User Manual

EVIDAS

Release 1.4

HBM

A04766_01_E00_04

Publication information

Edition notice

This publication is intended for engineers and other personnel using EVIDAS to perform measurement tasks.

Every effort has been made to ensure that all the information contained in this publication is correct at the time of publishing. However, the manufacturer may need to update the information as a result of product surveillance and development activities, leading to a new version of this publication.

Training and Support

For training and support contact the local HBM representative or the manufacturer (Manufacturer, 4).

Copyright

Trademarks

EVIDAS is a registered trademark of Hottinger Baldwin Messtechnik GmbH.

The EVIDAS logo is a trademark of Hottinger Baldwin Messtechnik GmbH.

All other trademarks are the property of their respective owners.

Feedback

Every effort has been made to ensure that this publication fulfills the intended purpose. All feedback on any aspect of this publication is welcome and is considered during updates. Should you have any such feedback, contact the local HBM representative or the manufacturer (Manufacturer, 4).

Symbols

ⓘ	Additional information on a step
⇗	Cross-reference
⇨	List of related topics
✓	Step result
▭	Table title
▢	Illustration title

Manufacturer

Hottinger Baldwin Messtechnik GmbH
Im Tiefen See 45
64293 Darmstadt
GERMANY

Phone	+49-6151-8030
Fax	+49-6151-8039100
eMail	info@hbm.com
Web	www.hbm.com

Table of Contents

1 Quick Start Guide

1.1 Installing EVIDAS

Prerequisites

- Internet connection.

To install EVIDAS

1 Download EVIDAS from the HBM website.

ⓘ If you want to install EVIDAS on an offline computer, copy the setup file to the offline computer.

2 Double-click the setup file, follow the instructions, and select **Create a desktop icon**.

✓ EVIDAS is installed and the EVIDAS icon  is displayed on the Windows desktop.

3 If you want to use EVIDAS as a free viewer for data files, proceed here:
Using EVIDAS as a viewer, 22

4 If you want to test a full version of EVIDAS for 30 days, proceed here:
Using a trial version of EVIDAS, 21

5 If you want to use an EVIDAS single license:

a) Purchase a single license, e.g., through the HBM online shop. You receive an email with an activation key.

b) Proceed with one of the following options:
Activating an EVIDAS single license online, 13
Activating an EVIDAS single license manually, 15

6 If you want to use an EVIDAS network license, proceed here:
Registering for an EVIDAS network license, 19

1.2 Activating EVIDAS

To work with EVIDAS, you must activate it, i.e., specify that you are an authorized user. Activation is a secure process in which the product features are enabled.

How you activate EVIDAS, depends on the license type you have.

Network license

Your organization has purchased and installed EVIDAS network licenses to be shared by authorized users:

Registering for an EVIDAS network license, 19

Single license

You have purchased an EVIDAS single license and received an email with an activation key.

To activate an EVIDAS single license on an online computer:

Activating an EVIDAS single license online, 13

To activate an EVIDAS single license on an offline computer:

Activating an EVIDAS single license manually, 15

No license

If you do not have a license, you can still use EVIDAS:

- Using EVIDAS as a viewer, 22
- Using a trial version of EVIDAS, 21

1.2.1 Activating an EVIDAS single license online

Use this procedure to activate an EVIDAS single license on an online computer.

Prerequisites

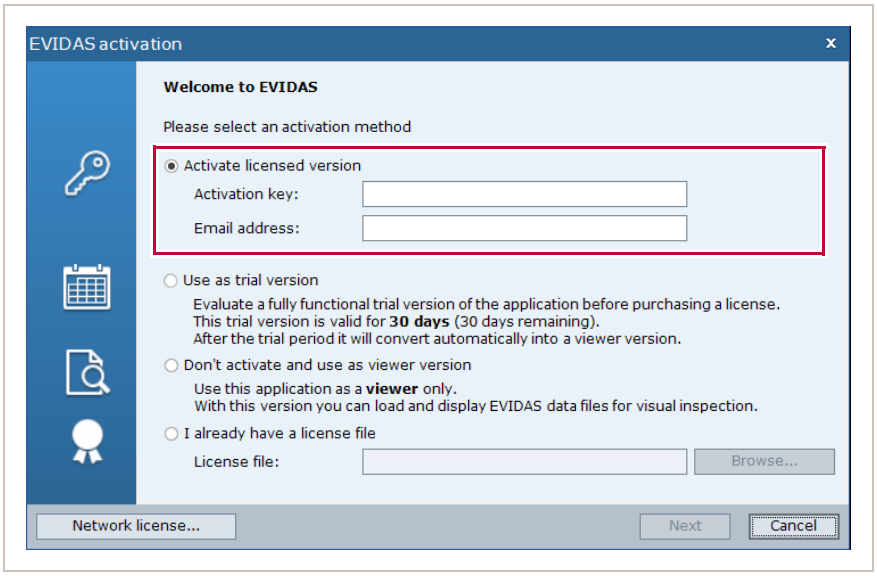
- EVIDAS is downloaded and installed (Installing EVIDAS, 11).
- EVIDAS single license is purchased.
- Internet connection.

To activate an EVIDAS single license online

1 Double-click  .

✓ The **EVIDAS activation** dialog box is displayed.

2 Select **Activate licensed version**.

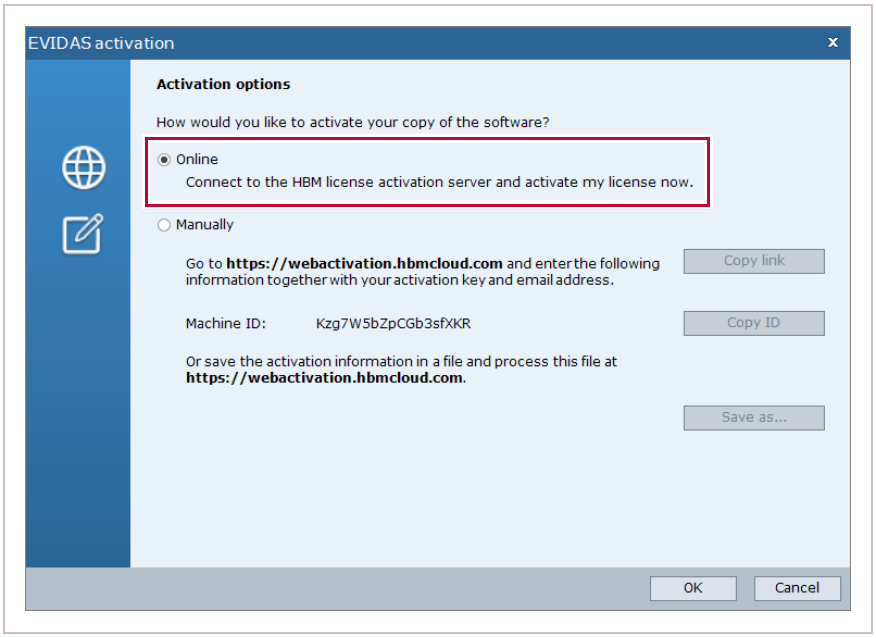


3 Enter your activation key and email address.

4 Click **Next**.

✓ The EVIDAS activation options are displayed.

5 Select **Online**.



6 Click **OK**.

ⓘ Please wait until notified that EVIDAS is activated.

1.2.2 Activating an EVIDAS single license manually

Use this procedure to activate an EVIDAS single license on an offline computer.

Prerequisites

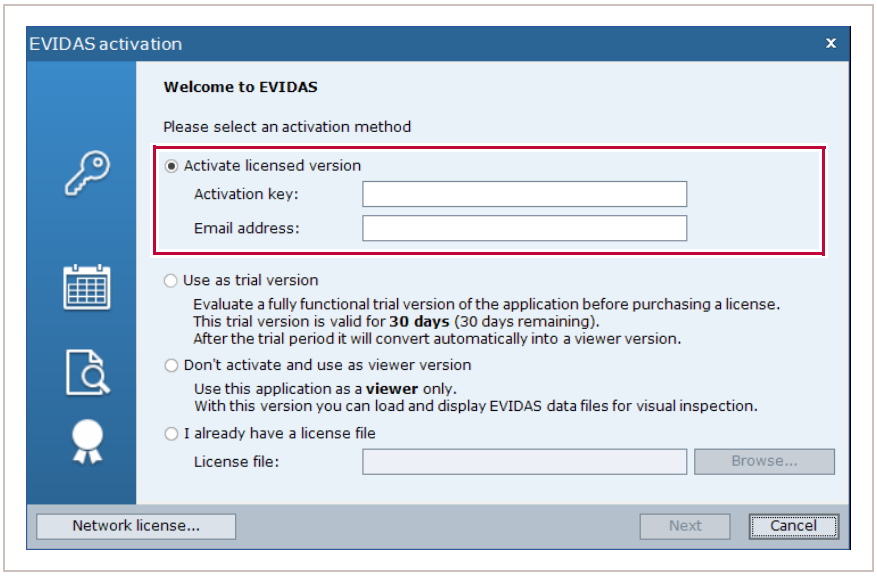
- EVIDAS is downloaded and installed (Installing EVIDAS, 11).
- EVIDAS single license is purchased.
- Internet connection via a second computer.

To activate an EVIDAS single license manually

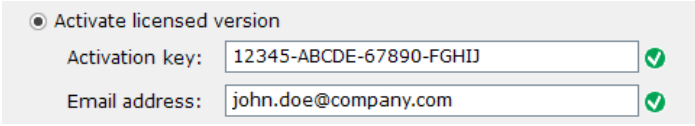
1 Double-click  .

✓ The **EVIDAS activation** dialog box is displayed.

2 Select **Activate licensed version**.



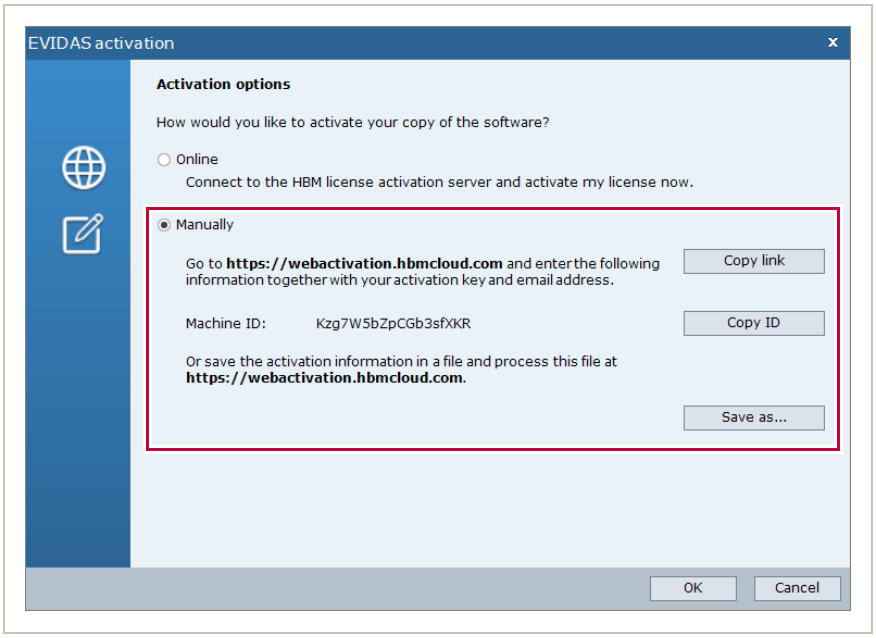
3 Enter your activation key and email address.



4 Click **Next**.

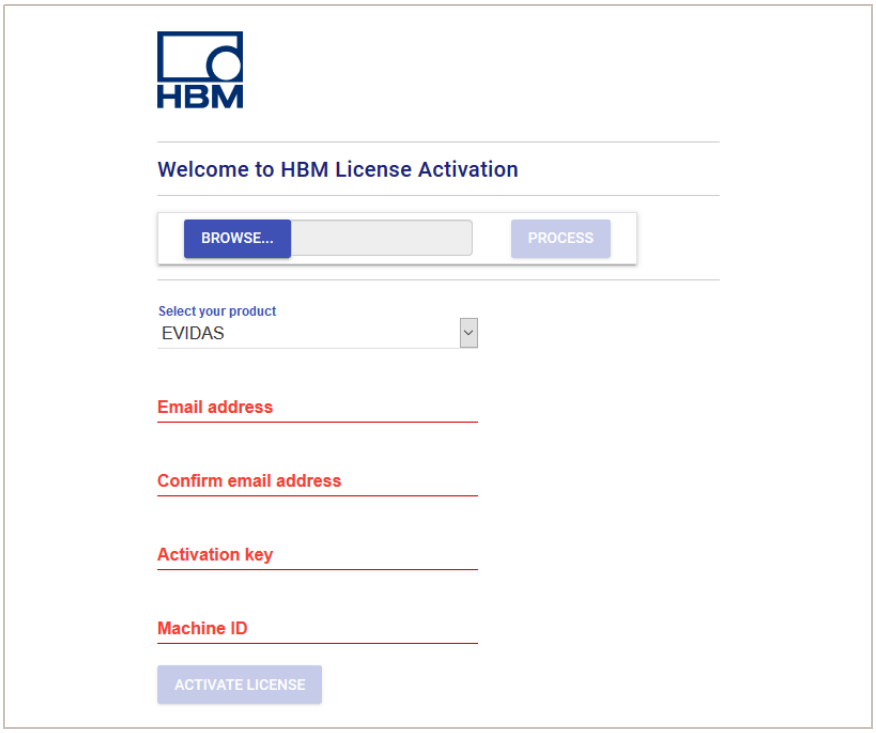
✓ The EVIDAS activation options are displayed.

5 Select **Manually**.



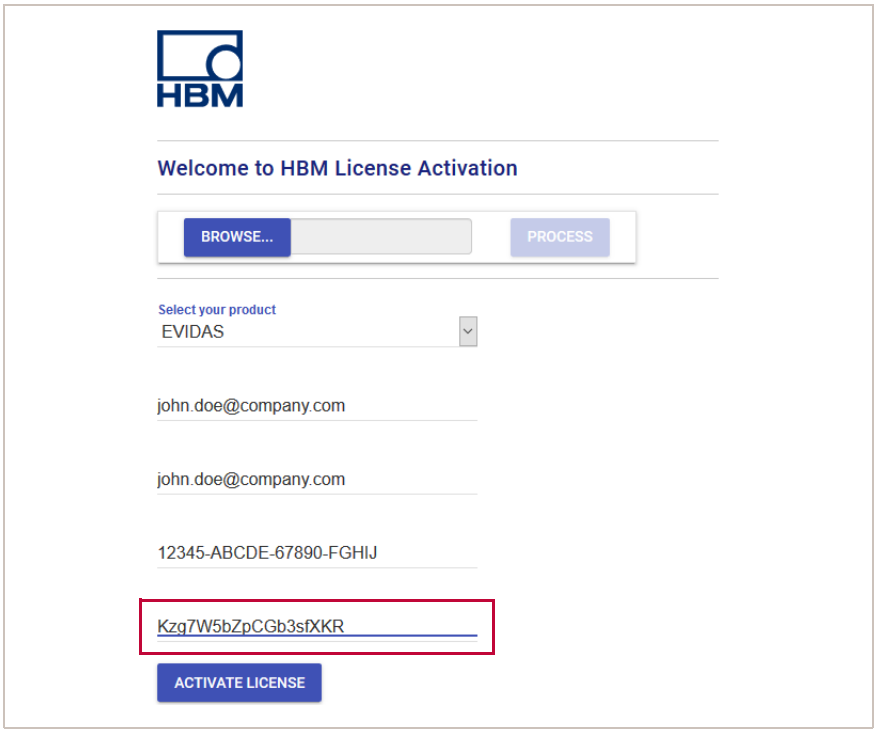
6 Click **Copy link** and paste the URL to the address bar of your web browser.

✓ The **HBM License Activation** webpage is displayed.



7 Enter your email address and activation key.

8 In EVIDAS, in the **Activation options** dialog box, click **Copy ID** and paste it to the **Machine ID** text box on the **HBM License Activation** webpage.



ⓘ Alternatively to steps 7 and 8, in **Activation options**, click **Save as** to save the activation data to a file. On the **HBM License Activation** webpage, click **BROWSE** and navigate to the file you created, and then click **PROCESS** to fill in the activation data from the file.

9 Click **ACTIVATE LICENSE**.

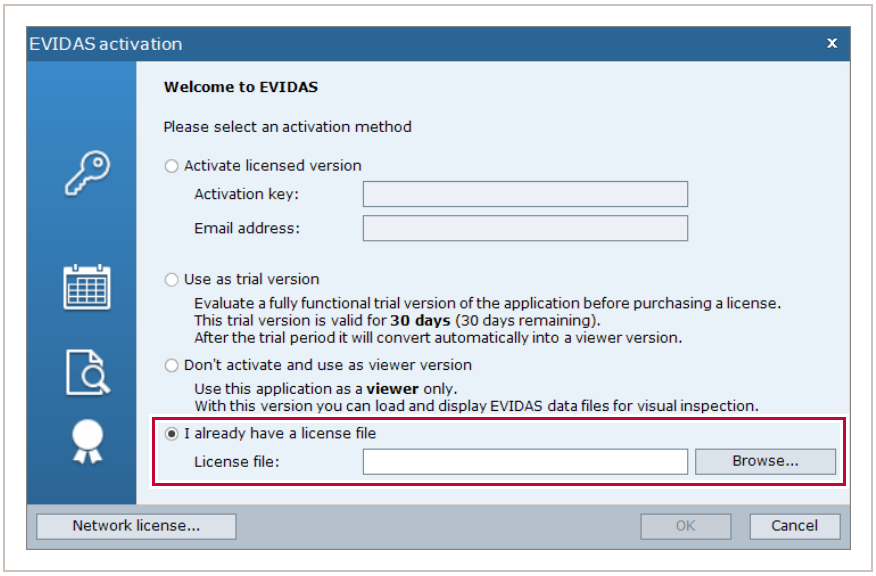
ⓘ You will receive an email with a license file.

10 Copy the license file to a folder on the offline computer.

11 On the offline computer, double-click .

✓ The **EVIDAS activation** dialog box is displayed.

12 Select **I already have a license file** and navigate to it.



13 Click **OK**.

ⓘ Please wait until notified that EVIDAS is activated.

1.2.3 Registering for an EVIDAS network license

If your organization has installed EVIDAS network licenses on a license server, use this procedure to register for using a network license.

You can then use EVIDAS on your computer whenever at least one of the network licenses is available. If you quit EVIDAS, the license is returned to the pool of network licenses and available to another authorized user.

If you plan to work away from your network, you can request to borrow a network license and take it with you for a limited period of time (Borrowing an EVIDAS network license, 37).

