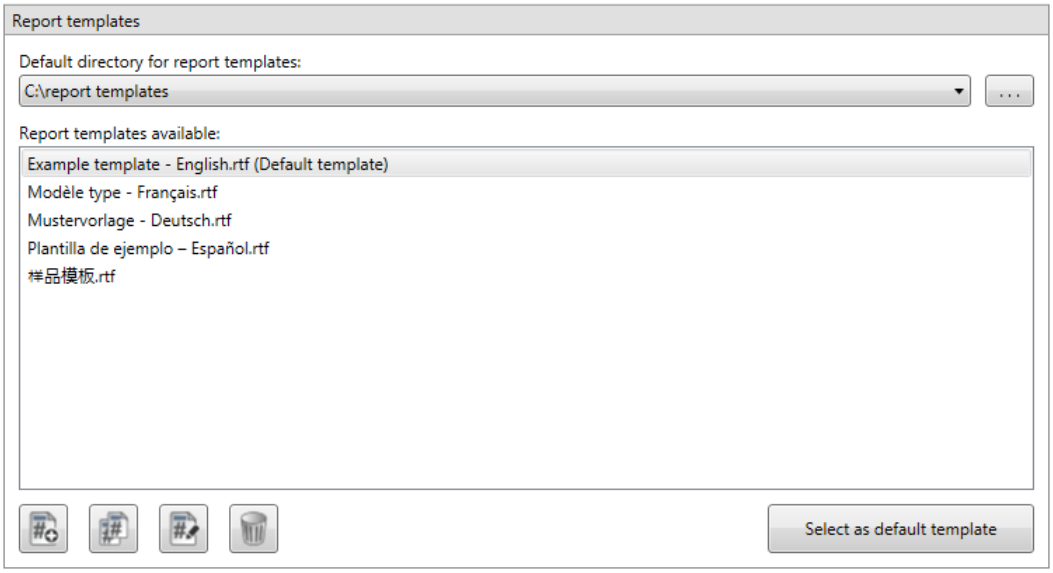
Report templates

Here, you can change the directory to which your report templates are to be saved by default. The directory is automatically created at the following path at installation:

C:\report templates

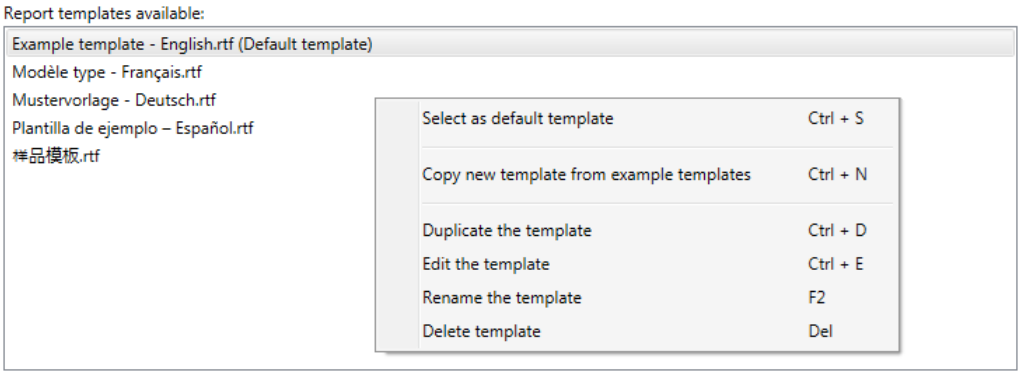
Click ... to search for a new directory and set it as a default directory for your report templates. You can now find directories that you have previously selected in the list box.

Here, you can also manage the available report templates, create new templates and set a template as the default template:



Here you have the following options:

	Here, you can copy a new template from the sample templates 96.
	Here, you can duplicate selected templates 98 e.g. to edit the copy.
	Here, you can edit selected templates 97 and modify them to meet your requirements.
	Here, you can delete the selected templates after confirmation.
Select as default template	Here, you can define the selected template as the default template 96.
Rename the template	This option is only available in the context menu (see below). Click this option to open a dedicated dialogue to change the template name 99.
Context menu	Right-click the selected template to open the context menu to access the options described above: **Copy new template from sample template**, **Duplicate template**, **Edit template**, **Rename template** and **Delete template**.



The following hotkeys can also be used for the options described above:

Ctrl+S	Select as default template
Ctrl+N	Copy new template from sample template
Ctrl+D	Duplicate template
Ctrl+E	Edit template
F2	Rename template
DEL	Delete template

5.11.1 Creating and editing report templates

The SmartUtility software creates reports on the basis of templates in RTF format. A sample template in every program language is included in the scope of delivery. By default this can be found in the **C:\report templates** directory. Using an editor and these sample templates as a basis, you can create new templates 96 and directly edit all existing templates 97 or first create a copy 98. You can also rename a template 99 and define an existing template as the default template 100. Details about these actions can be found in the following sections.

Future versions of SmartUtility will contain new sample templates if required. If you want to create a new template or edit an existing one, you will be informed about new sample templates. You can stop receiving these messages with the option **Do not show this message again**. After reinstallation, the message will appear again.

If you rename or add templates in Windows Explorer, reload the template list to see the result. To do this, reselect the directory for report templates:
Open the list box and click the name of the directory:

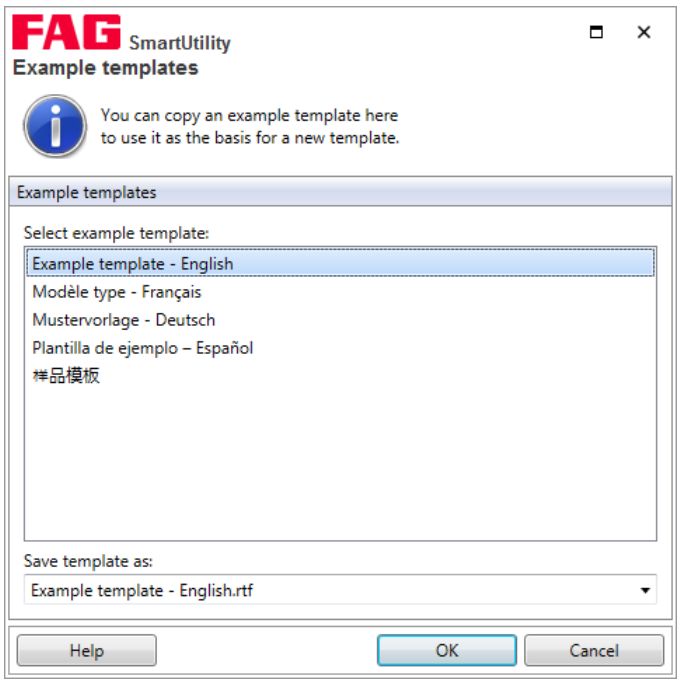
Copy new template from sample template

This function can be found in the following location in the SmartUtility software:

- In the **Create report** wizard, **step 2**
- In the menu under **More actions > Manage templates**
- In the menu under **More actions > Settings > Report**

Proceed as follows:

1. Click to open the window to select a sample template:



2. Select the sample template on which you want to base a new template.
3. Under **Save template as**, you must specify a name for the new template. By default, this is **New report template - [language]**.
4. Click **OK.** The new template will be saved automatically in the default directory for report templates.

If you accidentally delete all templates in the template directory, you can use this function to create new templates. The sample templates in the various languages are embedded in the SmartUtility software and will still be available.