Prerequisites

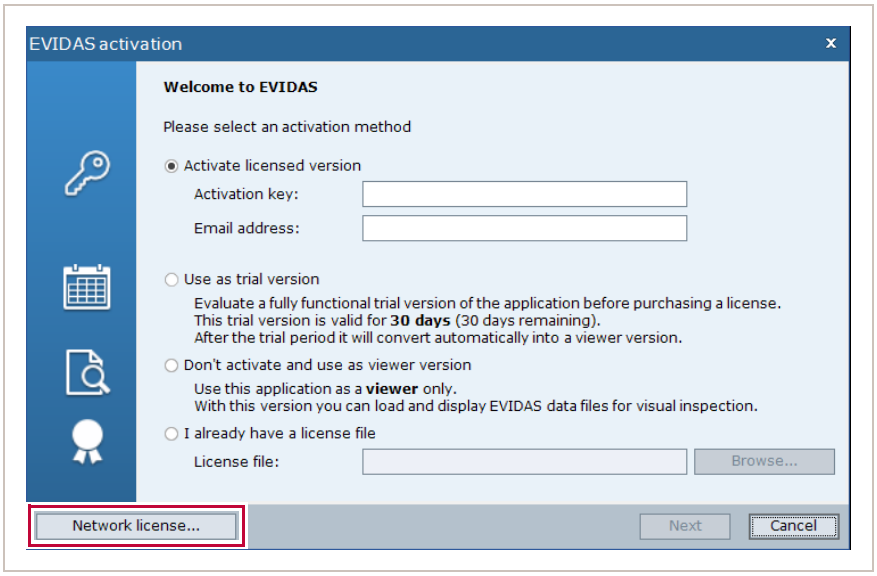
- EVIDAS is downloaded and installed on your computer (Installing EVIDAS, 11).
- EVIDAS network licenses are installed on a license server.
- Your network administrator provided you with the name and port of the license server.
- Network connection.

To register for an EVIDAS network license

1 Double-click .

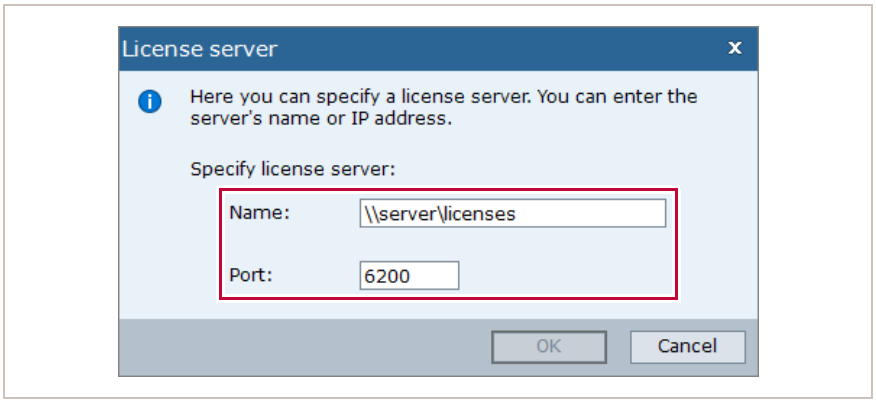
✓ The **EVIDAS activation** dialog box is displayed.

2 Click **Network license**.



✓ The **License server** dialog box is displayed.

3 Enter the name and port of the license server.



ⓘ If you do not know the name and port of the license server, contact your network administrator.

4 Click **OK**.

✓ You are now registered to use an EVIDAS network license.

1.2.4 Using a trial version of EVIDAS

Use this procedure to test a full version of EVIDAS for 30 days.

Prerequisites

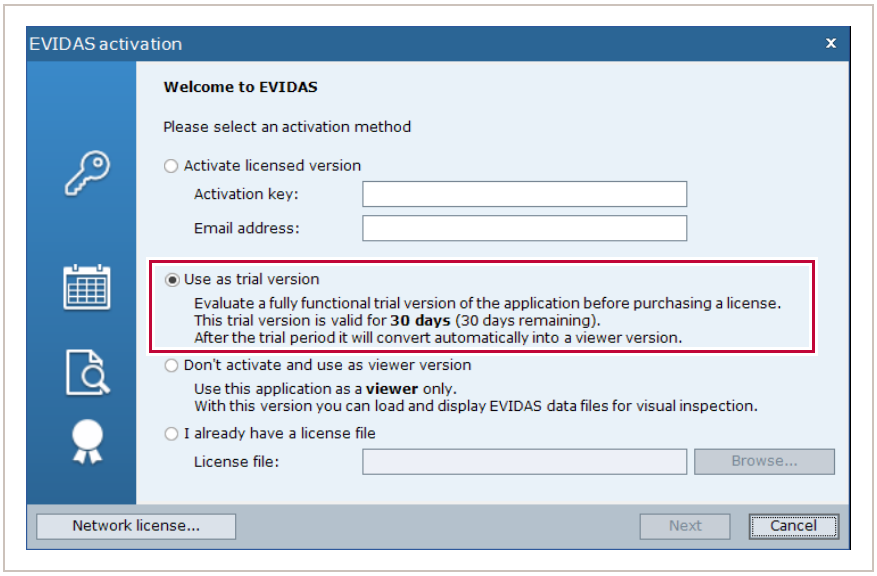
- EVIDAS is downloaded and installed (Installing EVIDAS, 11).

To use a trial version of EVIDAS

1 Double-click  .

✓ The **EVIDAS activation** dialog box is displayed.

2 Select **Use as trial version**.



3 Click **Next**.

✓ A trial version of EVIDAS is started.

ⓘ To activate an EVIDAS single or network license, click **File > Info > Activate software** (File, 219).

1.2.5 Using EVIDAS as a viewer

Use this procedure to activate EVIDAS as a free viewer for data files (*.pnrf).

Prerequisites

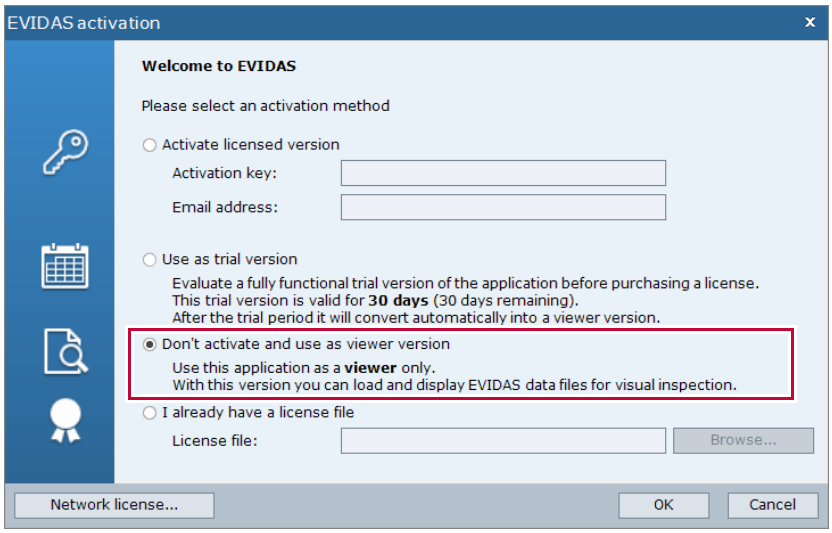
- EVIDAS is downloaded and installed (Installing EVIDAS, 11).

To use EVIDAS as a viewer

1 Double-click  .

✓ The **EVIDAS activation** dialog box is displayed.

2 Select **Don't activate and use as viewer version**.



3 Click **OK**.

✓ EVIDAS is started in viewer mode.

ⓘ To test a full version of EVIDAS for 30 days or to activate an EVIDAS single or network license, click **File > Info > Activate software** (File, 219).

1.3 Activating the HBM cloud

You must activate the HBM cloud (HBM cloud, 208) before you can upload data files to it.

Prerequisites

- EVIDAS is activated (Activating EVIDAS, 12).
- Internet connection.

To activate the HBM cloud

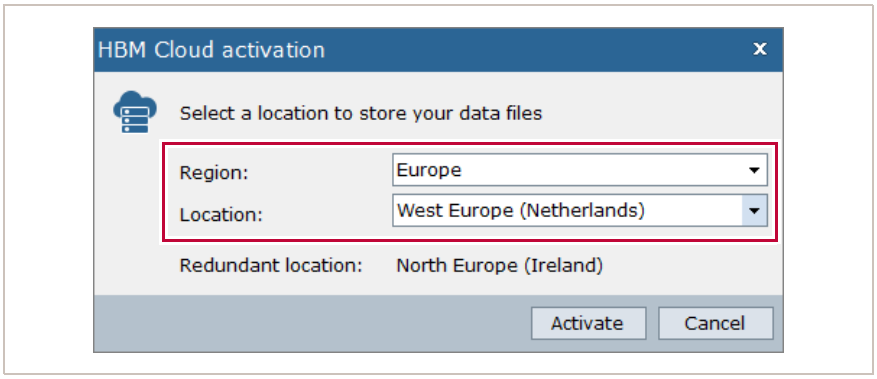
1 Click the **File** tab.

2 Click **Info**.

3 Click **Activate HBM cloud ...**.

✓ The **HBM cloud activation** dialog box is displayed.

4 Select the location you want your data to be stored at, e.g.:



ⓘ A redundant location is specified automatically for the event that the cloud at the prefered location fails, e.g., due to maintenance work. The redundant location holds a backup of your cloud data.

5 Click **Activate**.

✓ The HBM cloud is activated. You receive an email with a link and a password for web login (HBM cloud storage website, 209).

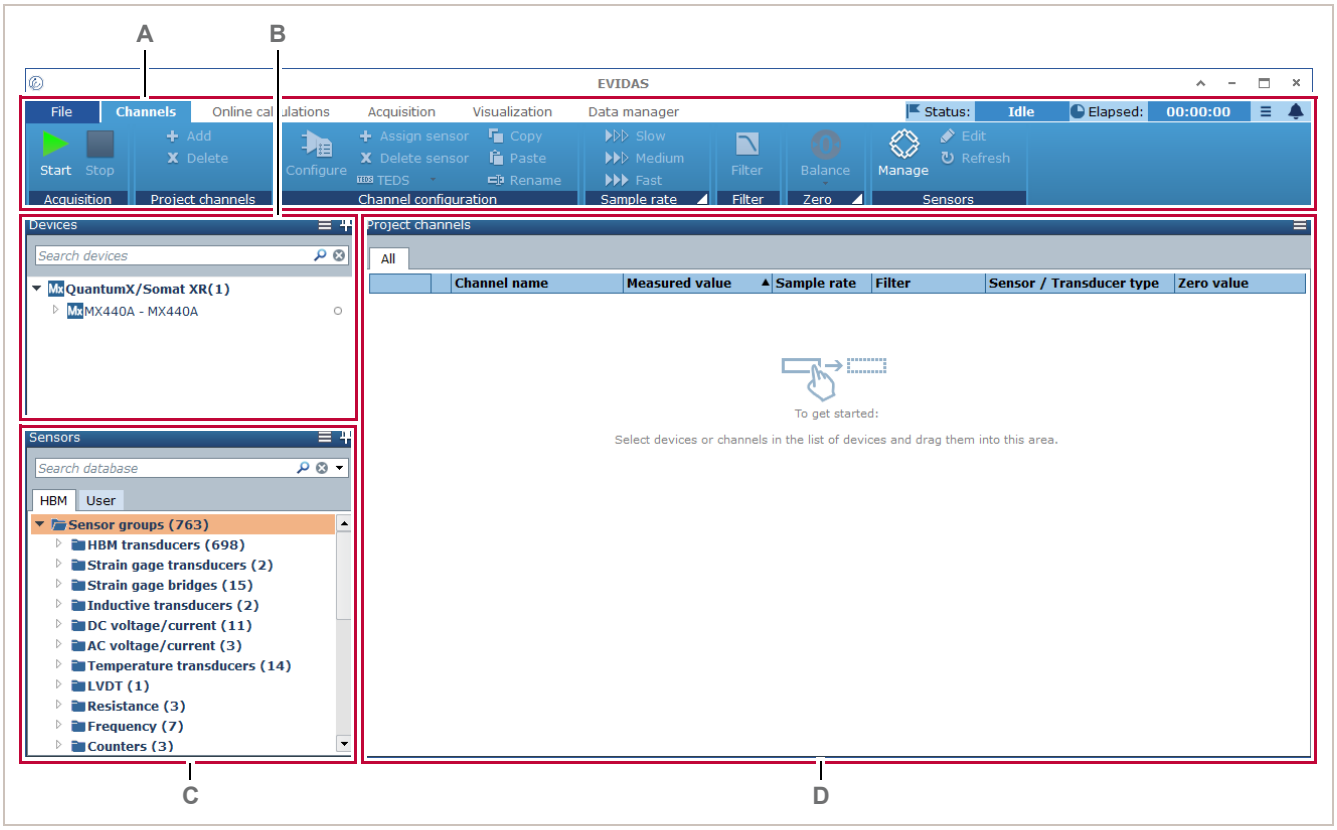
ⓘ You can specify that EVIDAS automatically uploads a copy of each new data file to the HBM cloud (Cloud storage, 138).

1.4 Getting started

To start EVIDAS

1 On the Windows desktop, double-click .

✓ The program is started.



A Ribbon, 39↗

B Devices panel, 104↗

C Sensors panel, 230↗

D Project channels panel, 107↗

ⓘ If you already have an EVIDAS project file (*.evidas), double-click the filename to start EVIDAS and to open the project. The default folder for project files is C:\Users\Public\Documents\HBM\EVIDAS\Projects. Alternatively, use **File > Open** (Opening a project, 71↗).

To visualize a channel

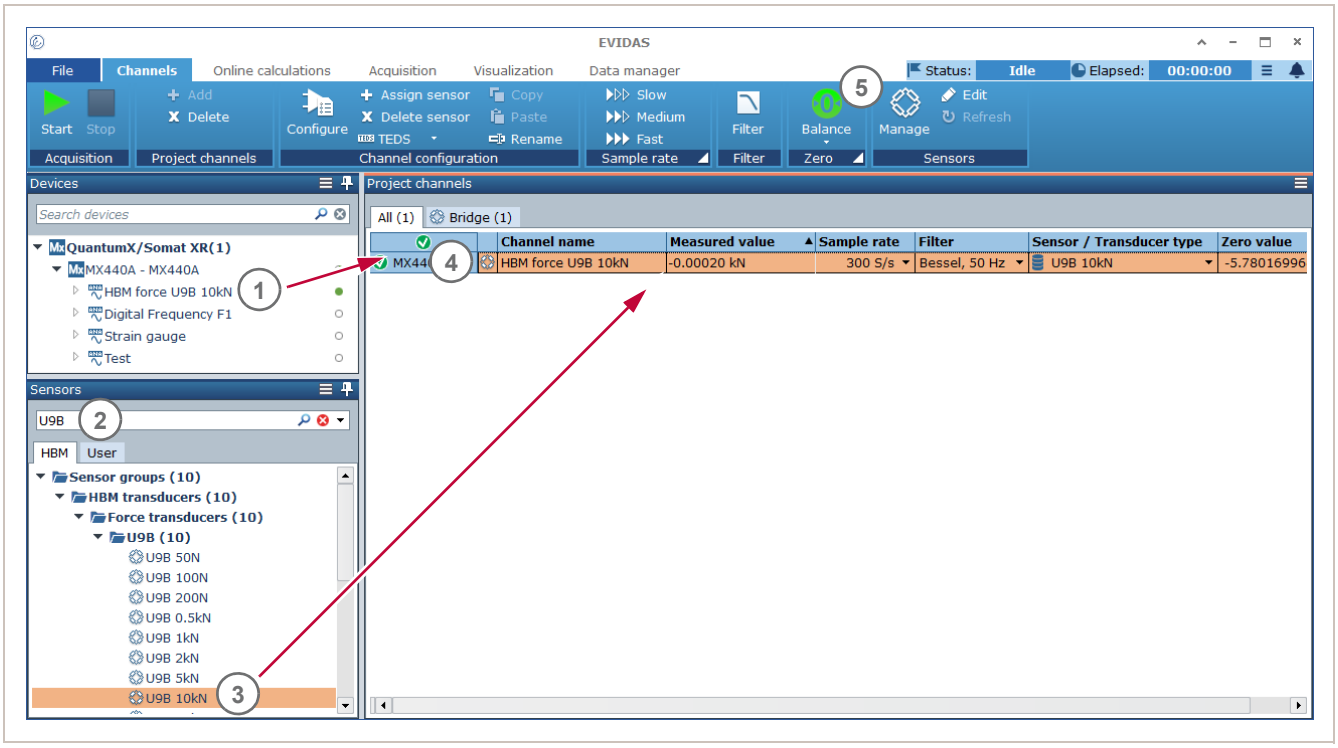
1 In **Devices**, click a channel and drag it to **Project channels**.

2 In **Sensors**, enter the required sensor type into the search box, e.g., **U9B** for a force transducer.

✓ The matching sensors are displayed.

3 Click the appropriate sensor and drag it to the project channel.

✓ The project channel is configured with the settings from the sensor database.



ⓘ In the **Sensor / Transducer type** column, indicates that the settings are taken from the sensor database.

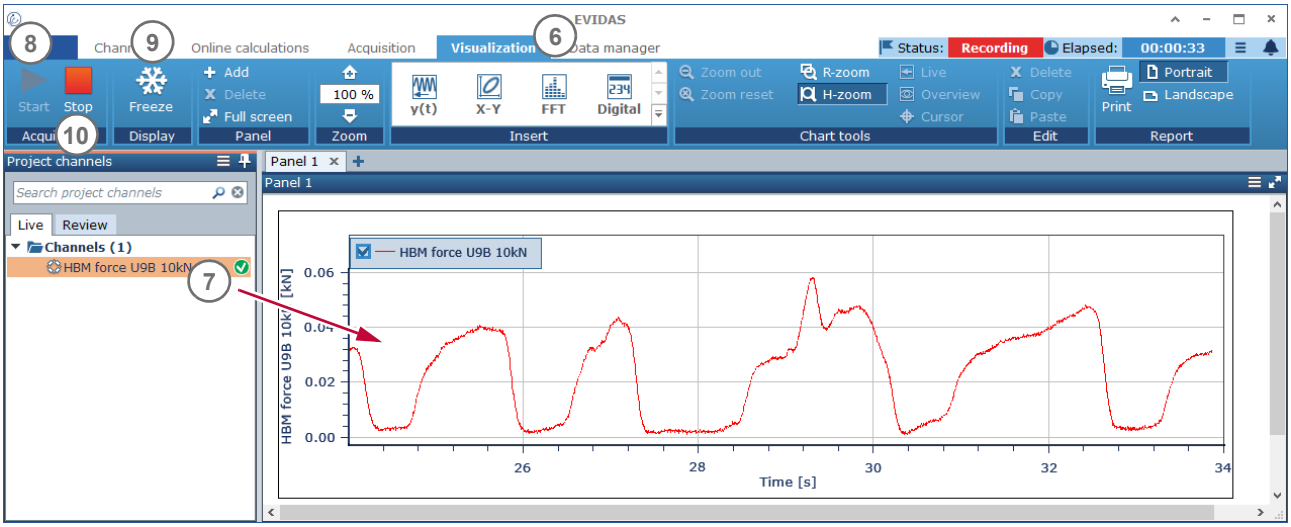
4 Click the project channel.