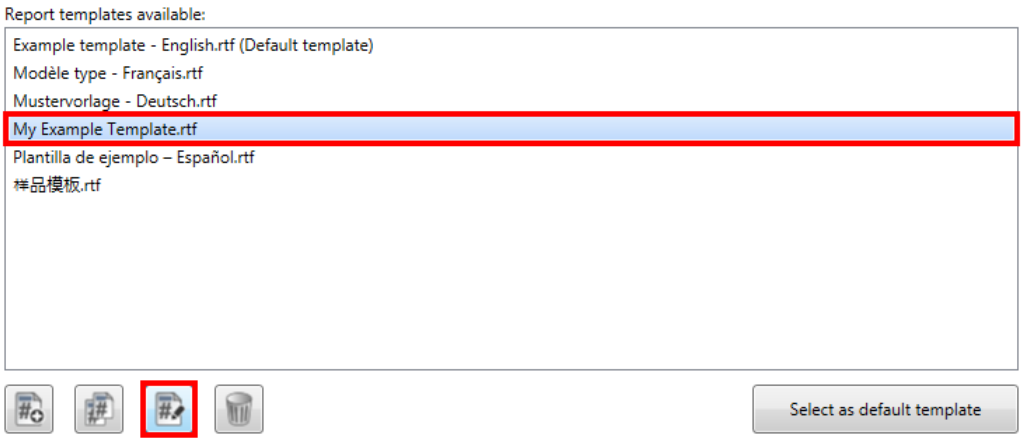
Edit template

This function can be found in the following location in the SmartUtility software:

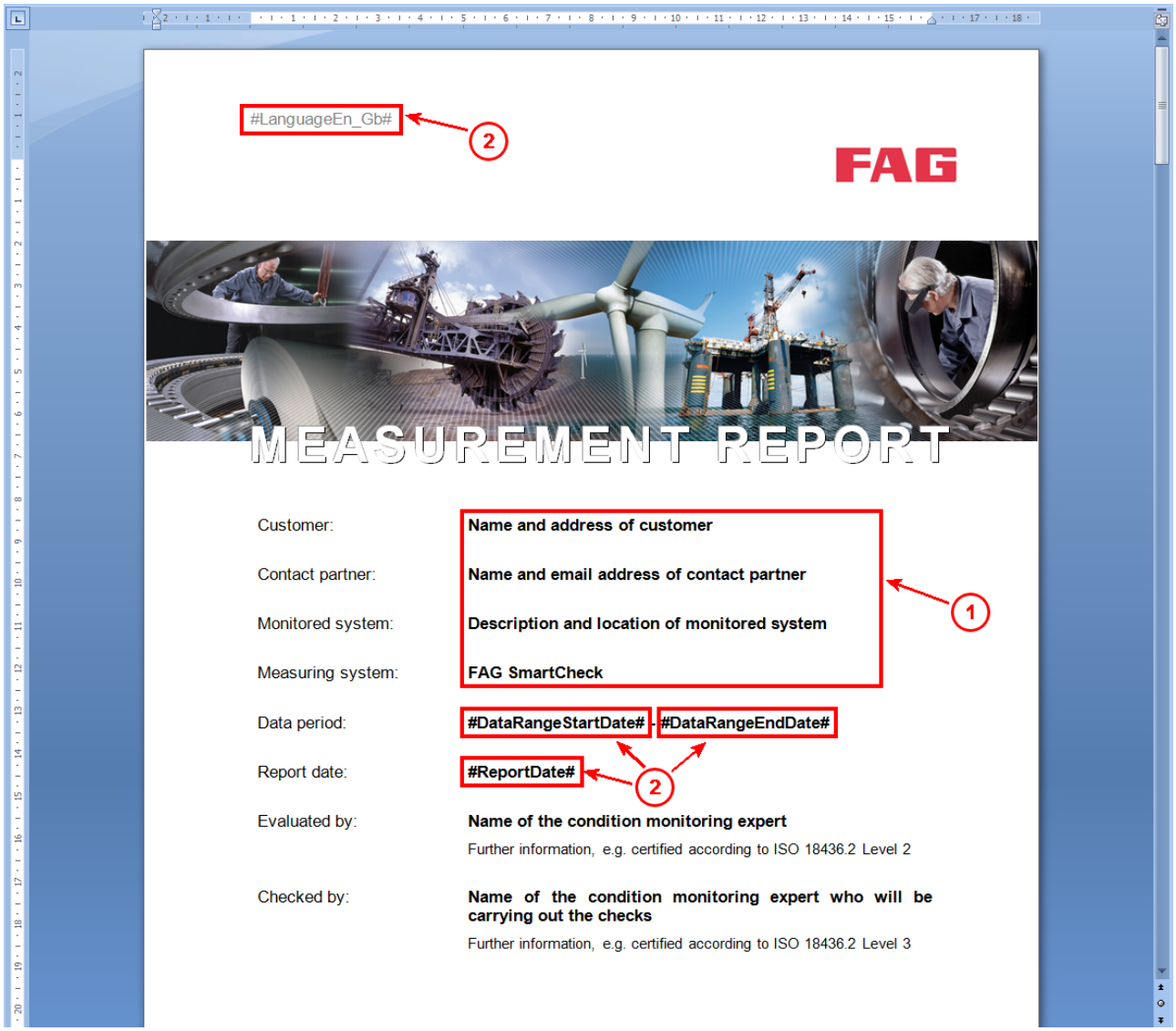
- In the **Create report** wizard, **step 2**
- In the menu under **More actions > Manage templates**
- In the menu under **More actions > Settings > Report**

Proceed as follows:

1. Select the template that you want to edit.



2. Click  to open the template in the default editor for RTF files. In this example, this is MS Word:



You have the following editing options:

- **Static content (1)**: You can edit, add to and delete the static content of the template such as the headings, contacts and footers as desired.
- **Dynamic content (2)**: Dynamic content from your devices and measurement data can be included in the report via predefined text keys, or "tags" 101, which can be identified by the hash symbol **#**. You can delete tags from the template or add additional tags to the template. Please note the rules for tags in report templates 100.

3. Saving the new template. This will be saved automatically in the default directory for report templates.

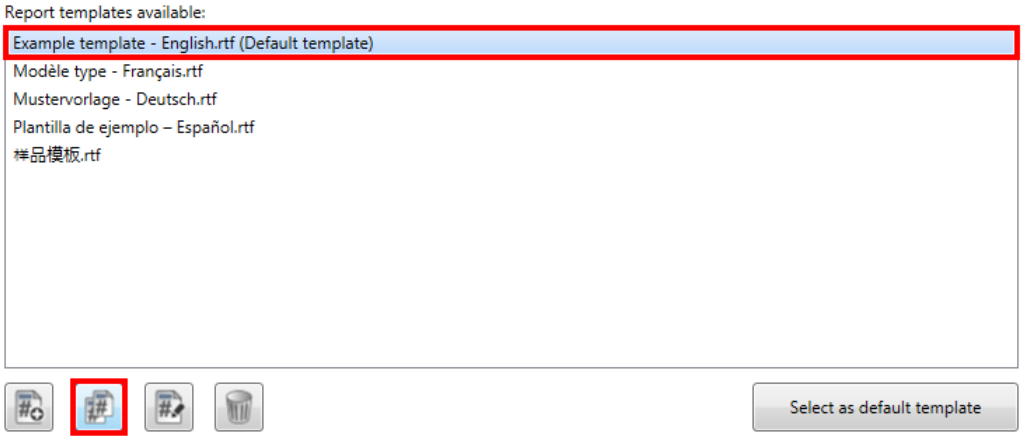
Copy template

This function can be found in the following location in the SmartUtility software:

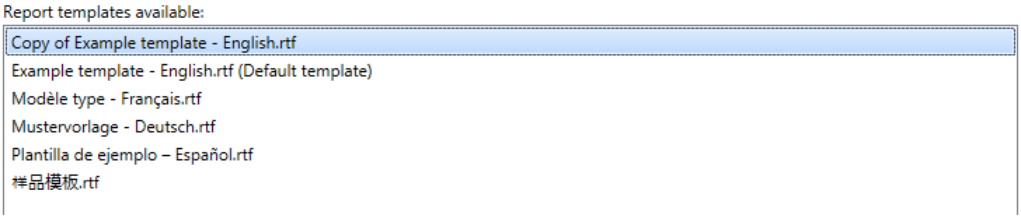
- In the **Create report** wizard, **step 2**
- In the menu under **More actions > Manage templates**
- In the menu under **More actions > Settings > Report**

Proceed as follows:

1. Select the template you want to copy.



2. Click to create a copy of the template. The copy will appear in the list immediately. The name takes the following format: **Copy of [name of the copied template]**. The copy will be saved automatically in the default directory for report templates.