5 Click **Balance** to determine a zero value for the unloaded sensor.

6 Click the **Visualization** tab.

7 Drag the project channel to **Panel 1** and click, e.g., **Insert y(t) chart**.
✓ An empty chart is displayed.

8 Click **Start**.
✓ The measured values are visualized and saved in a file.

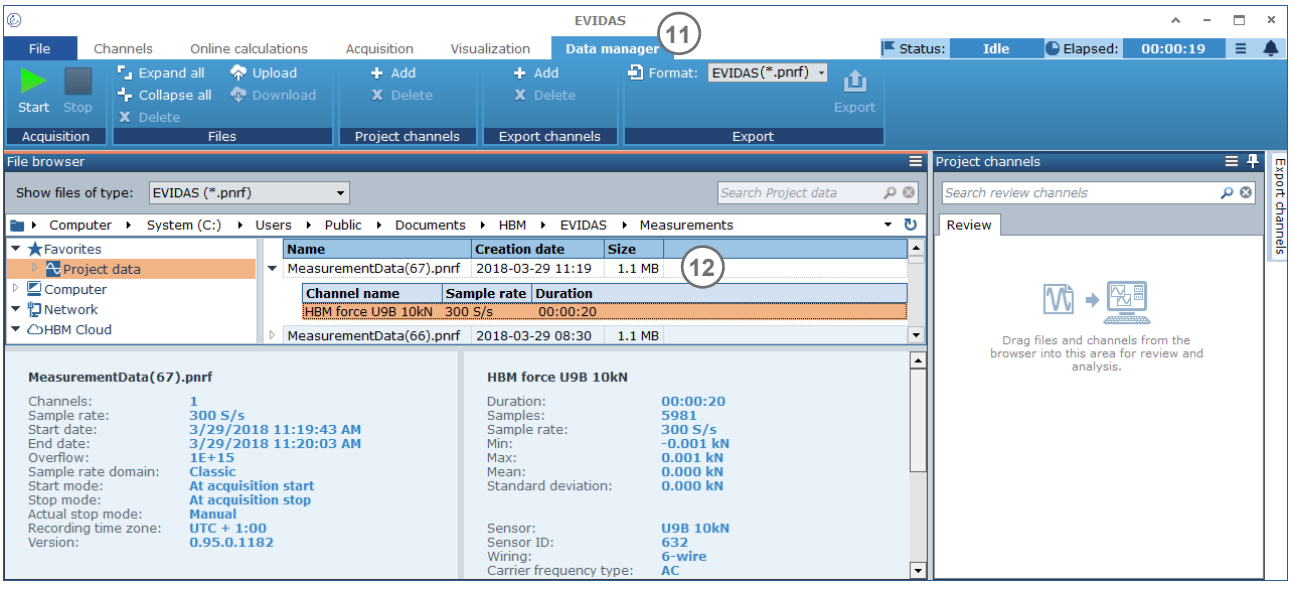


9 Click **Freeze** to pause or continue visualization.

10 Click **Stop** to stop data acquisition.

11 Click the **Data manager** tab.

12 Click the file containing the measured values.
✓ The traceability data of the measurement is displayed below the file list.



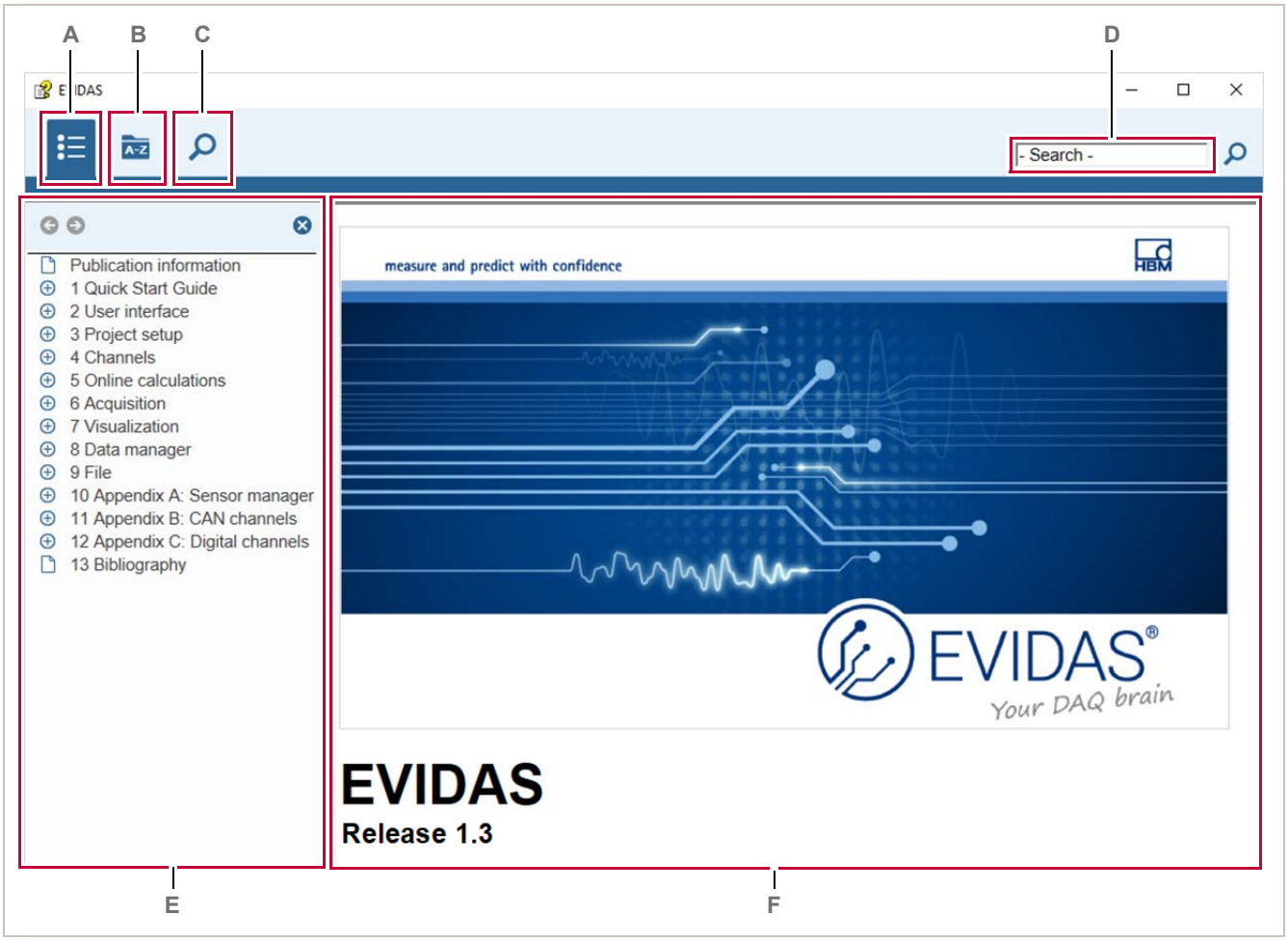
1.5 Getting help

You have the following options to get help on EVIDAS:

Type of Assistance	Description
Online help	Contains a full description of EVIDAS. To open and search the online help, see: Online help, 28 Searching the online help, 32
Printable help (PDF)	Contains a full description of EVIDAS. To open the printable help file, see: Info panel, 222
Support website	To access the support website, see: Info panel, 222

1.5.1 Online help

To open the online help, press F1.



A Table of contents tab

B Index tab

C Search tab

D Search text box

E Navigation panel

F Content panel

☐ EVIDAS online help

The online help contains a full description of EVIDAS.

You can change the size and position of the online help window. These settings are applied the next time you open the online help.

⇨ Getting help on a panel, 29
Getting help on an icon, 30
Getting help on a visualization object, 31
Searching the online help, 32

1.5.2 Getting help on a panel

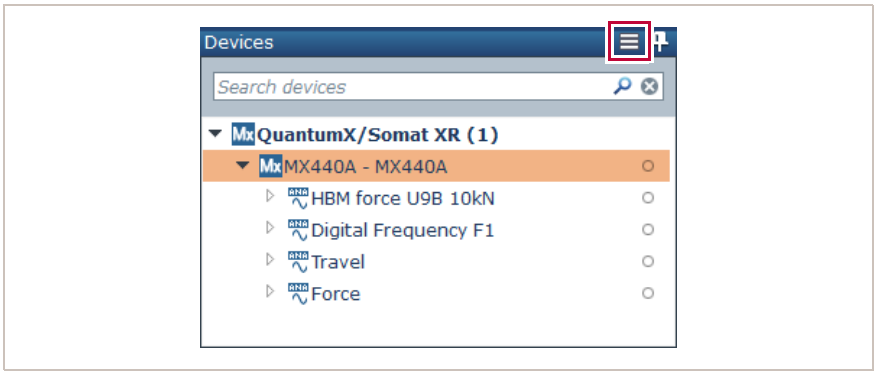
Each panel in EVIDAS has a burger menu, and each burger menu has a link to the online help.

Prerequisites

- EVIDAS is started.

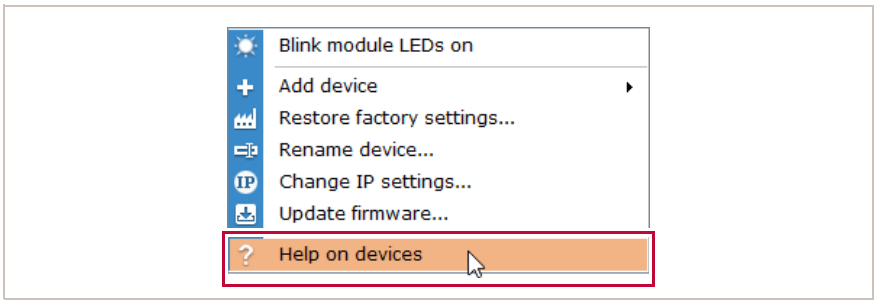
To get help on a panel

1 Click the burger menu button of any panel, e.g., of the **Devices** panel.



✓ The burger menu is displayed.

2 On the burger menu, click the **Help on …** entry.



✓ The online help displays information on the respective panel.

1.5.3 Getting help on an icon

On the ribbon, each icon has a tooltip. Some tooltips have a link to the online help.

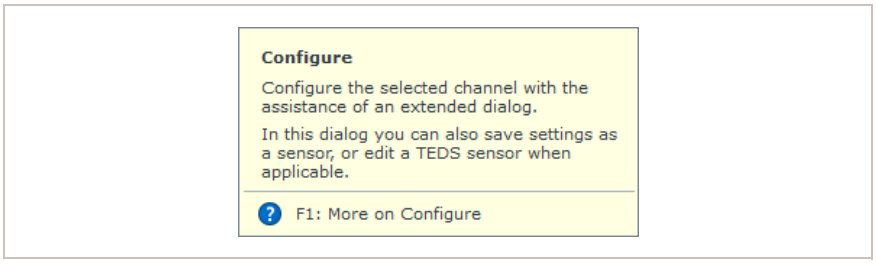
Prerequisites

- EVIDAS is started.

To get help on an icon

1 Point to an icon, e.g., on the **Channels** tab, point to **Configure**.

✓ A tooltip is displayed.



ⓘ If the tooltip contains an **F1: More on …** line, you can get context-sensitive help.

2 While the tooltip is displayed, press F1.

✓ The online help displays information on the respective icon.

1.5.4 Getting help on a visualization object

Each visualization object, e.g., a y(t) chart, offers a link to the online help.

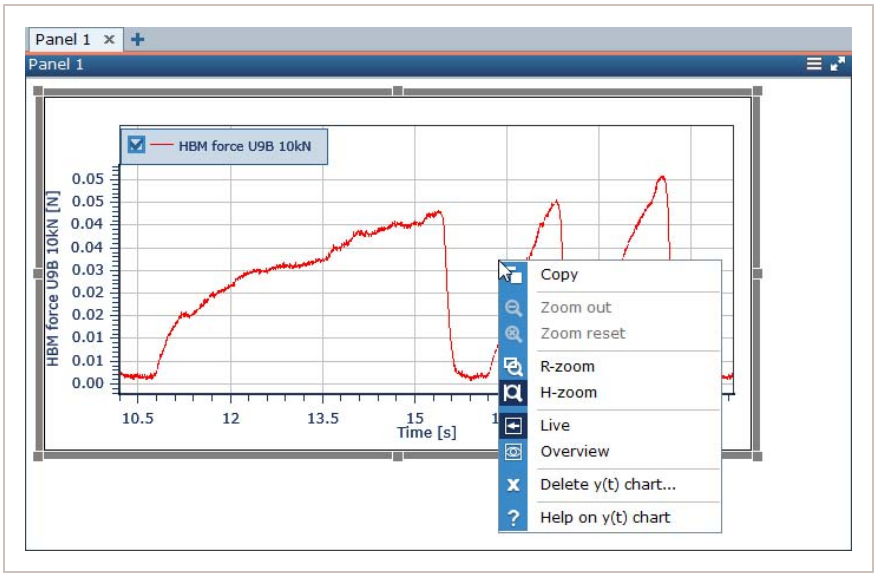
Prerequisites

- EVIDAS is started.

To get help on a visualization object

1 On the visualization panel, right-click an object, e.g., a y(t) chart.

✓ The context menu is displayed.



2 On the context menu, click the **Help on …** entry.

✓ The online help displays information on the contextual tab set of the visualization object.

1.5.5 Searching the online help

Prerequisites

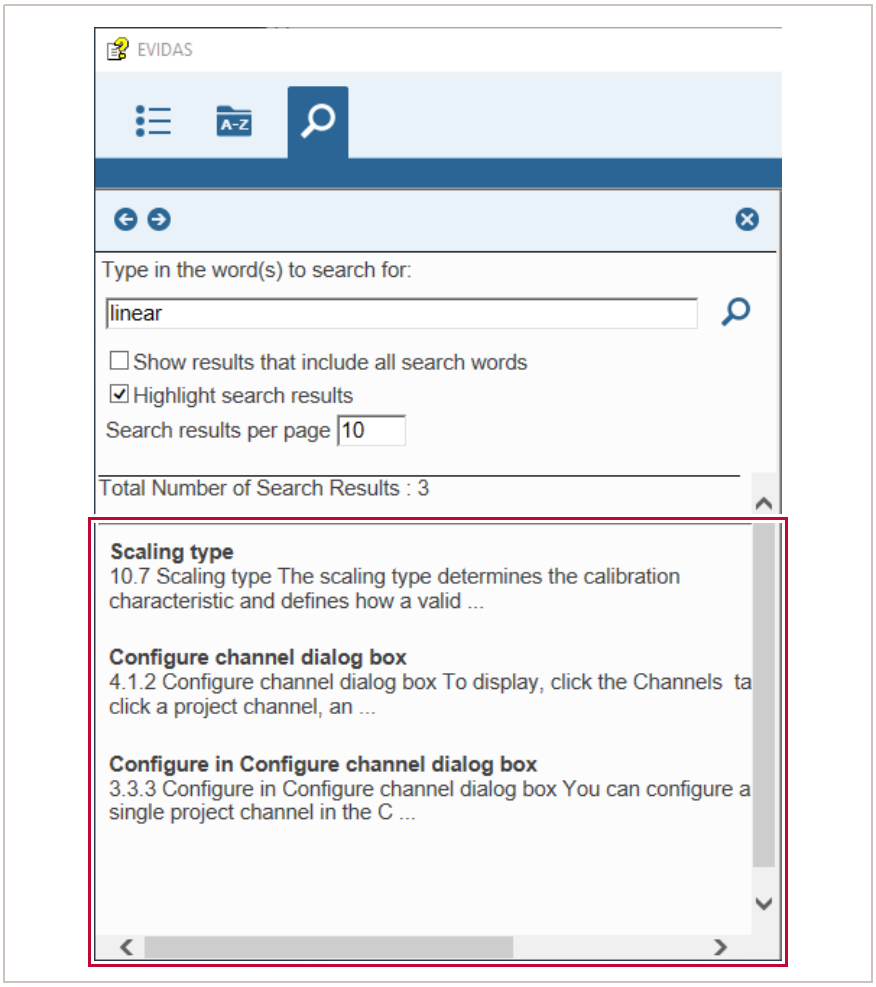
- The EVIDAS online help is displayed.

To search the online help

1 In the search text box, enter a search string, e.g., **linear**.

2 Press Enter.

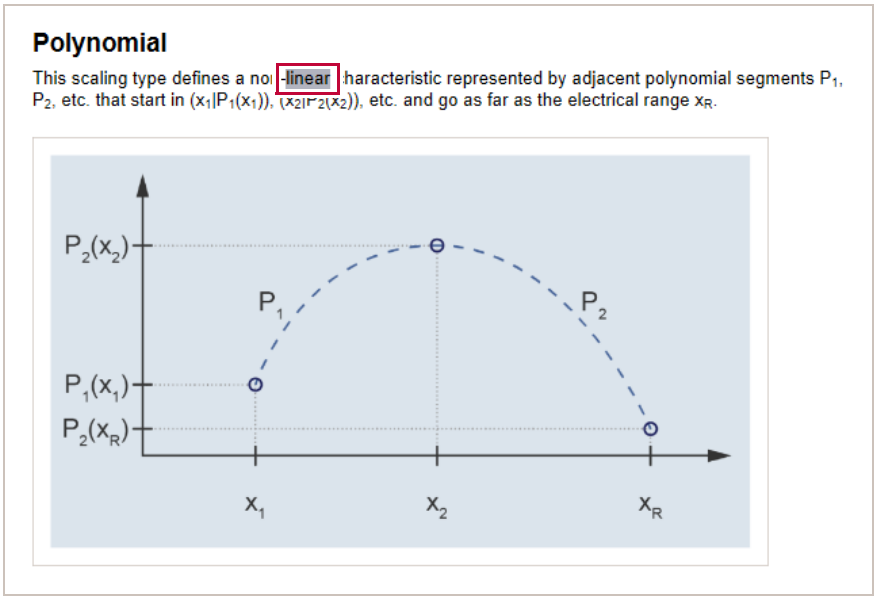
✓ On the navigation panel, the search results are displayed.