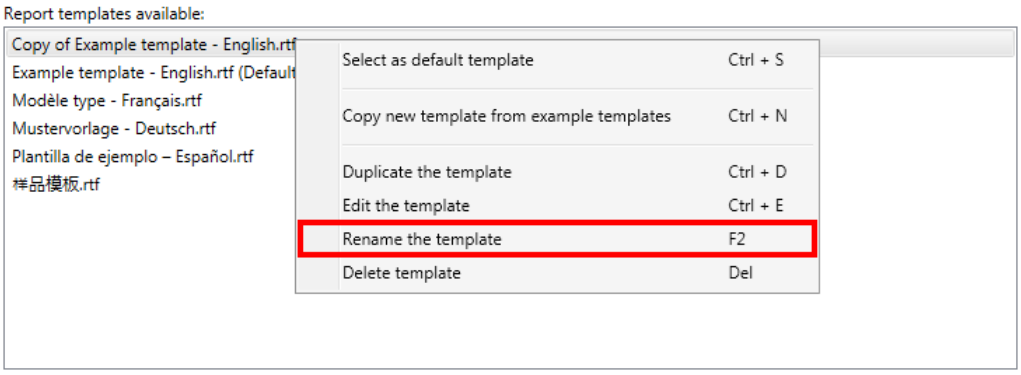
Rename the template

This function can be found in the context menu of the template list in the following locations in the SmartUtility software:

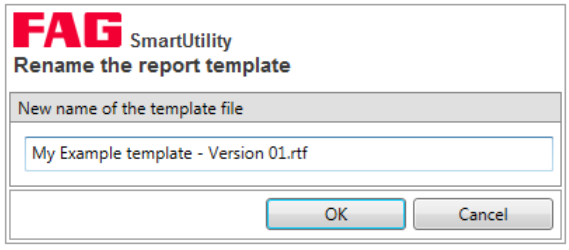
- In the **Create report** wizard, **step 2**
- In the menu under **Manage report templates**
- In the menu under **More actions > Settings**

Proceed as follows:

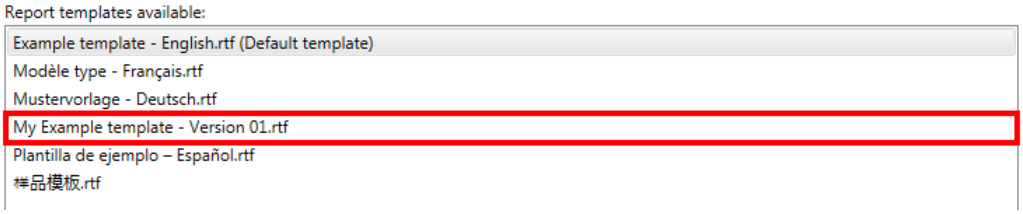
1. Select the template that you want to rename and right-click to open the context menu:



2. Select the **Rename template** option. The **Rename template** dialogue will open:



3. Enter the new name of the report template and click **OK**. The report template appears in the list with the new name:



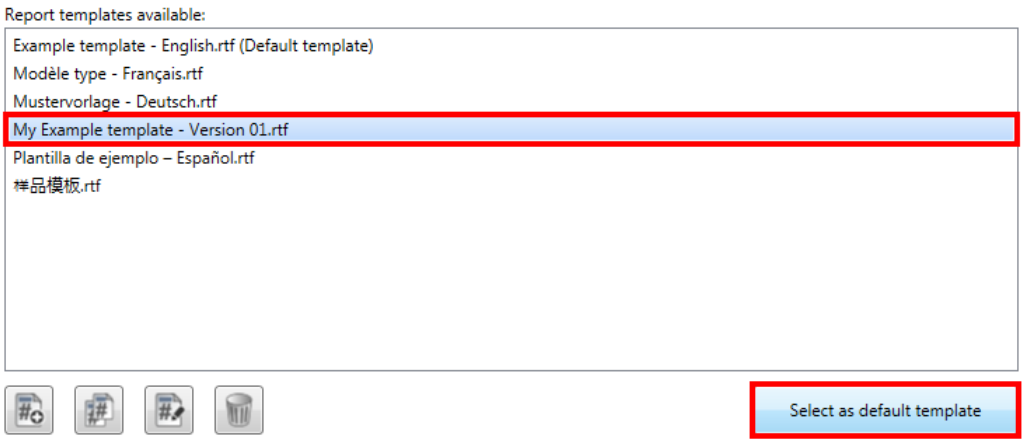
Define default template

This function can be found in the following location in the SmartUtility software:

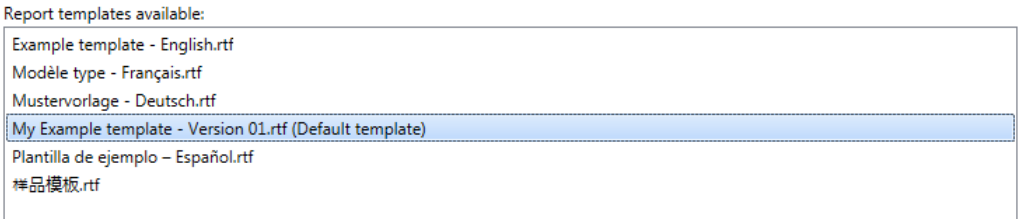
- In the menu under **Manage report templates**
- In the menu under **More actions > Settings**

Proceed as follows:

1. Select the template that you want to define as the default template:



2. Click **Select as default template**. The template will be highlighted accordingly and treated as the default template 81 by the **Create report** wizard.



5.11.2 Using tags in report templates

Predefined text keys or "tags" are important components of each report template: Tags are used to insert dynamic content, i.e. information on your devices and measurement data, into the report. For the reports in the SmartUtility software, numerous predefined tags are available. When using these tags, certain rules must be observed.

With this in mind, the subsequent sections contain the following information:

- **Important information on working with tags** 100: Basic information on the function and format of tags.
- **Language tags** 101: A list of supported language tags and information on their function.
- **Time tags** 101: A list of supported time tags and information on the corresponding dynamic content.
- **Device tags** 102: A list of supported device tags and information on the corresponding dynamic content.

Important information on working with tags

Functions

The predefined tags differ according to their functions:

- **Placeholder tags**: The majority of tags serve as placeholders. In the finished report, content that is inserted

dynamically while the report is created appears where the tags are positioned. For example, the tag **#ReportDate#** can therefore be replaced with "19/05/2015".

- **Language tags:** These tags determine the program language in which the dynamic content for all other tags appears. For example, when the language tag **#LanguageDe_De#** is used, the dynamic content appears in German. In the sample templates, the language tag is found on the first page in the top left-hand corner.
- **Device block tags**: The two device block tags **#BeginDeviceBlock#** and **#EndDeviceBlock#** mark the start and end of a device block. This device block must contain at least one placeholder tag that is used to insert the device-specific dynamic content. Placeholder tags for device-specific content function only if they are embedded into device block tags.

Format and processing

When inserting tags, the following requirements must be met:

- Each tag starts and ends with a hash #, for example: #ReportDate#.
- Within a single tag, the formatting must be consistent. It is not permitted to change the font or colour.
- Only tags defined in the SmartUtility software for the purpose of creating the report can be used. These tags are listed in the following sections. It is not possible to create your own tags.
- Placeholder tags in a report template can be deleted, moved to a different location or replaced with other available tags as required.
- If device block tags are deleted, the corresponding placeholder tags lose their function.

Overview of all language tags

This overview shows all language tags defined in the SmartUtility software for the purpose of creating the report.

Tag	Explanation
#LanguageDe_DE#	The program language used to display the dynamic content in the finished report is German.
#LanguageEn_Gb#	The program language used to display the dynamic content in the finished report is English.
#LanguageEs_Es#	The program language used to display the dynamic content in the finished report is Spanish.
#LanguageZh_Cn#	The program language used to display the dynamic content in the finished report is Chinese.
#LanguageFr_Fr#	The program language used to display the dynamic content in the finished report is French.

Language tags effect the language only of text added dynamically via tags. Fixed text specified by the sample template remains unchanged. Such text includes headings, header text or individual details of contact persons and system designations, for instance.

You must install the corresponding Windows language pack for the language in which you would like to create a report. If foreign-language entries in the **Logbook** section are incorrect or display with invalid characters, you must also install the Windows language pack for the language of the logbook entries. Note that some language packs are only available for the Professional or Ultimate versions of the operating system.

If a report does not contain a language tag, the dynamic content appears in the current system language. If the SmartUtility software does not support your system language, the content is inserted in English.

Overview of all time tags

These tags are used to insert information on the creation date and the overall data period into the report. Each of these tags can be used independently of any other tags. The format of the date and time correspond to the language specified by the language tag.

Tag	Explanation and example

#ReportDate#	Date on which the report was created.	18/05/2015
#ReportTime#	Local time at which the report was created.	13:45
#ReportTimeGmt#	Local time at which the report was created. Brackets after the time contain the difference to the standard time zone GMT (Greenwich Mean Time).	13:45 (GMT +02:00)
#DataRangeStartDate#	First day of the data period. The oldest data to be included in the report dates back to this day.	18/02/2015
#DataRangeStartTime#	Time at which the first data of the data period was measured.	09:43
#DataRangeStartTimeGmt#	Time at which the first data of the data period was measured. Brackets after the time contain the difference to the standard time zone GMT (Greenwich Mean Time).	09:43 (GMT +02:00)
#DataRangeEndDate#	Last day of the data period. The most recent data to be included in the report dates back to this day.	18/05/2015
#DataRangeEndTime#	Time at which the last data of the data period was measured.	17:57
#DataRangeEndTimeGmt#	Time at which the last data of the data period was measured. Brackets after the time contain the difference to the standard time zone GMT (Greenwich Mean Time).	17:57 (GMT +02:00)

Overview of all device tags

The device tags are used to enter device-specific information into the report. All of these tags function only if they are embedded into a device block. The device block is defined by the tags **#BeginDeviceBlock#** (start of the block) and **#EndDeviceBlock#** (end of the block).