ⓘ Topics that contain the search string in their heading are displayed at the top of the list. Headings are displayed in bold print.

3 Click a search result.

✓ The content is displayed and the search string is highlighted.



ⓘ You can search for any string or substring, e.g., **linear**, **linea**, **line**, or **lin**.
To search for a phrase, you must enclose it in quotation marks, e.g., **“non-linear characteristic”**.
If you use several search strings at once, e.g., **background** and **color**, the default search is an OR-search, i.e., all topics that contain either **background** or **color** are search results.
If you want the search results to contain both search strings, select the **Show results that include all search words** option.
You can also use the Boolean operators AND, OR, and NOT in a search string, e.g., **background AND NOT color**.

1.6 Renewing an EVIDAS license

When purchasing an EVIDAS license, you normally receive one year of maintenance with it which includes:

- Updates of EVIDAS.
- Access to the HBM cloud.
- HBM support.

Once the maintenance period expires, you need to purchase an extension of maintenance, e.g., for another year.

After purchasing, you receive an e-mail that contains a new EVIDAS license file (*.lic).

Proceed with one of the following procedures:

- Renewing an EVIDAS single license on an online computer, 35↗
- Renewing an EVIDAS single license on an offline computer, 36↗

1.6.1 Renewing an EVIDAS single license on an online computer

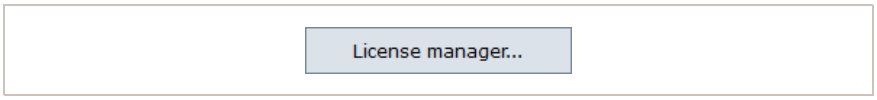
Prerequisites

- EVIDAS is installed and activated (Activating an EVIDAS single license online, 13).
- EVIDAS maintenance extension is purchased.
- Internet connection.

To renew an EVIDAS single license online

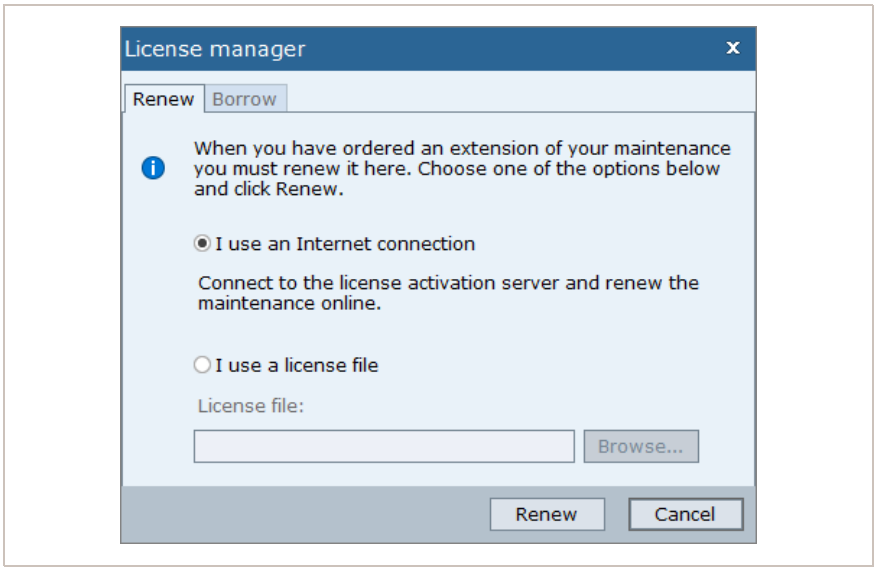
1 Click the **File** tab.

2 On the **Info** panel, click the **License manager** button.



✓ The **License manager** dialog box is displayed.

3 On the **Renew** tab, select **I use an Internet connection**.



4 Click **Renew**.

ⓘ Your computer connects to the EVIDAS license activation center. The EVIDAS license file on your computer is identified and replaced by a new license file specifying the maintenance extension you purchased.

1.6.2 Renewing an EVIDAS single license on an offline computer

Prerequisites

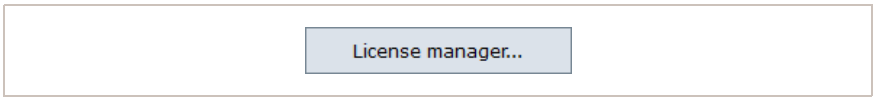
- EVIDAS is installed and activated (Activating an EVIDAS single license manually, 15).
- EVIDAS maintenance extension is purchased.
- Internet connection via a second computer.

To renew an EVIDAS single license offline

1 Copy the EVIDAS license file that you received in the confirmation e-mail to the offline computer.

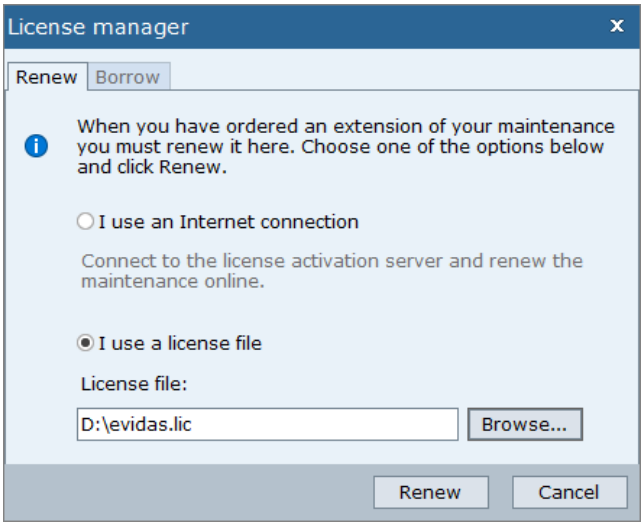
2 Click the **File** tab.

3 On the **Info** panel, click the **License manager** button.



✓ The **License manager** dialog box is displayed.

4 On the **Renew** tab, select **I use a license file** and navigate to the new license file.



5 Click **Renew**.

ⓘ The EVIDAS license file on the offline computer is replaced with the new license file. The new license file specifies your maintenance extension.

1.7 Borrowing an EVIDAS network license

If you use an EVIDAS network license and need to work with EVIDAS while being away from your network, you can borrow a license to take with you.

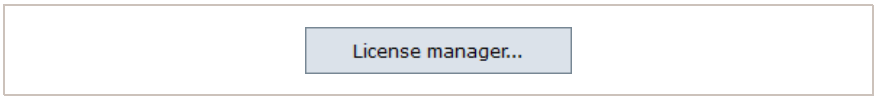
Prerequisites

- You are registered for using an EVIDAS network license (Registering for an EVIDAS network license, 19).

To borrow an EVIDAS network license

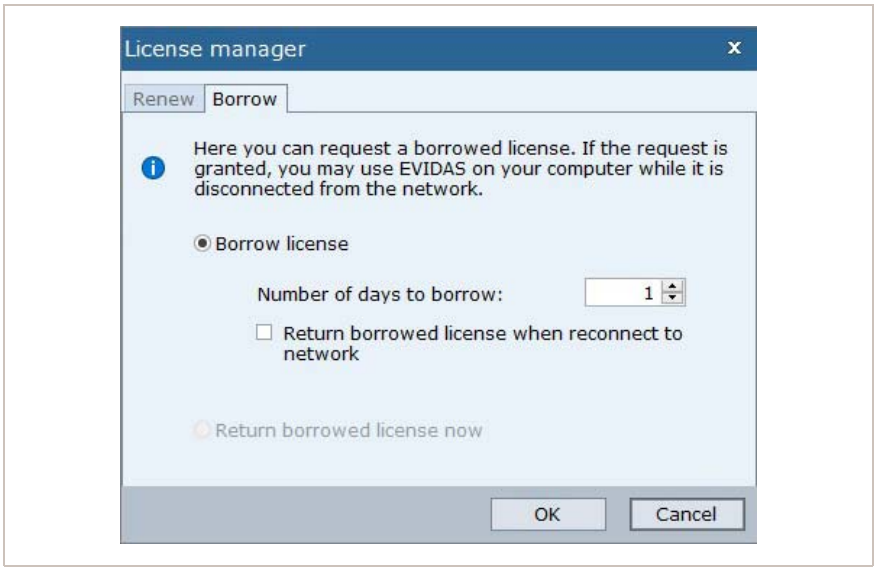
1 Click the **File** tab.

2 On the **Info** panel, click the **License manager** button.



✓ The **License manager** dialog box is displayed.

3 On the **Borrow** tab, select **Borrow license** and the number of days you want to borrow.



4 Select **Return borrowed license when reconnect to network**.

ⓘ Alternatively, you can choose to return a borrowed license manually. In this case, do not select **Return borrowed license when reconnect to network**.
However, when reconnecting to your network, you must then open the **Borrow** tab again, select the **Return borrowed license now** option, and click **OK**.

5 Click **OK**.

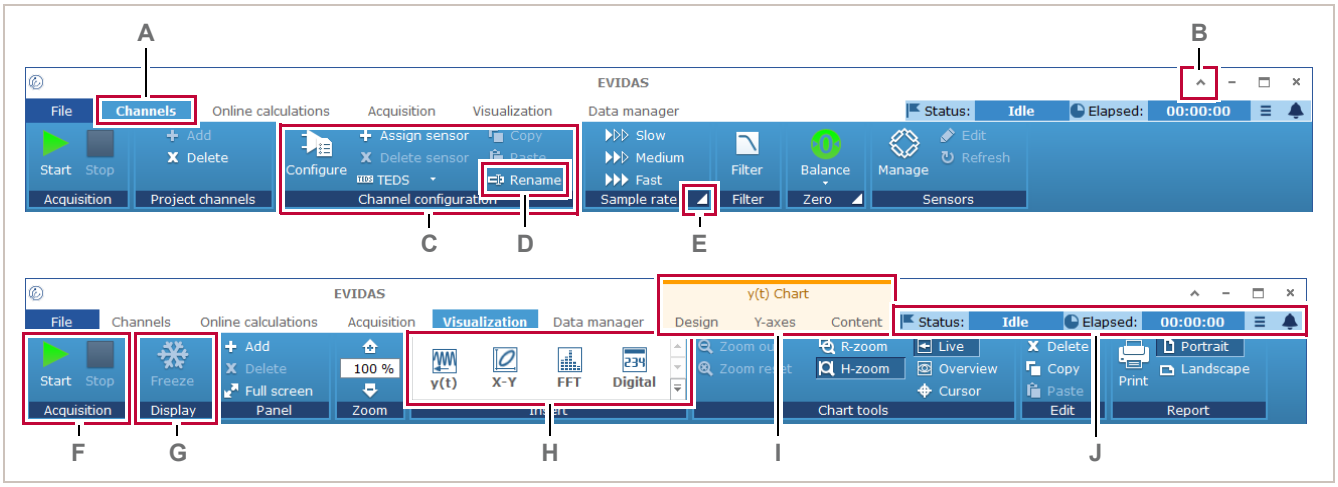
2 User interface

2.1 Ribbon

A ribbon is a command bar that helps you to find and use commands with a minimum number of clicks.

2.1.1 Ribbon elements

The EVIDAS ribbon organizes the program's features into a series of tabs. Each tab contains commands for a specific part of the measuring task. The commands are grouped and labeled.



A Tab, 39
B Collapse / Expand button, 40
C Group, 39
D Command, 40
E Dialog Box Launcher, 40
F Start / Stop button, 40
G Freeze button, 40
H In-Ribbon Gallery, 40
I Contextual tabs, 43
J Status panel, 41

Tab

A tab contains commands for a specific part of the measuring task (Ribbon commands, 41), e.g., for configuring project channels.

The tabs are organized from left to right following the measurement workflow from channel configuration to review and export.

To display a tab, click its label, e.g., **Channels**.

Group

A group is a set of related commands, e.g., to start and stop data acquisition.

Command

A command either performs a specific function directly, e.g., **Delete**, or displays options in a dialog box or menu, e.g., **Rename**.

Frequently used commands have large buttons, e.g., **Start** and **Stop**.

Start / Stop button

The following commands are available on all tabs:

Command	Description
Start	To start data acquisition. EVIDAS will switch to **Visualization** and start recording. If you did not define any visualization objects, EVIDAS will visualize the first four active project channels in y(t) charts, digital meters, and a data table.
Stop	To stop data acquisition.

Freeze button

This command is available on the **Visualization** tab:

Command	Description
Freeze	To pause or continue the visualization of all signals in all charts on the visualization panel, e.g., y(t) charts, X-Y charts, and FFT charts. Meanwhile, data acquisition continues in the background.

Dialog Box Launcher

Click this button to open a dialog box for defining group-related settings, e.g., to assign sample rates to the **Slow**, **Medium**, and **Fast** buttons in the **Sample rate** group.

Collapse / Expand button

Click to collapse the ribbon. Click to expand the ribbon.

Alternatively, press Ctrl+F1 or double-click a tab to collapse and expand the ribbon.

In-Ribbon Gallery

On the **Visualization** tab, the In-Ribbon Gallery displays the available visualization object types.

2.1.2 Ribbon commands

The following links explain the commands on the ribbon:

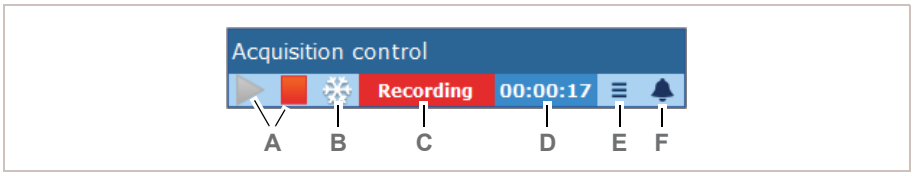
⇨ Channels tab, 74
Online calculations tab, 112
Visualization tab, 146
Data manager tab, 203
Sensor manager tab, 228

2.1.3 Status panel

The status panel is displayed in the right upper corner of the ribbon.

A Acquisition status, 41
B Time count, 42
C Burger menu, 42
D Notifications, 42

On the **Visualization** tab (Visualization commands, 146), if you click **Full screen**, the status panel turns into a floating panel, the **Acquisition control** panel.



A Start / Stop button, 40
B Freeze button, 40
C Acquisition status, 41
D Time count, 42
E Burger menu, 42
F Notifications, 42

To exit full screen mode, click on the visualization panel header or press Esc (Visualization panel, 177).

Acquisition status

Idle	EVIDAS is waiting for the operator to click **Start**.
Zeroing	EVIDAS is zero balancing the project channels before recording (Zero-balance option, 133).
Waiting	EVIDAS is waiting for a start trigger condition to be fulfilled (Trigger options, 129).
Recording	EVIDAS is recording.
Pause	EVIDAS is waiting for a repeated recording to start automatically (Repeat recording options, 133).

Time count

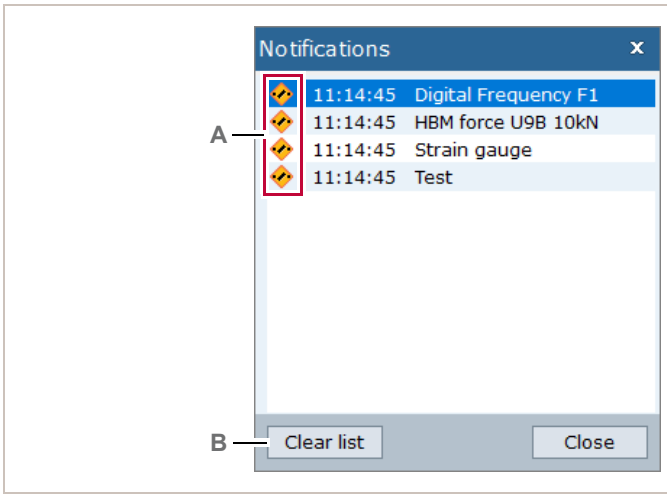
Event	Display
Before recording	00:00:00 (if this is the first recording)
During zero balancing	00:00:00
During recording	Time elapsed since start of recording.
Waiting for repeated recording	Countdown until start of repeated recording.
After recording	Duration of the last recording.

Burger menu

Command	Description
Help on status panel	To access context-sensitive help.

Notifications

To display, click or .



A Signal status icons, 109↗ B Clear list button, 42↗

The **Notifications** dialog box allows you to monitor the connection to the devices during recording. indicates at least one lost connection.

Once you stop the recording, the list of notifications is cleared.

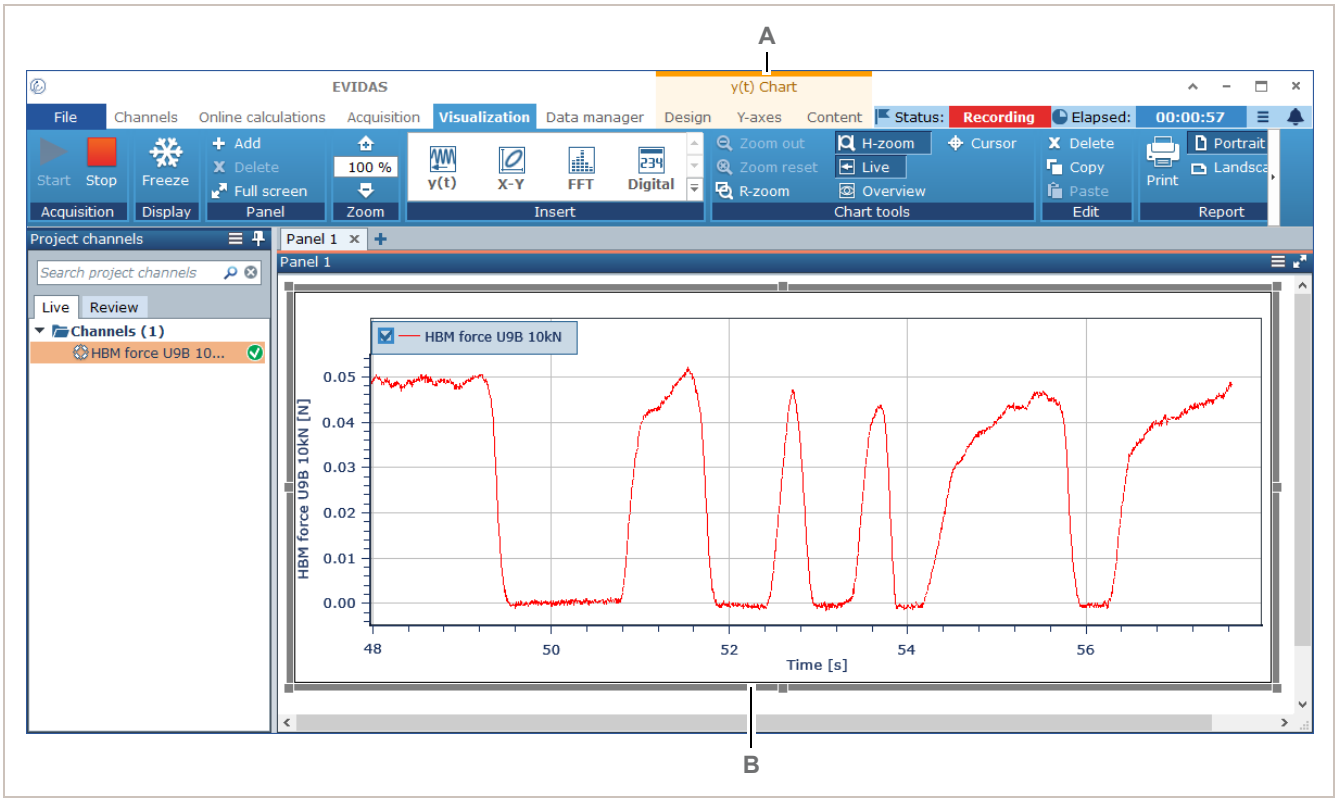
Clear list button

Click this button to clear the list of notifications. The notifications icon is reset to .