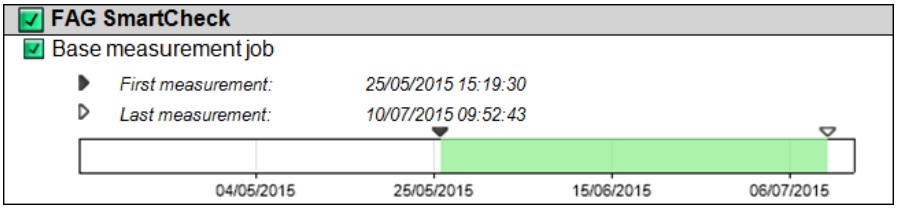
Tag	Explanation and example	
#BeginDeviceBlock# **#EndDeviceBlock#**	Start (**#BeginDeviceBlock#**) and end (**#EndDeviceBlock#**) of a device block. Each of the following tags must be contained within a device block in order to be populated when the report is created.	
#Counter#	Numbering for each device within a device block	1
#DeviceName#	Name of the device	FAG SmartCheck
#DeviceSerial#	Serial number of the device	f4:3d:80:00:07:55
#DeviceIp#	IP address of the device	172.28.205.60
#LastDataDownloadDate#	Date and time of the last data download This is the time at which data was downloaded from the device last. Regardless of the report period set, data that can be evaluated is available only up to this time.	09/03/2015 13:35:43
#TableDeviceInfo#	Table with the following information on the device: • Symbol for the overall alarm status	

<table>
<tr><td></td><td>

Name of the device
IP address
Serial number
Last data download
Firmware version

Example:
<table>
<tr><th colspan="2">FAG SmartCheck</th></tr>
<tr><td>IP address</td><td>172.28.205.60</td></tr>
<tr><td>Serial number</td><td>f4:3d:80:00:07:55</td></tr>
<tr><td>Data download</td><td>10/07/2015 09:54:09</td></tr>
<tr><td>Firmware</td><td>1.6.12</td></tr>
</table>
</td></tr>
<tr><td>#ReportAlarmStatusCurrentDevice#</td><td>The alarm display in the trend report is based on the device status, which is also displayed in the device table.</td></tr>
<tr><td>#ReportAlarmStatusLast#</td><td>The alarm display in the trend report is based on the last measured value from the selected period.</td></tr>
<tr><td>#TableTrendReport#</td><td>
Table with detailed trend information:

Name of the device
Symbol for the alarm status of the configuration
Configuration name
Time of the first and last measurement
Trend diagram
Symbol for the overall alarm status

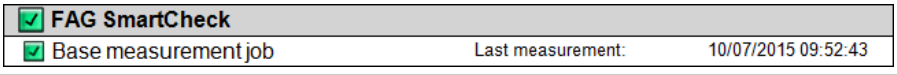
Example:



The trend diagram always covers the entire data period. This fact can lead to a loss of information within the diagram, especially if the overall data period is large.
Example scenario:

Date period: 3 months
Last alarm status of the device: Pre-alarm (data from 3 hours)
Trend diagram: The bar representing a period over several months is completely green. The yellow area for the pre-alarm in the last 3 hours disappears/is no longer visible as a result of scaling.

</td></tr>
<tr><td>#TableTrendReportShort#</td><td>
Table with a summary of the key trend information:

Symbol for the overall alarm status
Name of the device
Symbol for the alarm status of the configuration
Configuration name
Time of the last measurement

Example:


</td></tr>
</table>

#TableLogbook#	Table with logbook entries of the device **Example:**

FAG SmartCheck

	Created	Changed	User	Message
	07/07/2015 14:38:01	07/07/2015 14:38:01	system	Open current loop detected at analogue output!
	07/07/2015 14:36:07	07/07/2015 14:36:07	system	System time changed by user admin from 07/07/2015 14:25:38 to 07/07/2015 14:36:07.
	07/07/2015 14:24:42	07/07/2015 14:24:42	system	Current input error: input Analogue input 2 dropped below 4 mA (probably due to cable break).
	06/07/2015 22:22:28	06/07/2015 22:22:28	system	Base configuration: the alarm status has been changed from "Pre-alarm" to "No alarm".

The alarm display in the trend report can be manipulated using the tags **#ReportAlarmStatusLast#** or **#ReportAlarmStatusCurrentDevice#**. If these tags are not inserted, the **#ReportAlarmStatusLast#** setting is used by default: The alarm display is based on the last measured value from the selected period.

5.12 Rescan data directory

This version of the FAG SmartUtility Viewer software includes the introduction of a new file format. Data in the previous file format can no longer be loaded. This wizard enables you to convert data that has already been downloaded to the new file format. This means that you can continue to open the data without having to download new data from the FAG SmartCheck device.

You can also use this wizard to convert measurement data into raw format for analysis. This may be necessary, for instance, if you have activated the option **Download raw data only** in the **Download data** wizard.

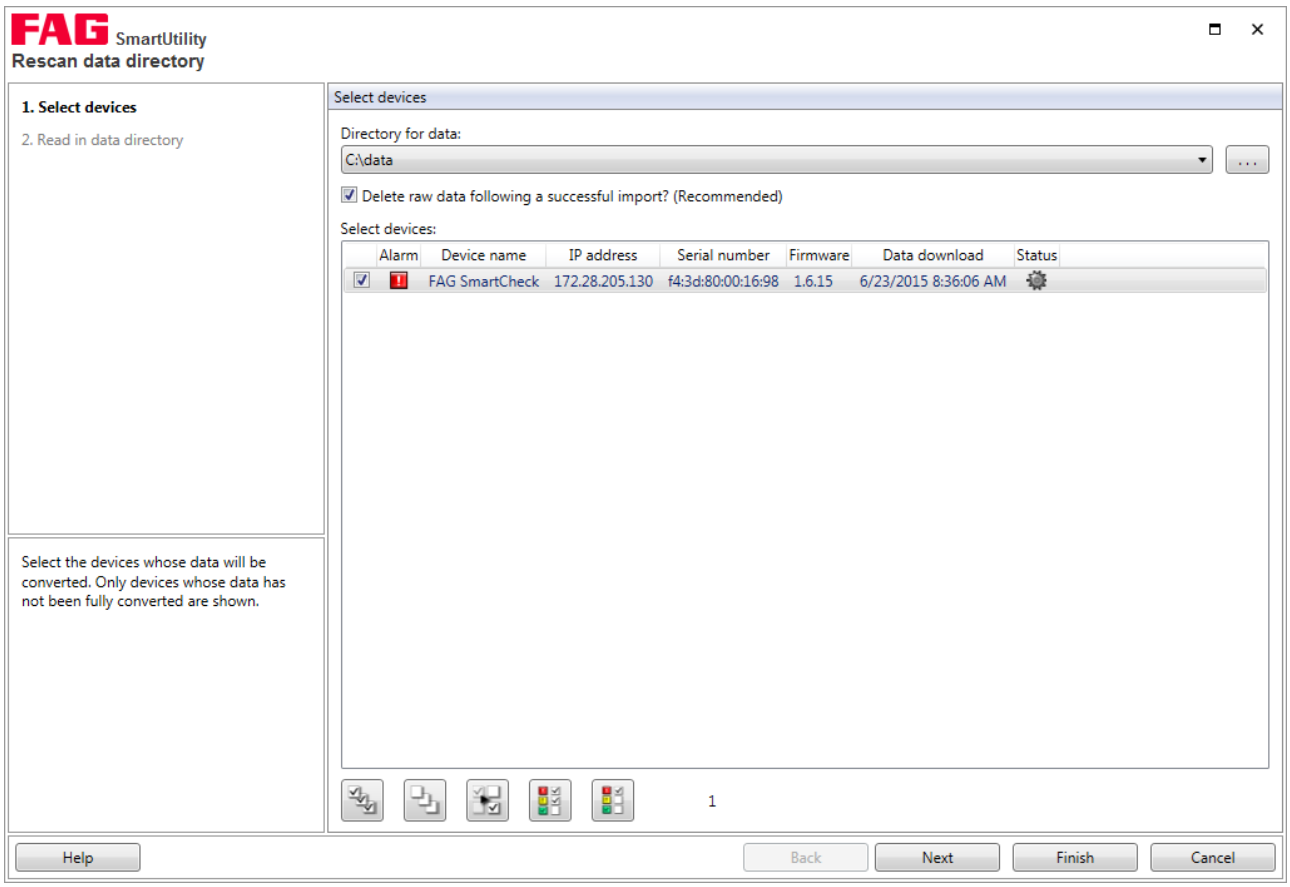
To reimport the data directory, proceed as follows:

Step 1:

Select the directory in which the data is located. This may be the default directory 108 or a directory you have selected yourself. If you activate the **Delete raw data following successful import?** option, the data will be deleted after the wizard has completed successfully. If you do not delete the data, the system will try to scan the data again the next time. This will make the procedure take longer.