⇨ Device lost behavior, 134↗

2.1.4 Contextual tabs

If you click a visualization object, e.g., a y(t) chart, a contextual tab set is displayed (Visualization panel, 177).



A Contextual tab sets, 43 **B** Selected object, 43

Contextual tab sets

EVIDAS has one contextual tab set for each visualization object, e.g., for y(t) charts (Contextual tabs for visualization objects, 150).

The contextual tab sets contain commands to edit the visualization objects.

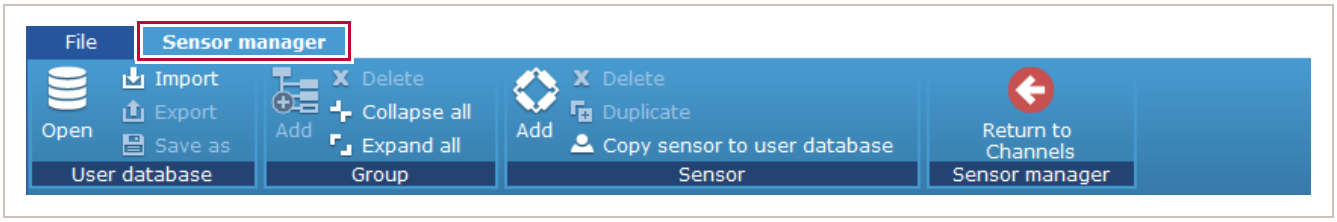
Selected object

To select a visualization object, click the object, e.g., a digital meter on the visualization panel.

The object is displayed with a grey frame and can be configured with the commands on the contextual tab.

2.1.5 Modal tabs

EVIDAS has one modal tab, the **Sensor manager** tab. While working on this tab, you cannot switch to other tabs.



To display the **Sensor manager** tab, on the **Channels** tab, click **Manage**.

To return to **Channels**, click **Return to Channels**.

⇨ Sensor manager tab, 228

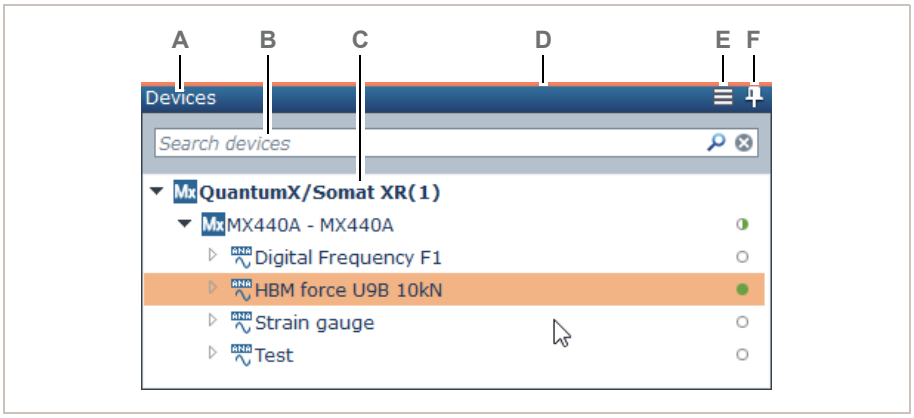
2.2 Panels

For each tab, the area below the ribbon contains one or more panels, e.g., the **Devices** panel, **Sensors** panel, and **Project channels** panel below the **Channels** tab (Channels, 73).

2.2.1 Panel elements

Each panel provides a specific functionality, e.g., to display the available data acquisition devices or to access the sensor database.

A panel has a border, a header, and a content area. Some panels have a search box (Search box, 45). You can resize a panel, change its position in the panel area (Docking, 51), or hide it (Auto Hide, 50).



A Header, 45
B Search box, 45
C Content area, 45
D Active panel bar, 46
E Burger menu button, 46
F Auto Hide button, 46

Header

The panel header has a title and buttons to modify the display of the panel and its contents.

To change the position of a panel, point to the header and drag the panel to the desired position (Docking, 51).

Search box

A search box enables you to search the content of a panel, e.g., the sensor database. As you type a search string, the result is displayed immediately and shows all entries containing the search string (Search box, 47).

Content area

The content area displays the content of the panel, e.g., the data acquisition devices and sensors connected to EVIDAS.

In some panels, a tree view is used to display the content (Tree view, 49).

Active panel bar

The active panel bar is an orange line that is displayed when you click a panel.

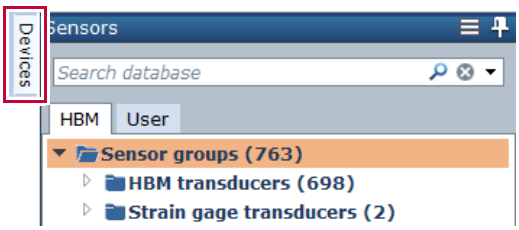
Burger menu button

Click this button to display commands related to the panel, e.g., to add a data acquisition device to the **Devices** panel.

With the burger menu, you can access context-sensitive help.

Auto Hide button

Click this button to hide a panel. A vertical tab is displayed instead, e.g.:

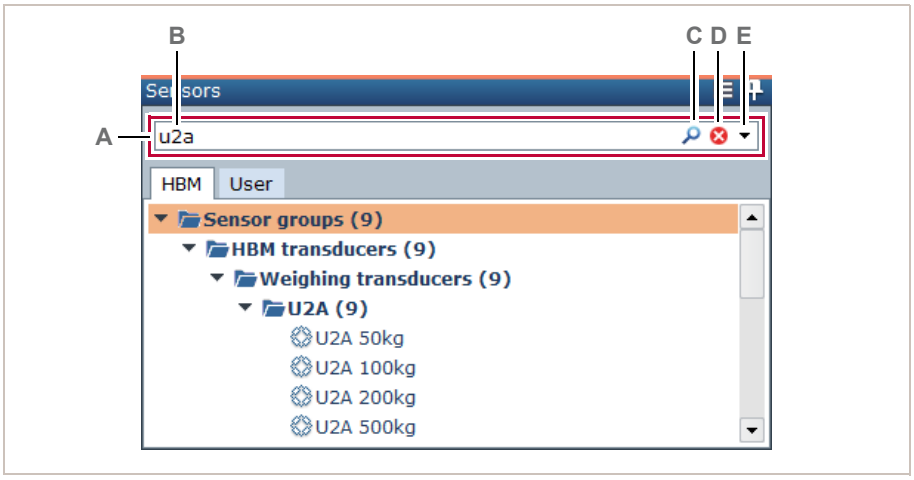


Click the vertical tab to display the panel temporarily.

Click away from the temporary panel to hide it again or click to restore it to its previous size and position (Auto Hide, 50).

2.2.2 Search box

A search box enables you to search the content of a panel, e.g., the sensor database.



A Search box, 47↗
B Search string, 47↗
C Search icon, 47↗
D Delete button, 47↗
E Filter menu button, 48↗

Search box

To filter the content of a panel, enter a search string into the search box.

Search string

As you type the search string, the result is displayed immediately and shows all entries containing the search string, e.g., first for **u**, then for **u2**, then for **u2a**.

Search icon

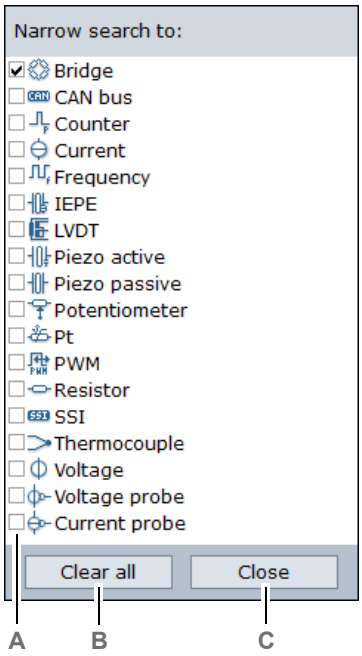
Identifies the text box as a search box.

Delete button

	No search string or selection on filter menu (Filter menu button, 48↗).
	The panel content has been filtered with a search string or with the filter menu (Filter menu button, 48↗). Click this button to delete the search string. However, this does not affect the selection on the filter menu.

Filter menu button

This button is available on the **Sensors** panel and displays the following filter menu that allows you to filter the sensor database prior to applying a search string.



A Filter check boxes, 48

B Clear all button, 48

C Close button, 48

Filter check boxes

To narrow the search, select one or more check boxes.

If you select a check box, the **Delete** button is displayed in red.

Clear all button

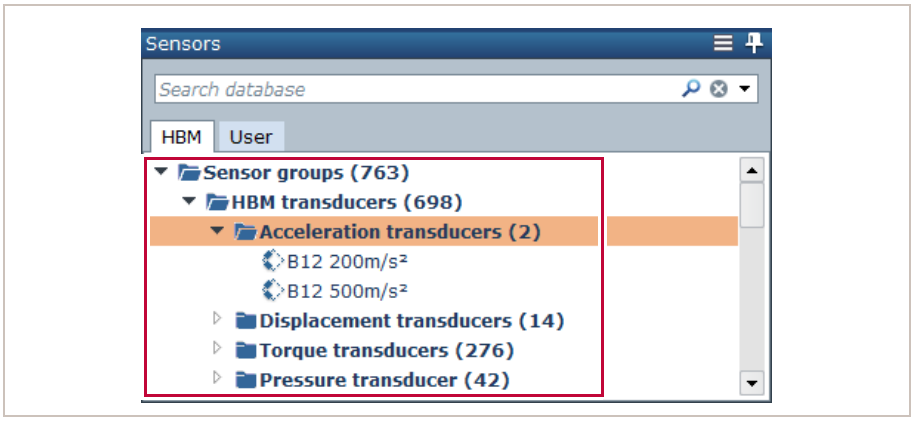
Click this button to clear the selection.

Close button

Click this button to save the selection and close the filter menu.

2.2.3 Tree view

Some panels, e.g., the **Sensors** panel, use a tree view to display their content.



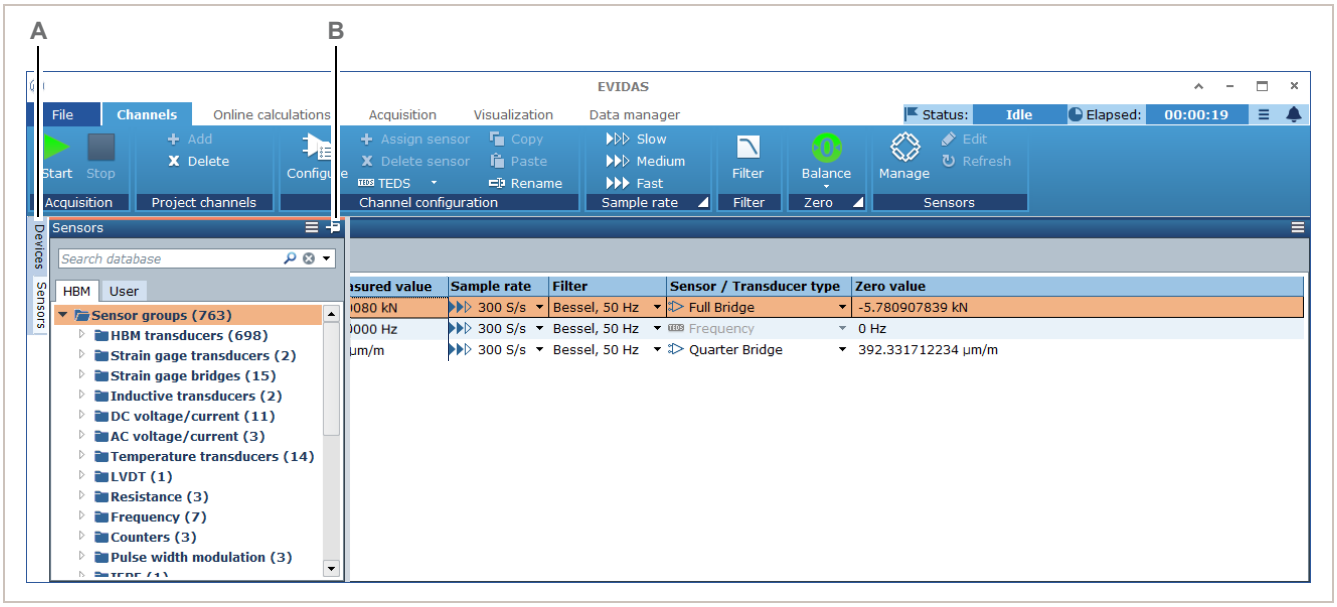
Click the following icons to expand / collapse the folders:

▷	Expand, i.e., display items on level below.
▼	Collapse, i.e., hide items on level below.

Alternatively, double-click the text following the icon.

2.2.4 Auto Hide

You can maximize panels by hiding other ones. You can only hide panels that are docked to one of the edges of the panel area (Docking, 51).



A Vertical tabs, 50 **B** Auto Hide button, 50

Vertical tabs

A vertical tab represents a hidden panel. Click a vertical tab to display the hidden panel temporarily (Auto Hide button, 46).

Click away from the temporary panel to hide it again.

Auto Hide button

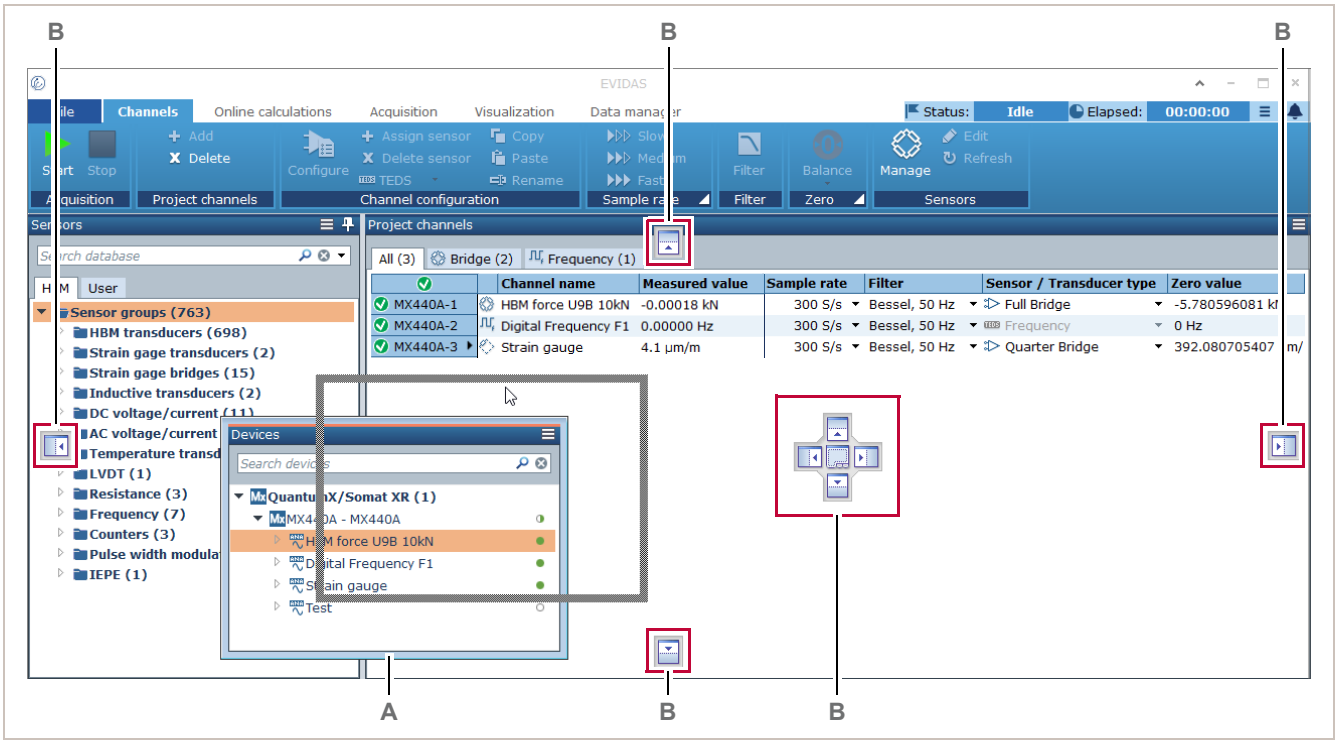
To hide a panel.

To restore a hidden panel to its previous size and position.

2.2.5 Docking

You can undock a panel from its position and dock it to a new one (Rearranging panels, 53). You can only undock and dock panels that are not hidden (Auto Hide, 50).

The following illustration shows the screen elements related to docking:



A Floating panel, 51 **B** Docking buttons, 52

Floating panel

A floating panel is a panel that has been undocked from its position and can be dragged to any position on the screen.

To undock a panel, point to its header and drag the panel from its position.

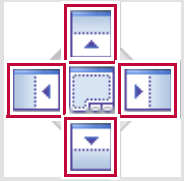
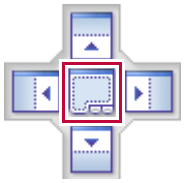
A floating panel can be docked to the following elements:

- The border of the panel area.
- The border of another panel.
- Another panel.

Docking buttons

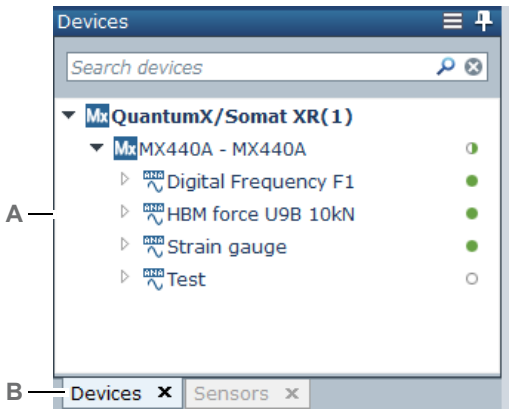
To dock a floating panel, point to its header, drag it from its position, and then point to one of the docking buttons.