Step 2:

Select the required SmartCheck device that contains the data you want to reimport. You can also select multiple devices. A symbol in the **Status** column tells you whether data has already been converted to the required format. If you move the mouse over the symbol 19, you will obtain more information about the conversion status.



Result:

The converted measurement data can be found in the directory you specified in **Step 1**, and can now be analysed with the SmartUtility Viewer.

If the original data or alarm information is no longer available, this procedure will automatically reset the alarm status. The alarm status will be displayed again the next time you download data from the FAG SmartCheck device or update the measurement data for all devices in the SmartUtility Viewer software.

5.13 Export data

Use this wizard to export measurement data already downloaded to a different save location, to forward it, for example, for analysis.

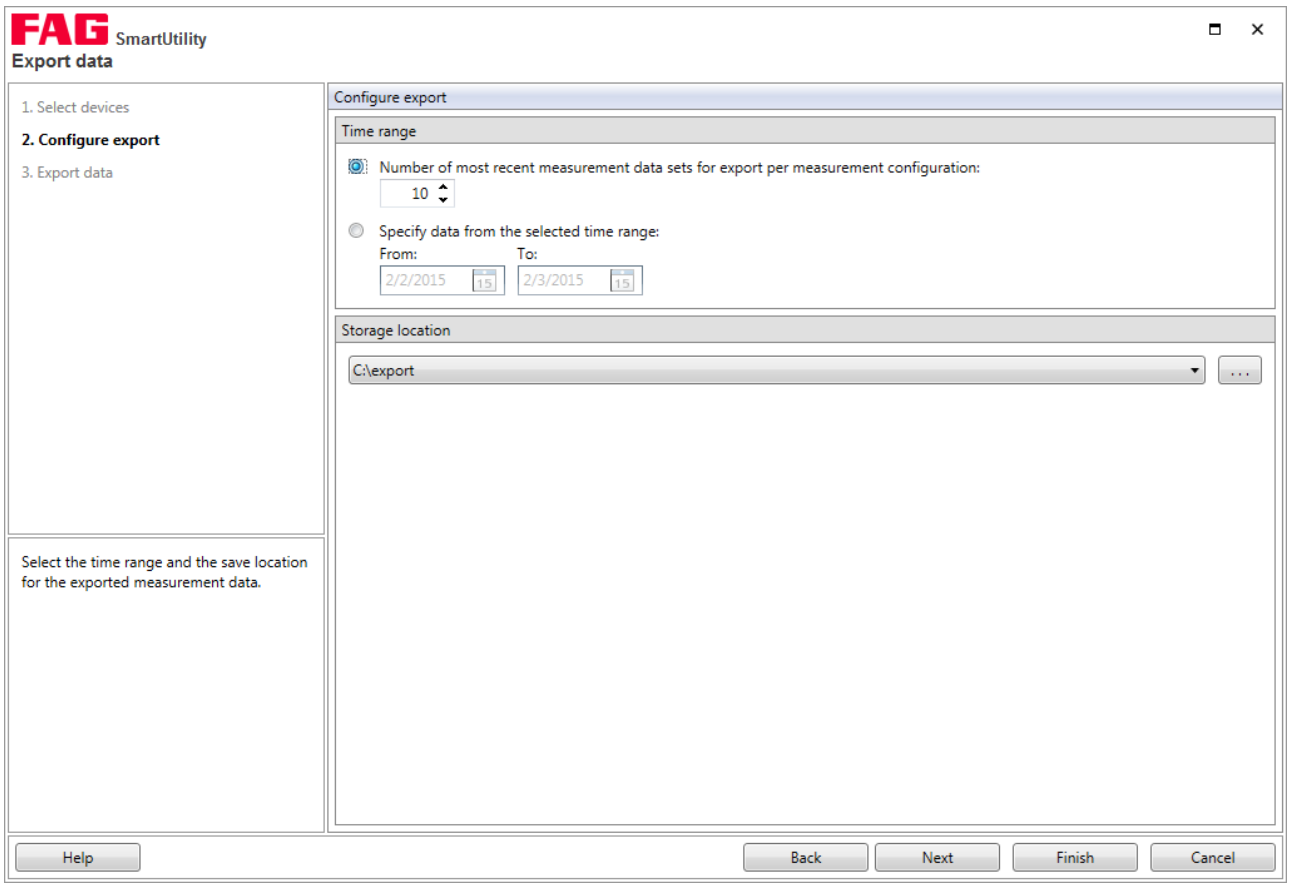
To export measurement data, proceed as follows:

Step 1:

1. Select the directory containing the measurement data. This may be the default directory 108 or a directory you have selected yourself.
2. Select the required SmartCheck device or devices whose measurement data you want to export.

Step 2:

Next, determine the time range for which the measurement data is to be exported. You also specify the save location for the data here.



Here you have the following options:

Time range	This is where you determine the time range for which the measurement data is to be exported: • **Number of most recent measurement data sets for export per measurement configuration:** Only the most recent measurement records (i.e. the time signals) are exported. The trends are always exported in full. • **Specify data from the selected time range:** Only the measurement data for the selected time period will be exported. If you activate this option, the default setting is one day. You can enter the dates for your desired time range in the **From**and **To** fields or select the dates from the calendar.
Storage location	Here you can determine the directory in which the exported measurement data is to be saved. If the list box is empty, click on ... to select a directory and specify this as the save location.

Step 3:

The data is exported and saved in the directory you specified in **Step 2**. Wait until the process is fully completed.

Result:

In the directory you specified in **Step 2** you will find a full data directory with the exported measurement data.

5.14 Import SmartWeb data

In the SmartCheck device, if you use the **Download measurement data** option, you will receive the measurement data in the format *.scd1. To be able to edit this measurement data with the SmartUtility software, you must import it into the SmartUtility software with this wizard.

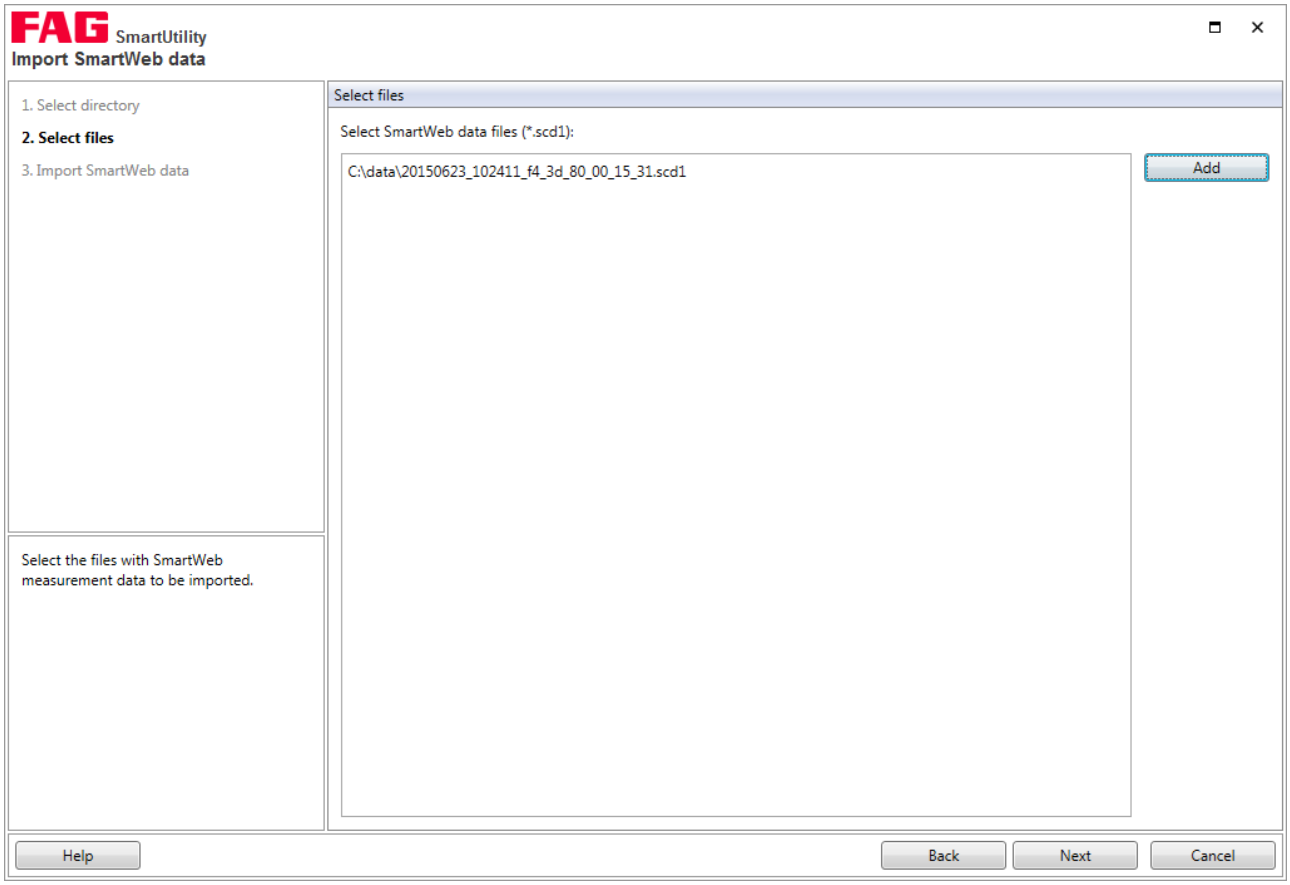
To import measurement data in the *.scd1 format, proceed as follows:

Step 1:

Select the directory to which the SmartWeb data is to be imported. This may be the default directory 108 or a directory you have selected yourself.

Step 2:

Select the required measurement data files in the format *.scd1. To do so, click on **Add**, browse for the required file and select it with **Open**. Repeat these steps for each additional measurement data file.



Step 3:

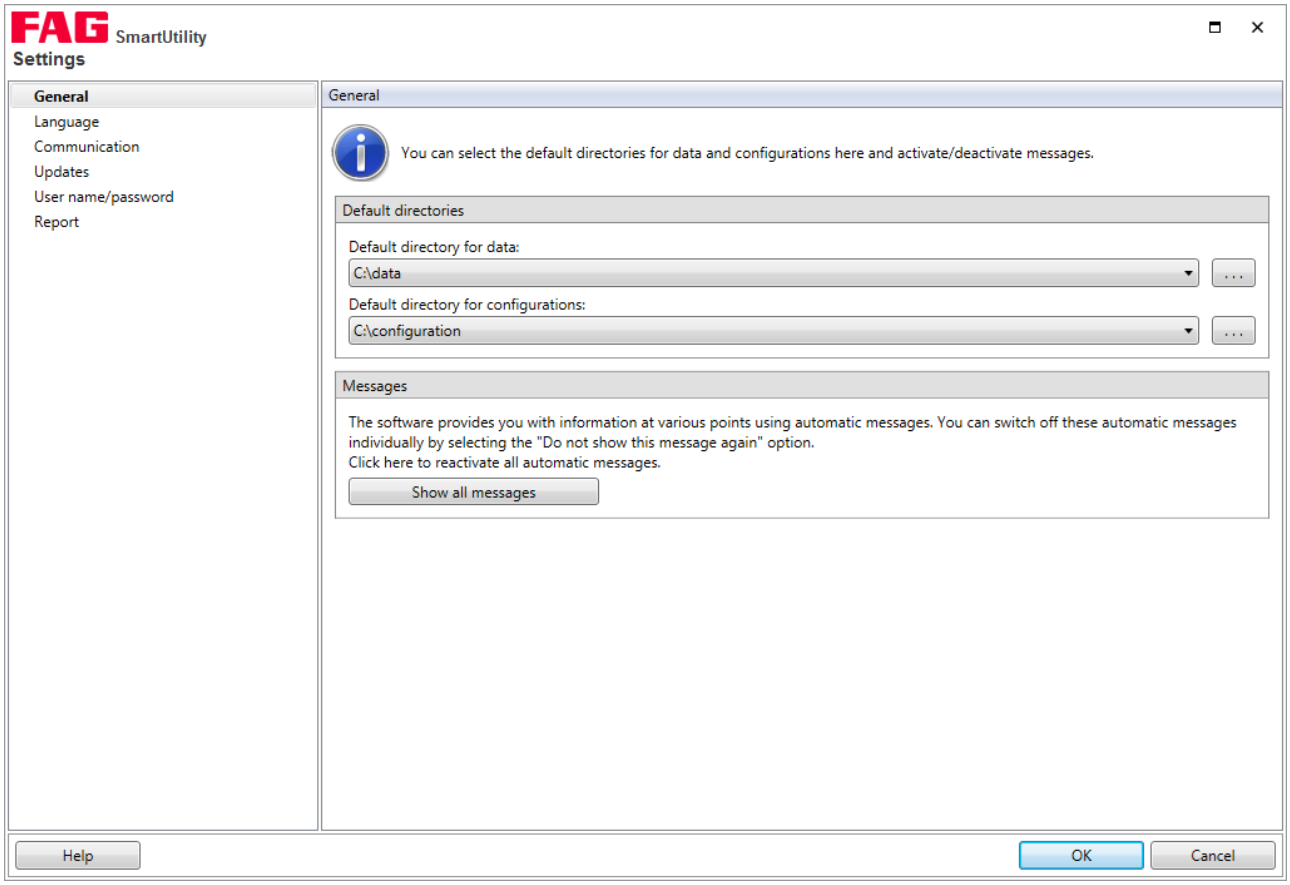
The measurement data files are imported and you are notified as to whether the import has been successful. The **Result** column contains details about the import procedure.

Result:

You will find the imported measurement data in the directory you specified in **Step 1**.

6 Settings

Default settings can be found under **More actions > Settings**. You can use these settings to adapt the SmartUtility software to suit your requirements. Click on a term on the left in order to specify the settings in the area on the right. You must confirm all changes by clicking **OK**:



Here you have the following options:

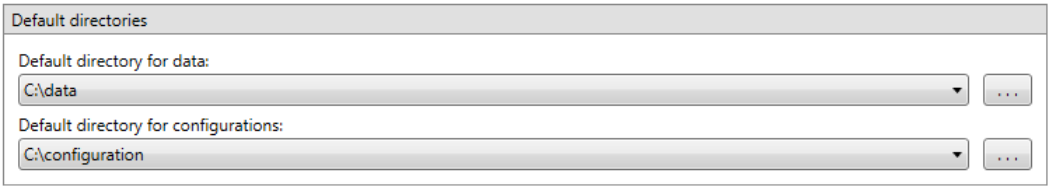
General

Default directories

Here you can change the default directories. These are used by the SmartUtility software for downloaded data and configuration files and are automatically created during installation at the following path:

- Windows 7:
  C:\data
  C:\configuration

Click ... to search for a new directory and set it as a default directory. You can now find directories that you have previously selected in the list box.

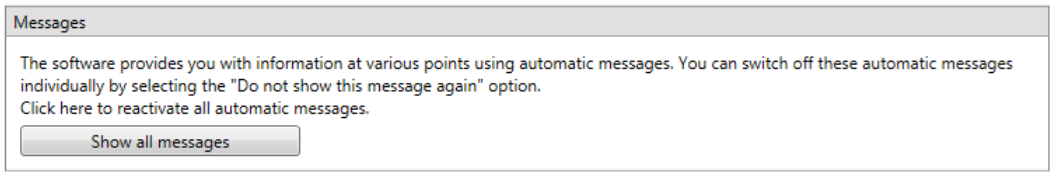


Make sure that the downloaded data is always saved to the default directory. If you want to specify your own directory, make sure that all the data is available in this directory. This is required for optimum trend analysis and evaluation of the measurement data in the Viewer.

Messages

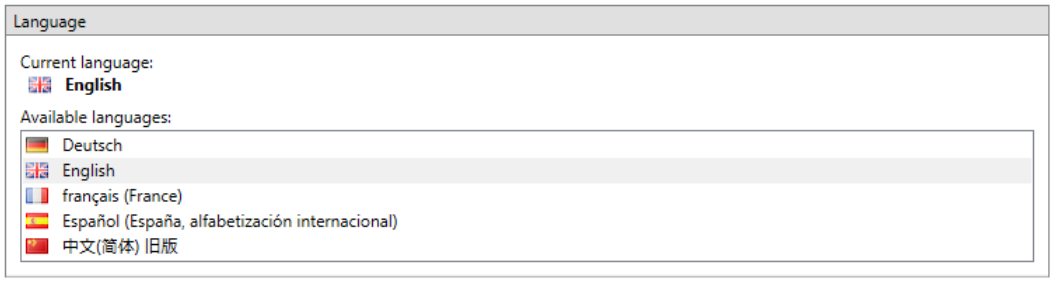
The SmartUtility software recognises various types of messages, e.g. warnings, general information and regular messages regarding updates. You can usually activate the **Do not show this message again** option in these messages so that the message no longer appears.

You can revert this change in the **Settings > Messages** section. Click **Show all messages** to view all warnings, information and other messages again.



Language

Here you can set the language in which the SmartUtility software interface is displayed. Click on an available language:



The following languages are available:

Sprache	Language	Idioma	Langue	Hànyu
Deutsch (German)	English (English)	Español (Spanish)	Français (French)	中文 (Chinese)

Communication

UDP ports

Here you can set the UDP port that is used to search for your devices and for configuration. In addition, with the option **Interval for the sensor search via UDP** you can determine the frequency at which the SmartUtility software is to search for devices and update the corresponding list with SmartCheck devices in the wizards.

By default, the UDP port is set to 19000 for the device search. The search interval is set to 120 seconds.

The following basic prerequisites apply for connecting to the computer:

- The UDP communication protocol must be enabled on the used port 19000 in existing firewalls.
- If the SmartCheck device has not been assigned an address via DHCP, it will have the IP address 192.168.1.100 by default. In this case, the IP address of your computer must be within the 192.168.1.x range.
- During the initial installation of the SmartUtility software, the setting for the UDP port is made automatically. These automatic settings are generally correct.

If there are any problems, please contact your system administrator.

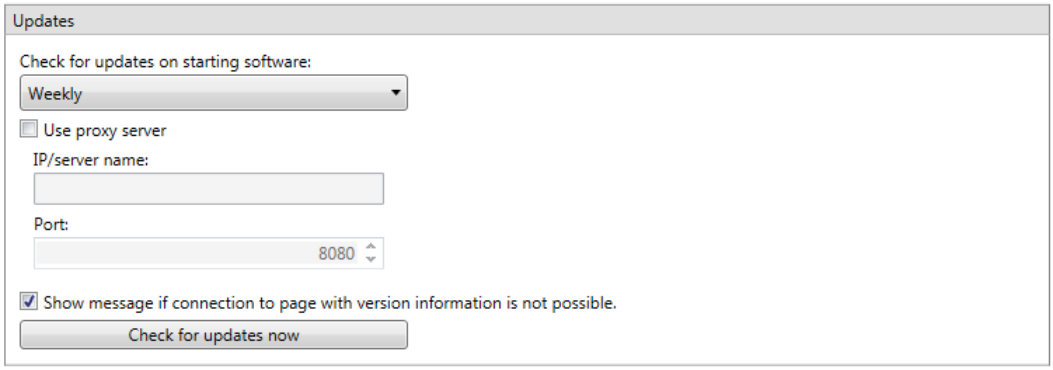
Maximum number of parallel device connections

If you have selected multiple devices for a task inSmartUtility, some tasks may be processed in parallel on multiple devices. For some jobs, there are a maximum number of parallel device connections with preset standard values. You can change the standard values here and adjust the performance of your network environment:

- **Downloading data:** By default, you can download data from 1 device.
- **Sending/downloading configurations:** By default, you can send/download 1 configuration at a time.
- **Update firmware:** By default, you can update the firmware of 20 devices at the same time. You can increase this value to a maximum of 30 devices.

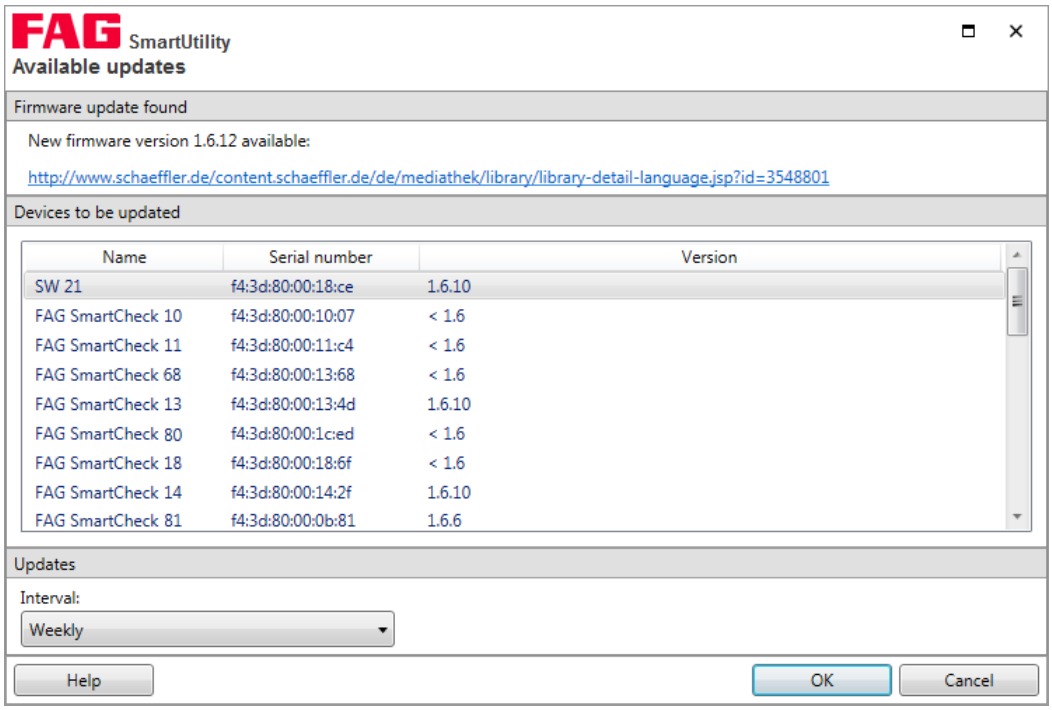
Updates

You can set the SmartUtility software to automatically search for updates for the SmartUtility software and the SmartCheck device. To do so, set the intervals at which you want the system to search for new updates. If you want to deactivate the update test, select **Never**:



You can also set a message to display if the SmartUtility software fails to access the website with the update information.

The **Check for updates now** button allows you to search for updates outside of a regular check. The following dialogue appears:



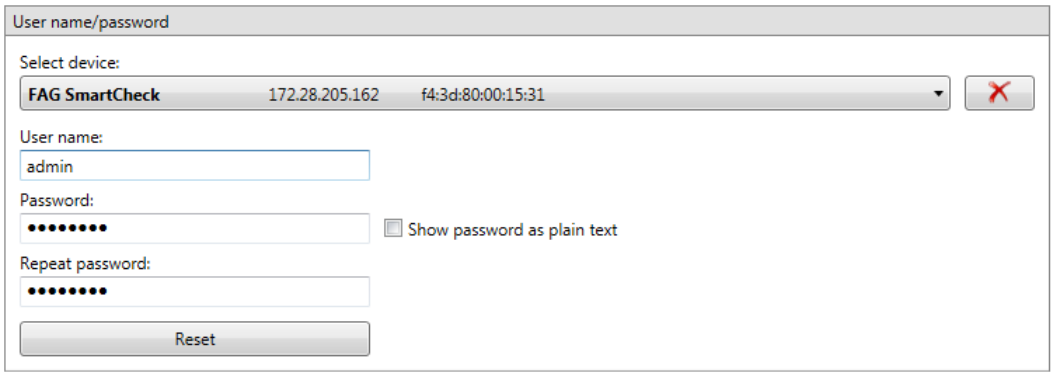
The following information and functions are to be found here:

- The first section of the dialogue contains information on whether any updates were found and where you can download them.
- The **Updates** section provides you once again with the selection list in which you can determine the frequency at which the system should search for updates.

If you are using a proxy server between the browser and the Internet in your company network, please activate **Use proxy server** and enter the address and port number of the proxy server. Please contact your system administrator for more detailed information about proxy settings.

User name/password

Here you can change the user name and password for every SmartCheck device that the SmartUtility software finds. Select the required device in the **Select devices** list. This list contains all the SmartCheck devices that have ever been detected. The IP address is displayed if the device is available or has been added manually. Enter the user name and password, and then repeat the password to confirm:



You can log in automatically to a SmartCheck device or the integrated SmartWeb software using the user name and password. To do so, your details here must match the user name and password stored in the SmartWeb software user management.
If this is not the case, you will not be logged in automatically. You will have to log in using the user name and password stored in the SmartWeb software.

Deleting devices

Manually entered devices remain stored in the SmartUtility software. If you no longer need to use a device, you can remove it from the device list. Select the required device from the **Select device** list and click on .