A grey frame indicates the docking position related to the button.

Button	Name	Description
	Dock-to-panel-area buttons	Use these buttons to dock a floating panel to one of the edges of the panel area.
	Dock-to-panel buttons	Use these buttons to dock a floating panel to one of the borders of another panel. As you drag a floating panel on top of another panel, these buttons are displayed on the panel below.
	Tabbed-panel button	Use this button to dock a floating panel on top of another panel. A tabbed panel is displayed (Tabbed panel, 52).

Release the pointer to dock the floating panel.

Tabbed panel



A Tabbed panel B Tabs

A tabbed panel contains several panels.

Click a tab to display the respective panel.

You can undock and dock a tabbed panel in the same way as a single panel.

To undock a panel from a tabbed panel, point to its tab and drag the panel from its position.

⇨ Rearranging panels, 53

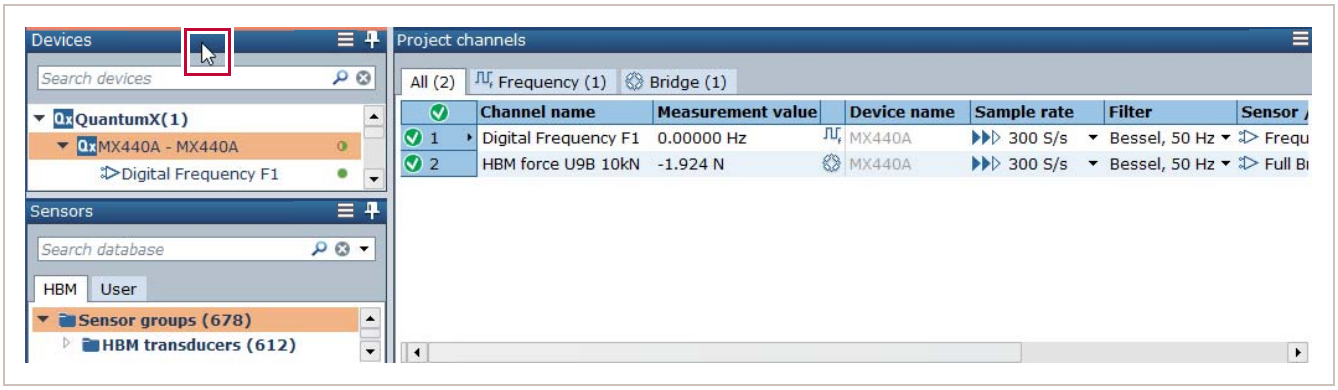
2.2.6 Rearranging panels

Using the panels in **Channels** as an example, this procedure explains how to rearrange panels.

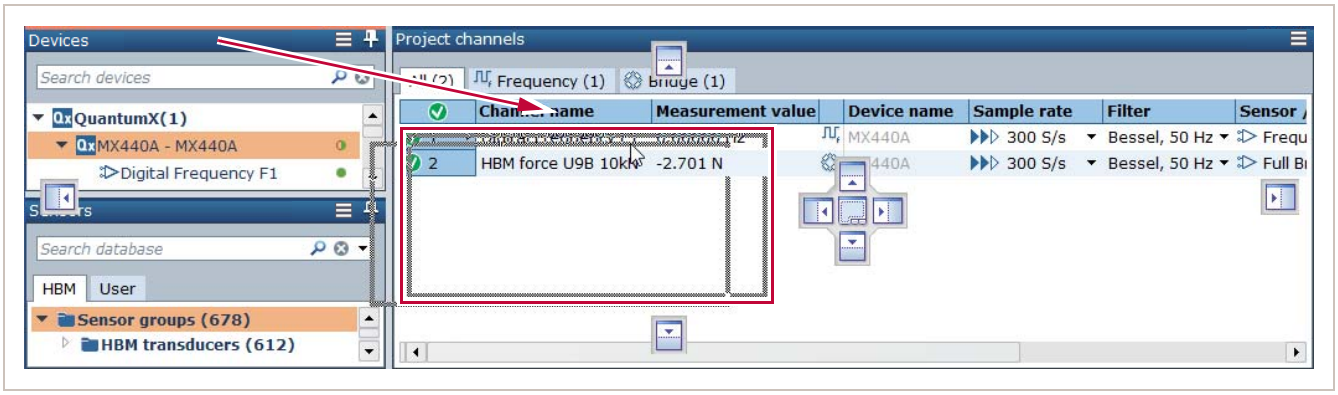
To rearrange panels

1 Click the **Channels** tab.

2 In **Devices**, click the header and hold down the pointer.

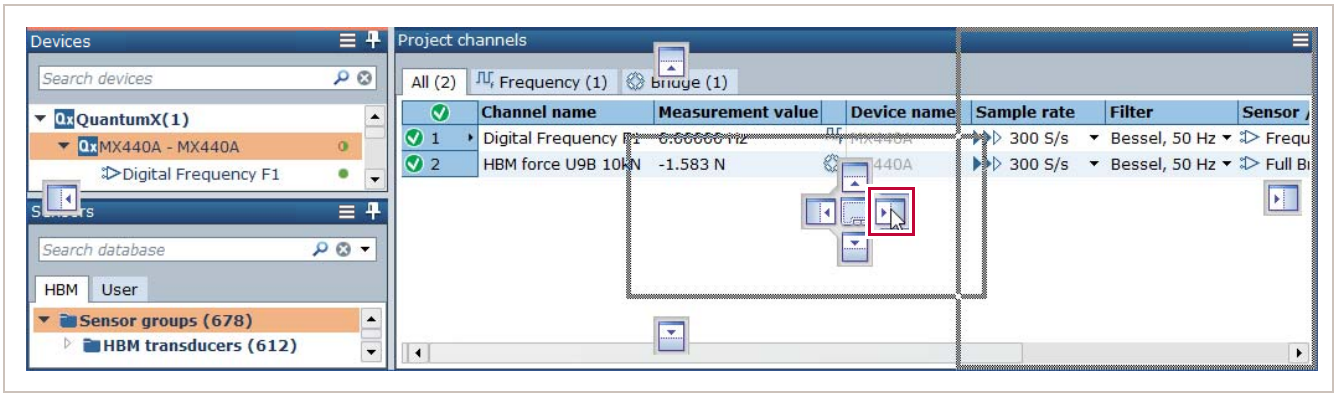


3 Drag the **Devices** panel onto the **Project channels** panel.

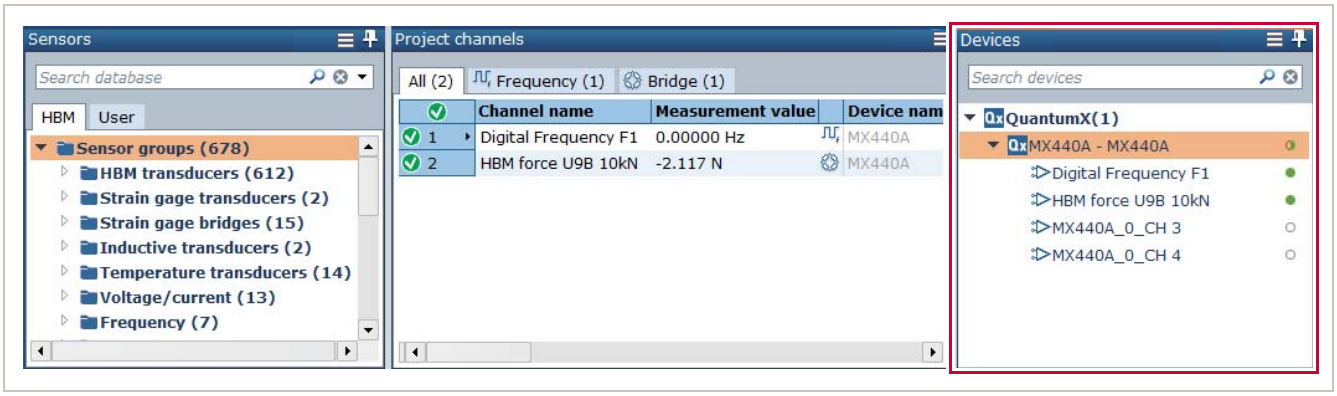


✓ The docking buttons are displayed.

4 Point to the right dock-to-panel button and release the pointer.

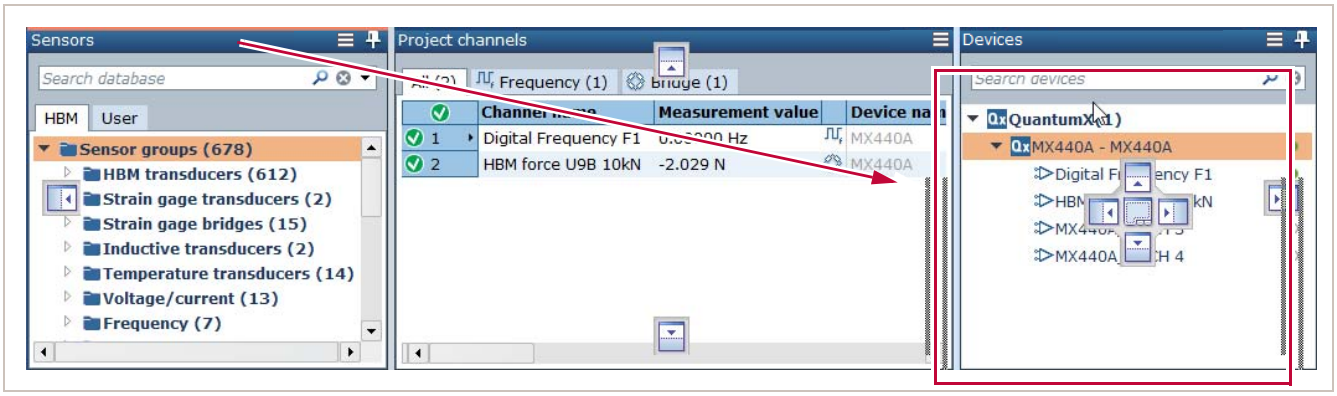


✓ The **Devices** panel is docked to the right of the **Project channels** panel.



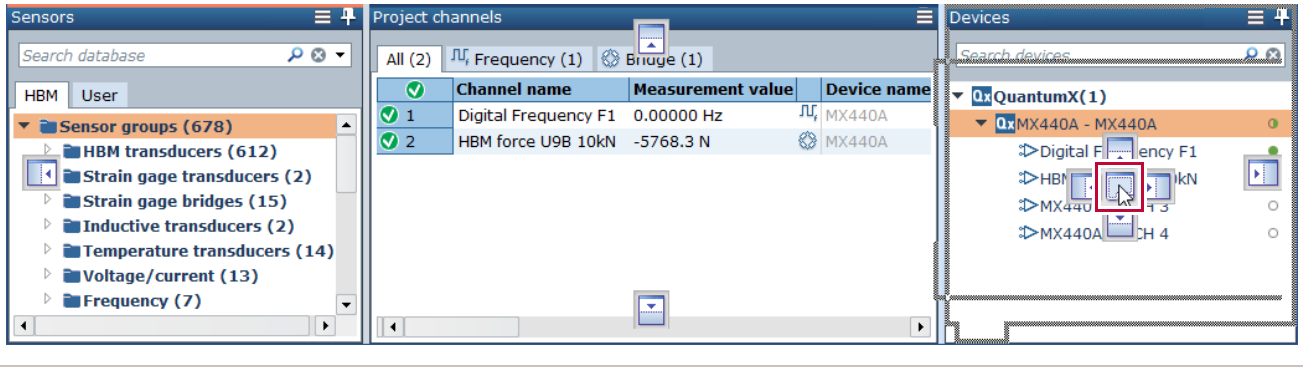
5 In **Sensors**, click the header and hold down the pointer.

6 Drag the **Sensors** panel onto the **Devices** panel.

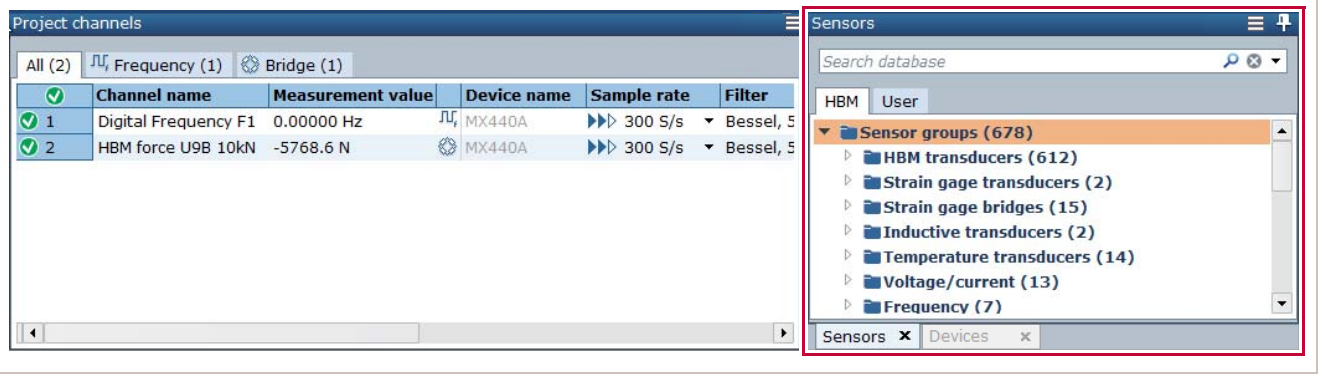


✓ The docking buttons are displayed.

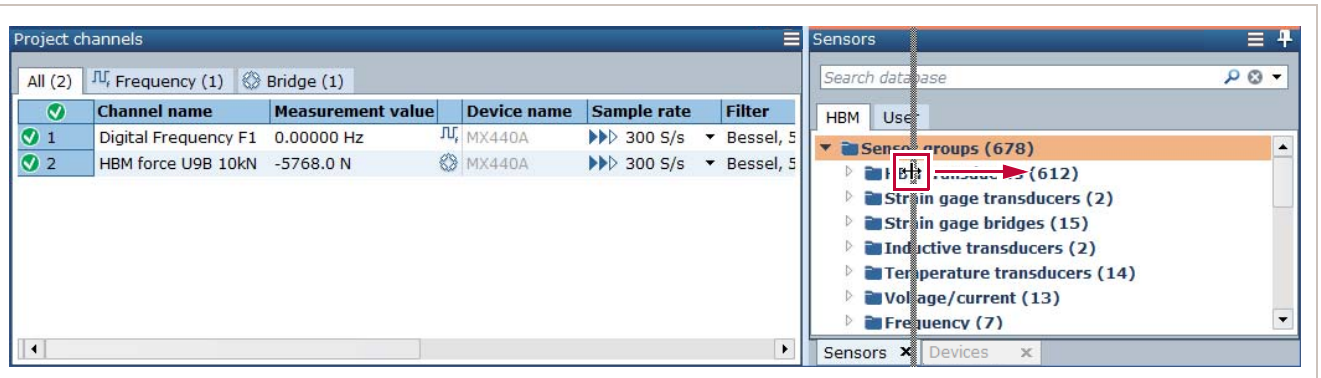
7 Point to the tabbed-panel button and release the pointer.



✓ The **Sensors** panel and the **Devices** panel are grouped together on a tabbed panel.



8 In **Project channels**, point to the right panel border and drag it to the desired position.



3 Project setup

An EVIDAS project is a set of configured channels for a measuring task that is saved in an EVIDAS project file (*.evidas). The default folder for project files is C:\Users\Public\Documents\HBM\EVIDAS\Projects.

To set up and work with a project, you must perform the following steps:

- Starting a new project, 58
- Selecting project channels, 59
- Configuring project channels, 60
- Saving a project, 70
- Opening a project, 71

In addition to the configured channels, the following settings are saved in the project file:

- Calculation channels (Online calculations, 111).
- Recording options (Start options, 127).
- Storage options (Data default folder and file name, 136).
- Visualization items (Visualization, 145).
- Review channels (Project channels panel in Data manager, 210).

However, information about the status of the panels, e.g., the auto hide status (Auto Hide, 50) or docking status (Docking, 51), is not stored in the project file.

3.1 Starting a new project

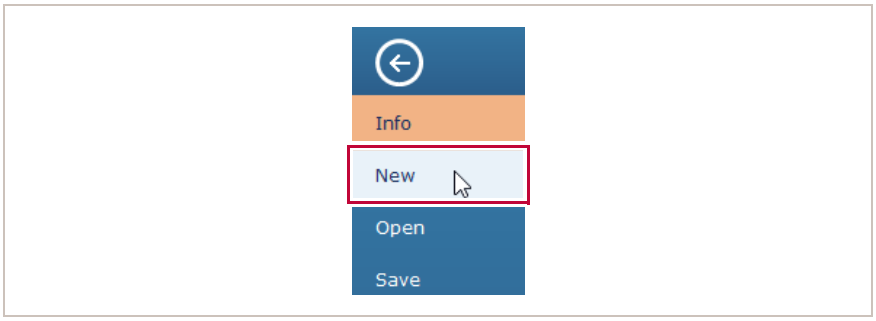
To start a new project, you must clear all current project channels and related settings in EVIDAS.

If you still need the project channels, save them in a project file (To save a project, 70).

To start a project

1 Click the **File** tab.

2 Click **New**.



✓ The current project channels and related settings are deleted.

3.2 Selecting project channels

You can select one channel, a group of channels, or all channels of a data acquistion device as project channels.

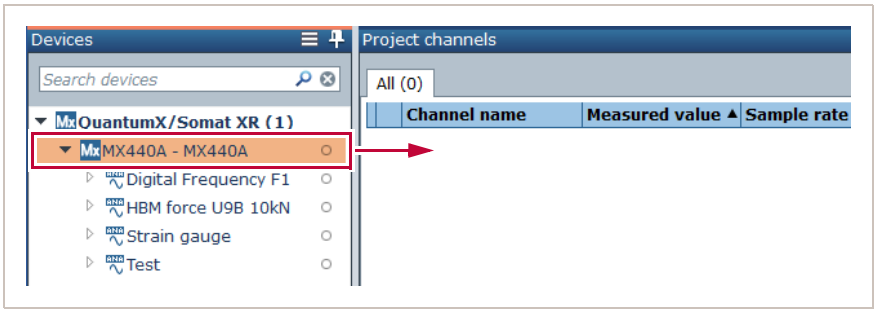
Prerequisites

- Data acquisition device and sensors are connected.
- EVIDAS is started.

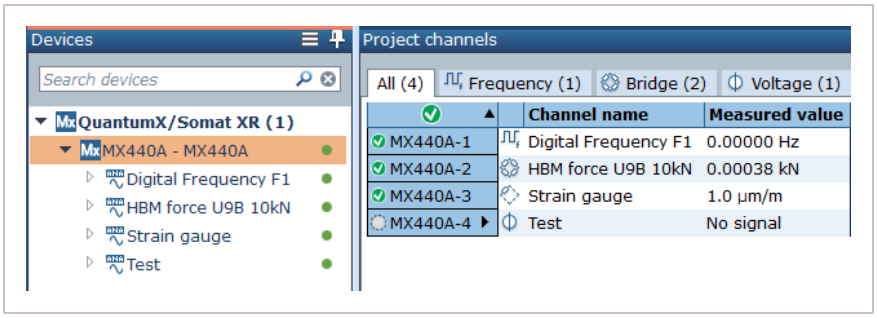
To select project channels

1 Click the **Channels** tab.

2 In **Devices**, click a data acquisition device and drag it to **Project channels**.



✓ All channels of the data acquisition device are project channels.



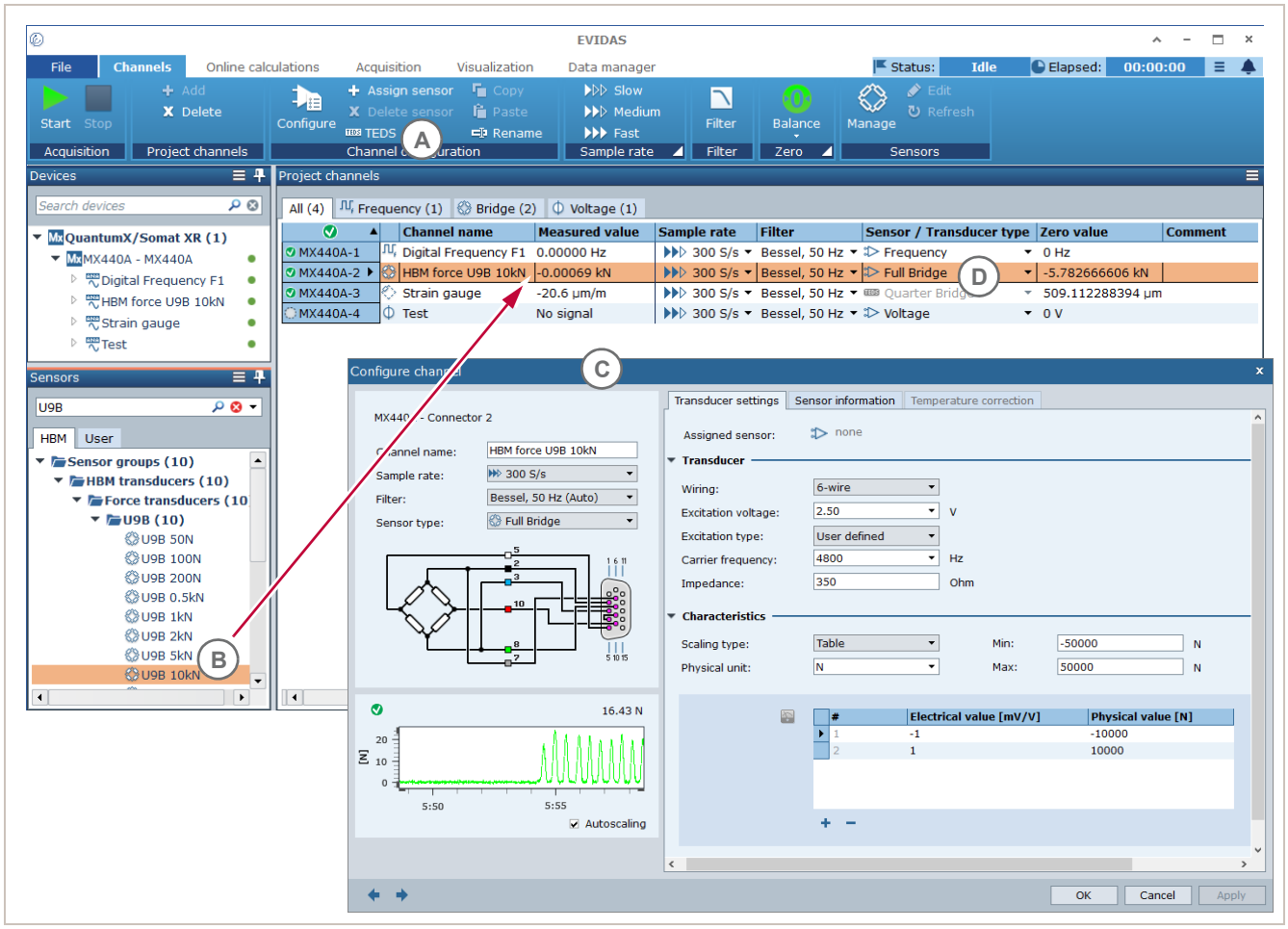
ⓘ In the **Measured value** column, a live representation of the measured values is displayed.

ⓘ The **All** tab displays all project channels. The other tabs display subgroups of channels (Project channels panel, 107).

ⓘ You only want a selection of channels as project channels? In **Devices**, hold down Ctrl while clicking the required channels. Or, hold down Shift while clicking the first and last of a range of channels. Drag the selected channels to **Project channels**.

3.3 Configuring project channels

You have the following options to configure a project channel:



Options to configure project channels

Option	Procedure	Description
A	Configure with TEDS sensor, 61	The project channel is configured automatically with the sensor settings from the TEDS (**T**ransducer **E**lectronic **D**ata **S**heet) chip in the TEDS connector / sensor.
B	Configure with HBM sensor database, 62	The project channel is configured with the default sensor settings from the HBM sensor database. For higher accuracy, you can enter the sensor settings from the sensor data sheet (Configure in Configure channel dialog box, 64).
C	Configure in Configure channel dialog box, 64	You can configure a single project channel in the **Configure channel** dialog box.
D	Configure on Project channels panel, 67	You can configure project channels on the **Project channels** panel.

3.3.1 Configure with TEDS sensor

A TEDS sensor has a connector with a built-in TEDS (**T**ransducer **E**lectronic **D**ata **S**heet) chip that contains the sensor settings.

Prerequisites

- Data acquisition device is connected.

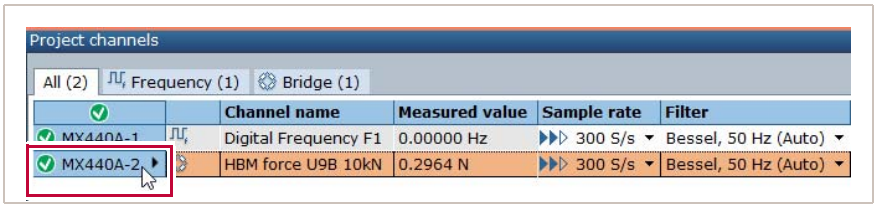
To configure a project channel with a TEDS sensor

1 Connect the TEDS sensor to the data acquisition device.

2 Click the **Channels** tab.

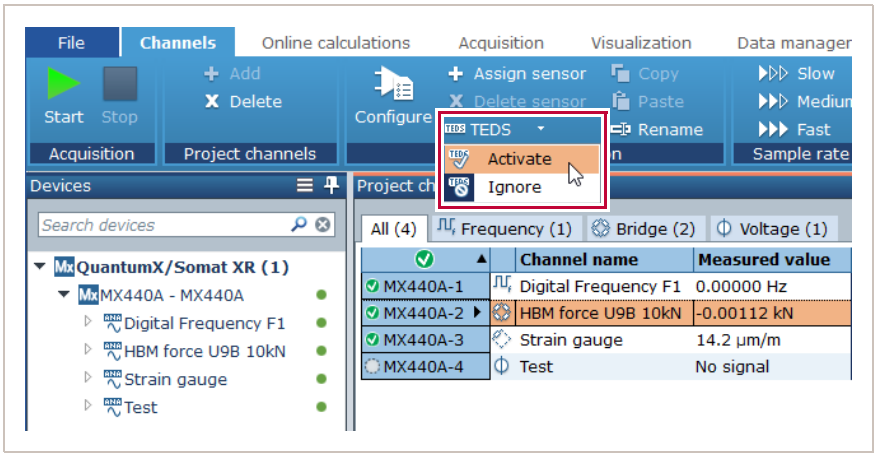
3 In **Devices**, drag the respective project channel to **Project channels**.

4 In **Project channels**, click the channel.



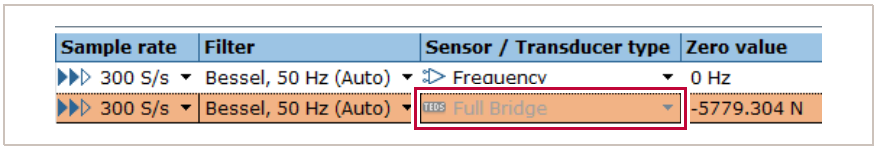
✓ The channel is highlighted.

5 Click TEDS **TEDS**, and then click **Activate**.



✓ EVIDAS scans the selected channel, reads the sensor settings from the TEDS chip, and configures the project channel automatically. Even if you connect the TEDS sensor later, the settings from the TEDS chip will be used.

✓ In **Project channels**, the sensor is identified as TEDS sensor.



⇨ TEDS options, 88

3.3.2 Configure with HBM sensor database

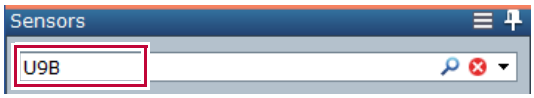
You can configure a channel using the default sensor settings from the HBM sensor database.

Prerequisites

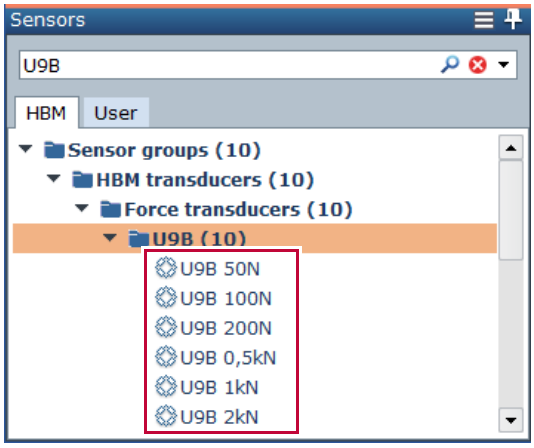
- Data acquisition device and sensors are connected.
- Project channels are selected (To select project channels, 59).

To configure a project channel with the HBM sensor database

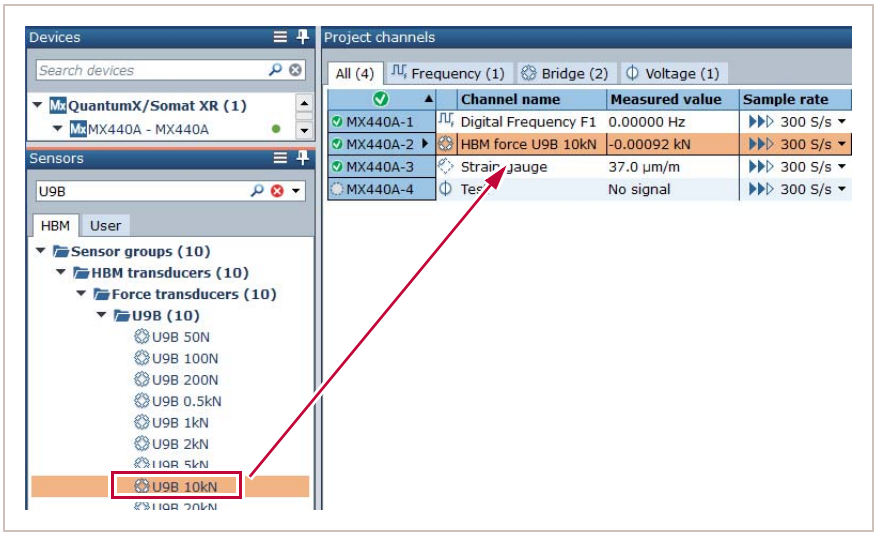
1 In the **Sensors** search box, enter the required sensor type, e.g., **U9B** for a HBM force transducer.



✓ The matching sensors are displayed.

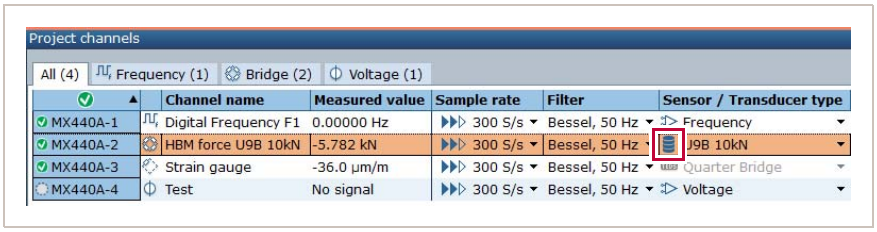


2 Click the appropriate sensor and drag it to its project channel.



✓ The project channel is configured with the settings from the sensor database.

ⓘ In the **Sensor / Transducer type** column, indicates that the settings are taken from the sensor database.



ⓘ For higher accuracy, you can enter the sensor settings from the sensor data sheet (Configure in Configure channel dialog box, 64).

3.3.3 Configure in Configure channel dialog box

You can configure a single project channel in the **Configure channel** dialog box (Configure channel dialog box, 76).

Prerequisites

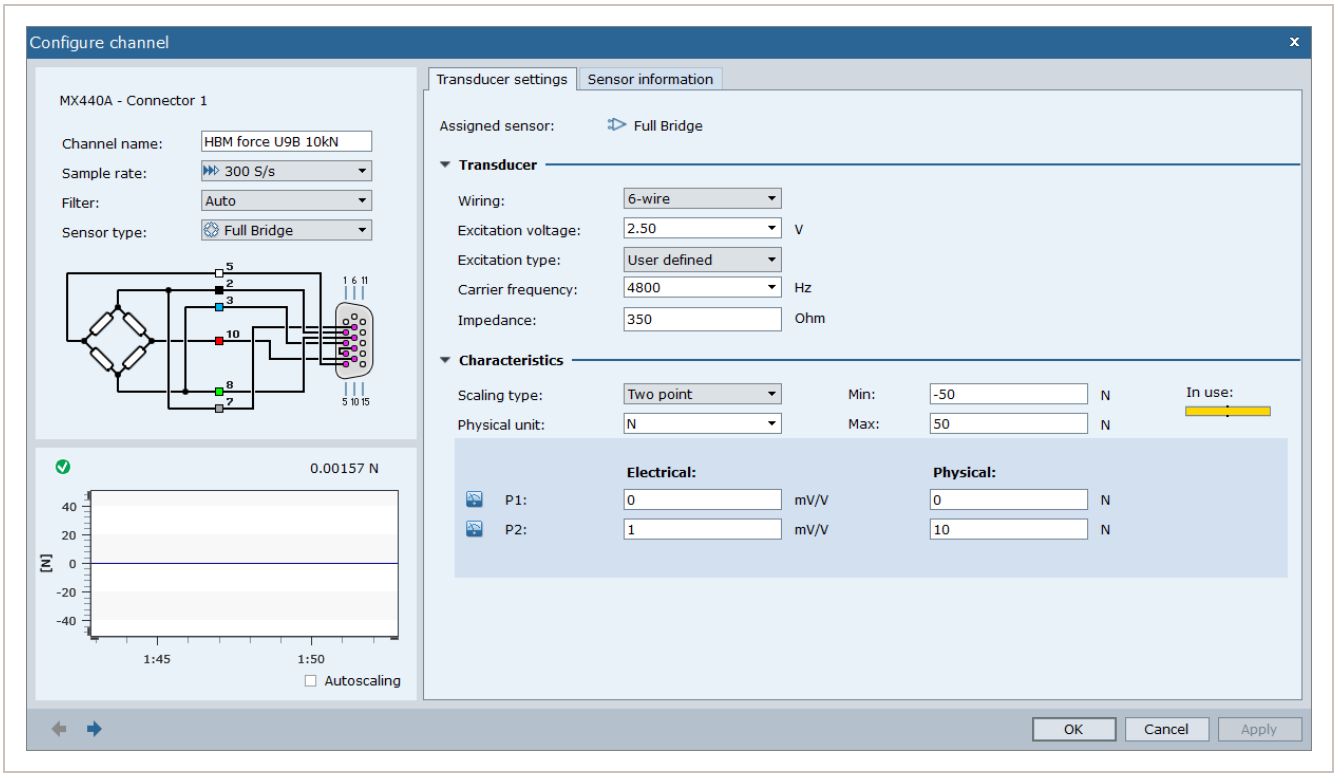
- Data acquisition device and sensors are connected.
- Project channels are selected (To select project channels, 59).

To configure a project channel in the Configure channel dialog box

1 In **Project channels**, click a channel.

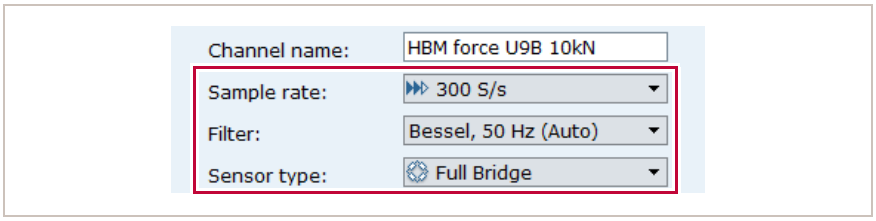
2 Click **Configure**.

✓ The **Configure channel** dialog box is displayed.

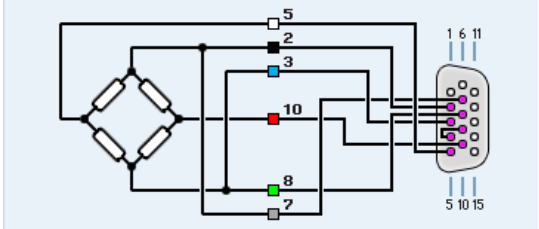


ⓘ Alternatively, to open this dialog box, double-click the row header of the channel.