Report

You can use the SmartUtility software to create reports in RTF format from the downloaded SmartCheck device data. The templates for these reports can be managed in the following locations within the SmartUtility software:

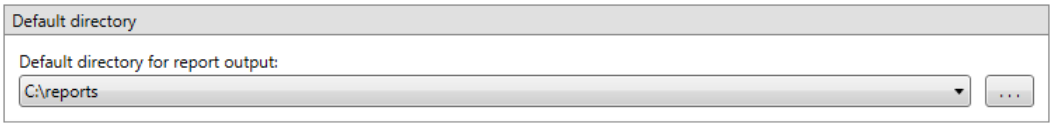
- In the menu under **More actions > Manage templates**
- In the menu under **More actions > Settings > Report**

Default directory

Here, you can change the directory to which the finished reports are saved by default. The directory is automatically created at the following path at installation:

C:\reports

Click ... to search for a new directory and set it as the default directory for reports. You can now find directories that you have previously selected in the list box.



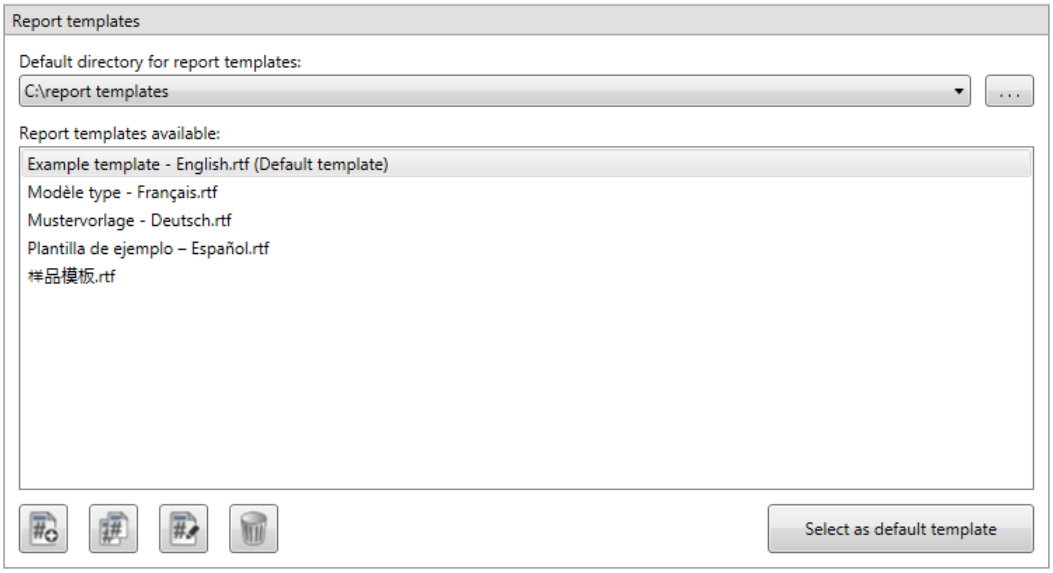
Report templates

Here, you can change the directory to which your report templates are to be saved by default. The directory is automatically created at the following path at installation:

C:\report templates

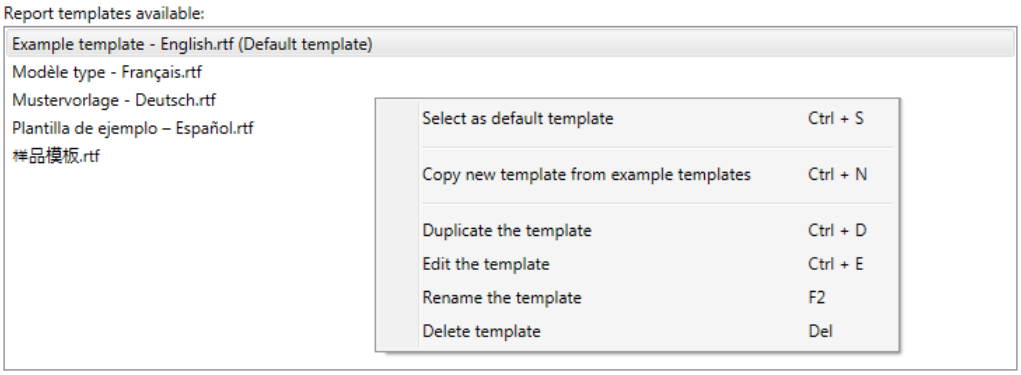
Click ... to search for a new directory and set it as a default directory for your report templates. You can now find directories that you have previously selected in the list box.

Here, you can also manage the available report templates, create new templates and set a template as the default template:



Here you have the following options:

	Here, you can copy a new template from the sample templates 96.
	Here, you can duplicate selected templates 98 e.g. to edit the copy.
	Here, you can edit selected templates 97 and modify them to meet your requirements.
	Here, you can delete the selected templates after confirmation.
Select as default template	Here, you can define the selected template as the default template 96.
Rename the template	This option is only available in the context menu (see below). Click this option to open a dedicated dialogue to change the template name 99.
Context menu	Right-click the selected template to open the context menu to access the options described above: **Copy new template from sample template**, **Duplicate template**, **Edit template**, **Rename template** and **Delete template**.

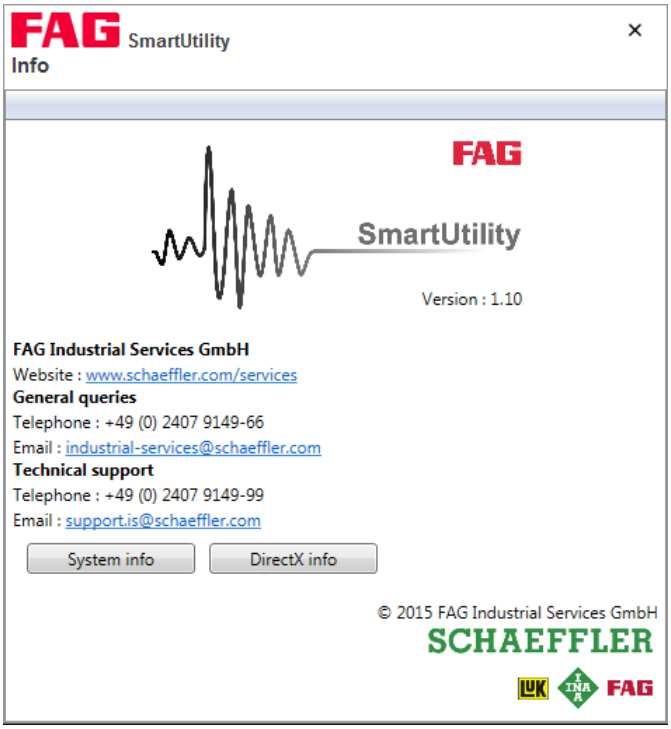


The following hotkeys can also be used for the options described above:

Ctrl+S	Select as default template
Ctrl+N	Copy new template from sample template
Ctrl+D	Duplicate template
Ctrl+E	Edit template
F2	Rename template
DEL	Delete template

7 Further information

Information on support and the current program version can be found under **More actions > Info**:



Here you have the following options:

- **Version**: Here you can view the current version of your SmartUtility software
- **Website:** Click this link to switch to the Schaeffler Industrial Services website.
- **Email**: Click this link to email a general query to Schaeffler Industrial Services.
- **System info**: Click this button to switch directly to the **System information** page of your Windows system.
- **DirectX info**: Click this button to switch directly to the DirectX diagnostic program.

Information and services relating to FAG SmartCheck

We offer a unique range of services for FAG SmartCheck — from training courses, technical mentoring during the induction phase and expert support with diagnostic issues, right through to customised service agreements including remote monitoring and reporting.

A selection of our comprehensive range of products and services for FAG SmartCheck can be found on the Internet at www.FAG-SmartCheck.com.

8 Manufacturer/support

Manufacturer

FAG Industrial Services GmbH

Kaiserstraße 100
52134 Herzogenrath
Germany

Tel.: +49 (0) 2407 9149-66
Fax: +49 (0) 2407 9149-59
Support: +49 (0) 2407 9149-99

Website: www.schaeffler.com/services
Further information: www.FAG-SmartCheck.com
Contact: industrial-services@schaeffler.com

Please send all correspondence directly to FAG Industrial Services GmbH!

A subsidiary of
Schaeffler Technologies AG & Co. KG

PO Box 1260
97419 Schweinfurt
Germany

Georg-Schäfer-Straße 30
97421 Schweinfurt
Germany

Support

Tel.: +49 (0) 2407 9149 99

Email: support.is@schaeffler.com

We provide support services for the FAG SmartCheck device and related software products. A detailed description of the type and scope of the support services we provide can be found online at www.FAG-SmartCheck.com.

SCHAEFFLER

LuK INA FAG