3 Select a sensor type, sample rate, and filter for the channel.

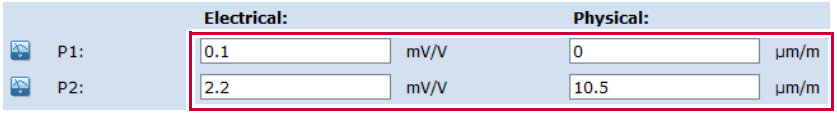


4 Check if the pinning of the sensor matches the wiring diagram of the sensor type (Channel settings, 77).

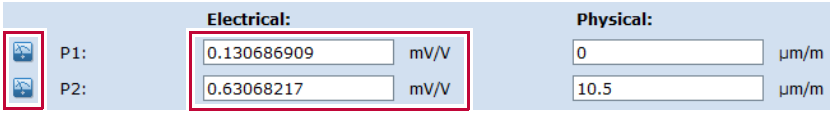


5 On the **Transducer settings** tab, enter the sensor settings from the sensor data sheet.

ⓘ Alternatively, you can calibrate the sensor by using your own calibration points, e.g., for the linear characteristic **Scaling type** = **Two point** by defining **P1** = (**0.1** mV/V | **0** µm/m) and **P2** = (**2.2** mV/V | **10.5** µm/m).

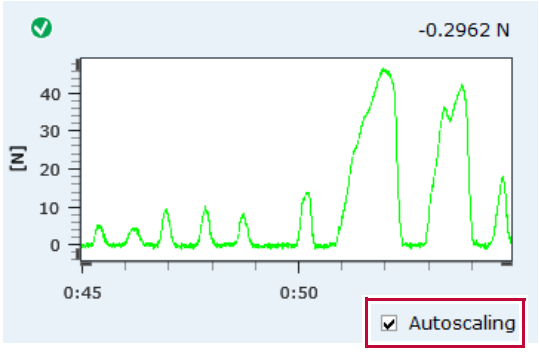


ⓘ If you need the actual electrical values for **P1** and **P2** while the sensor is under a defined load, click **Measure electrical value**.

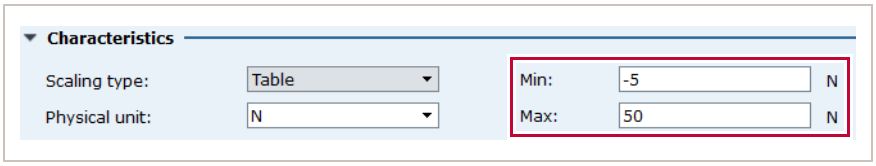


The **Measure electrical value** buttons are available for linear calibration characteristics, i.e., for the **Zero span**, **Two-point**, and **Table** scaling types (Scaling type, 238).

6 For the signal preview chart, if you want the scale of the y-axis to adjust automatically to the range of measured values, select the **Autoscaling** check box.



7 If you want a fixed scale, clear the **Autoscaling** check box, and then define a range of physical values on the **Transducer settings** tab.



✓ The notification bar is displayed.

8 To apply the changes, click **Apply** and then click **OK**.

ⓘ After changing a setting, you must rebalance the channel (Zero balance commands, 101).

9 If you want to create a user-defined sensor from your settings, on the **Sensor information** tab, enter a sensor name and the other metadata needed, and then click **Create sensor**.

3.3.4 Configure on Project channels panel

You can configure project channels directly on the **Project channels** panel with one of the following actions:

- Select an option from a drop-down list.
- Type a setting into a table cell.
- Copy settings from one project channel to another.

Prerequisites

- Data acquisition device and sensors are connected.
- Project channels are selected (To select project channels, 59).

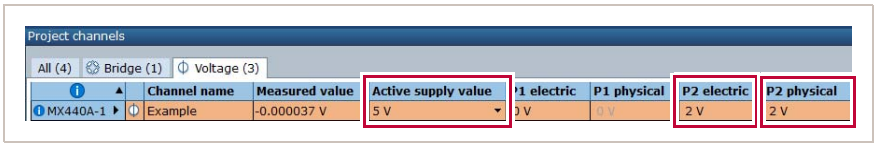
The following procedure illustrates your options.

To configure project channels on the Project channels panel

1 In **Project channels**, click a tab that contains a subgroup of channels, e.g., the **Voltage** tab.

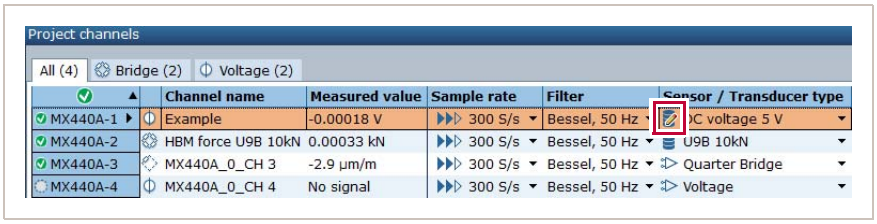
ⓘ On the **All** tab, you can edit the global channel settings. On the other tabs, you can edit the sensor settings (Project channels panel, 107).

2 Edit the sensor settings, e.g.:



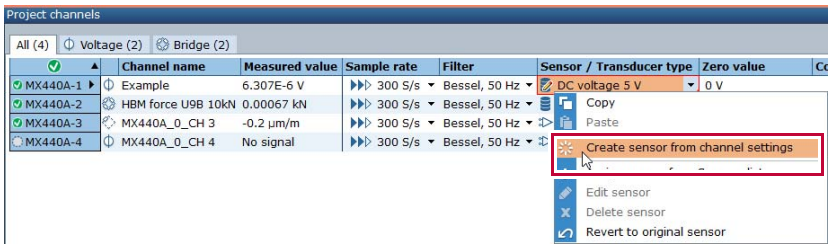
ⓘ The new settings are saved automatically.

3 Click the **All** tab.

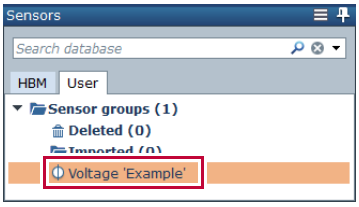


ⓘ In the **Sensor / Transducer type** column, indicates that the settings have been changed.

4 In the **Sensor / Transducer type** column, right-click the sensor, and then click **Create sensor from channel settings**.

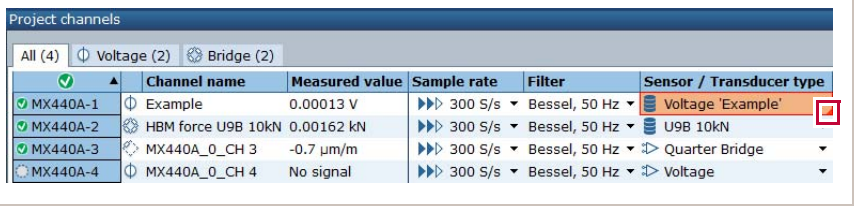


ⓘ The sensor is renamed and saved as a user-defined sensor.



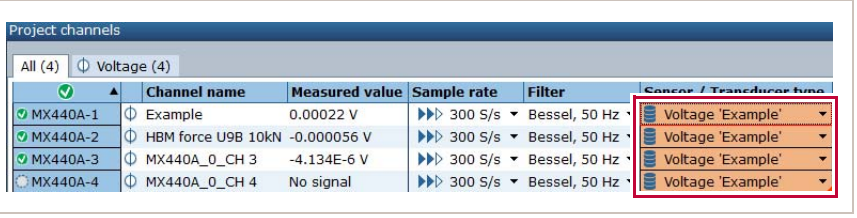
5 In the **Sensor / Transducer type** column, click the sensor again.

✓ In the right corner of the table cell, a red triangle is displayed.



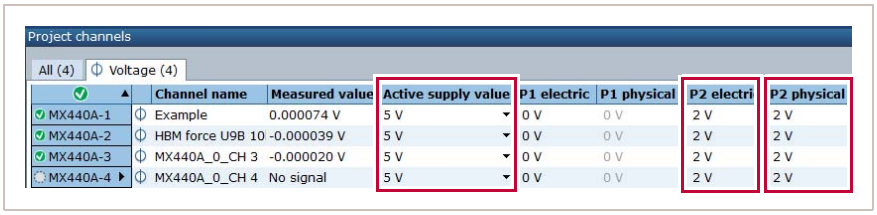
6 Point to the red triangle. When the pointer turns into a ↕+, drag the pointer to copy the sensor and its settings to the other project channels.

✓ The sensor is copied to the other channels.



7 Click the **Voltage** tab.

✓ All project channels are now configured with the same sensor settings.



3.4 Saving a project

If you want to reuse a set of project channels, you must save them to a project file.

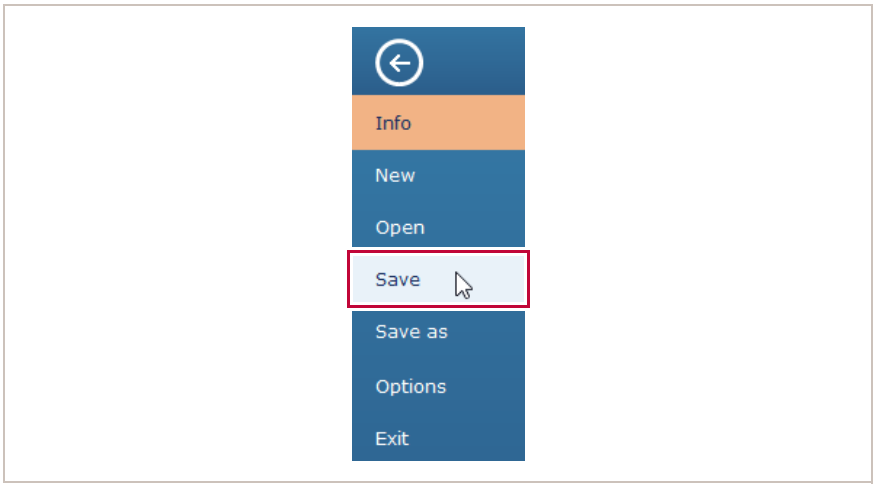
Prerequisites

- Project channels are selected (To select project channels, 59).
- Project channels are configured (Configuring project channels, 60).

To save a project

1 Click the **File** tab.

2 Click **Save**.



✓ The Explorer is displayed.

3 Enter a filename.

4 Click **Save**.

✓ The project channels and all related settings are saved in a project file (*.evidas).

⇨ File menu, 220
Opening a project, 71

3.5 Opening a project

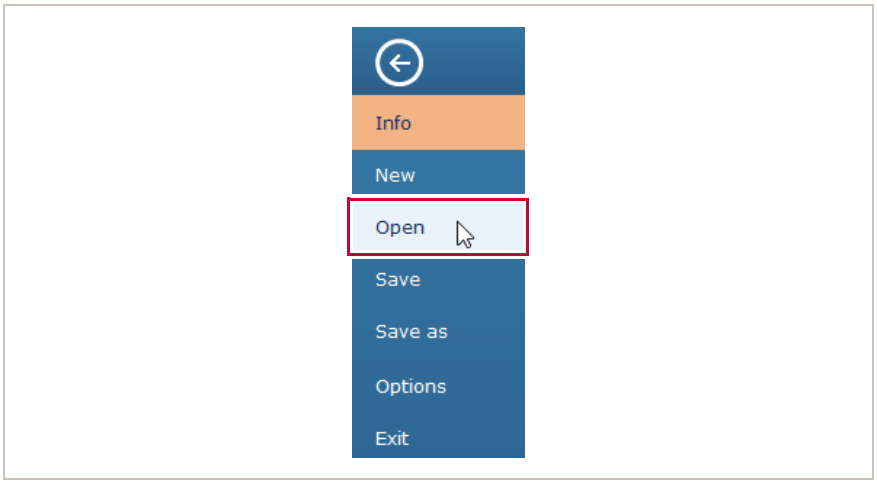
Prerequisites

- A project file (*.evidas) is available (Saving a project, 70).

To open a project

1 Click the **File** tab.

2 Click **Open**.



✓ The recently opened project files are displayed.

3 Click **Browse** and navigate to the project file.

ⓘ The default folder for project files is C:\Users\Public\Documents\HBM\EVIDAS\Projects.

4 Click **Open**.

✓ The project channels and all related settings are loaded.

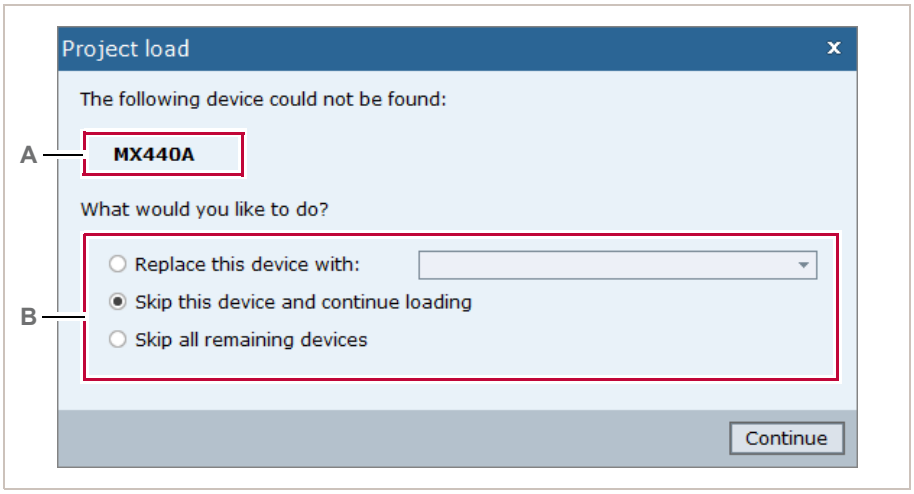
ⓘ If a data acquisition device is required by the project, but is not connected to the system, a hardware mapper is displayed (Hardware mapper, 72).

ⓘ Alternatively, use Windows Explorer to navigate to the project file, and then double-click the project file.

⇨ File menu, 220

3.6 Hardware mapper

To display, click the **File** tab, click **Open**, and then click a project file (*.evidas) that requires a data acquisition device which is currently not connected to the system.



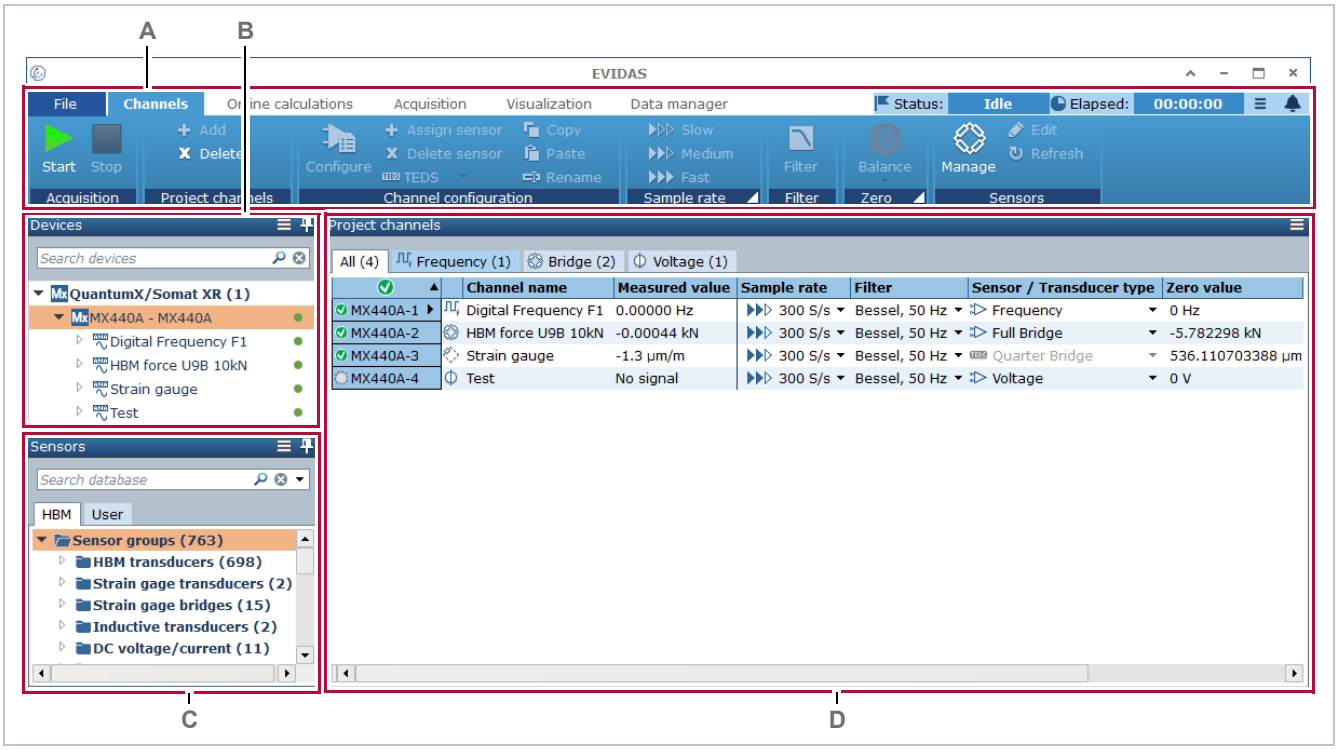
A Name of device not found B Options to proceed with loading the project

You have the following options to proceed with loading the project:

Option	Description
Replace this device with	To use another device, from the drop-down list, select one of the devices that are connected to the system.
Skip this device and continue loading	To skip the device not found and to continue loading the project. You may connect the required device after loading the project.
Skip all remaining devices	To skip the device not found and to skip all other devices required by the project, but not connected to the system, and to continue loading the project. You may connect the required devices after loading the project.

4 Channels

On the **Channels** tab and its panels, you select and configure the project channels.



A Channels tab, 74

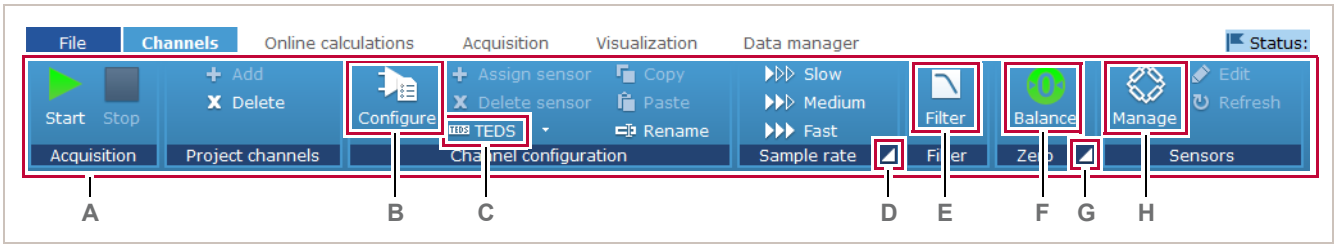
B Devices panel, 104

C Sensors panel, 230

D Project channels panel, 107

4.1 Channels tab

Some commands on the **Channels** tab display a dialog box or menu with further commands or options.



- **A** Channels command groups, 75
- **B** Configure channel dialog box, 76
- **C** TEDS options, 88
- **D** Sample rate group and domain options, 89
- **E** Filter options, 97
- **F** Zero balance commands, 101
- **G** Zero balance options, 103
- **H** Appendix A: Sensor manager, 227

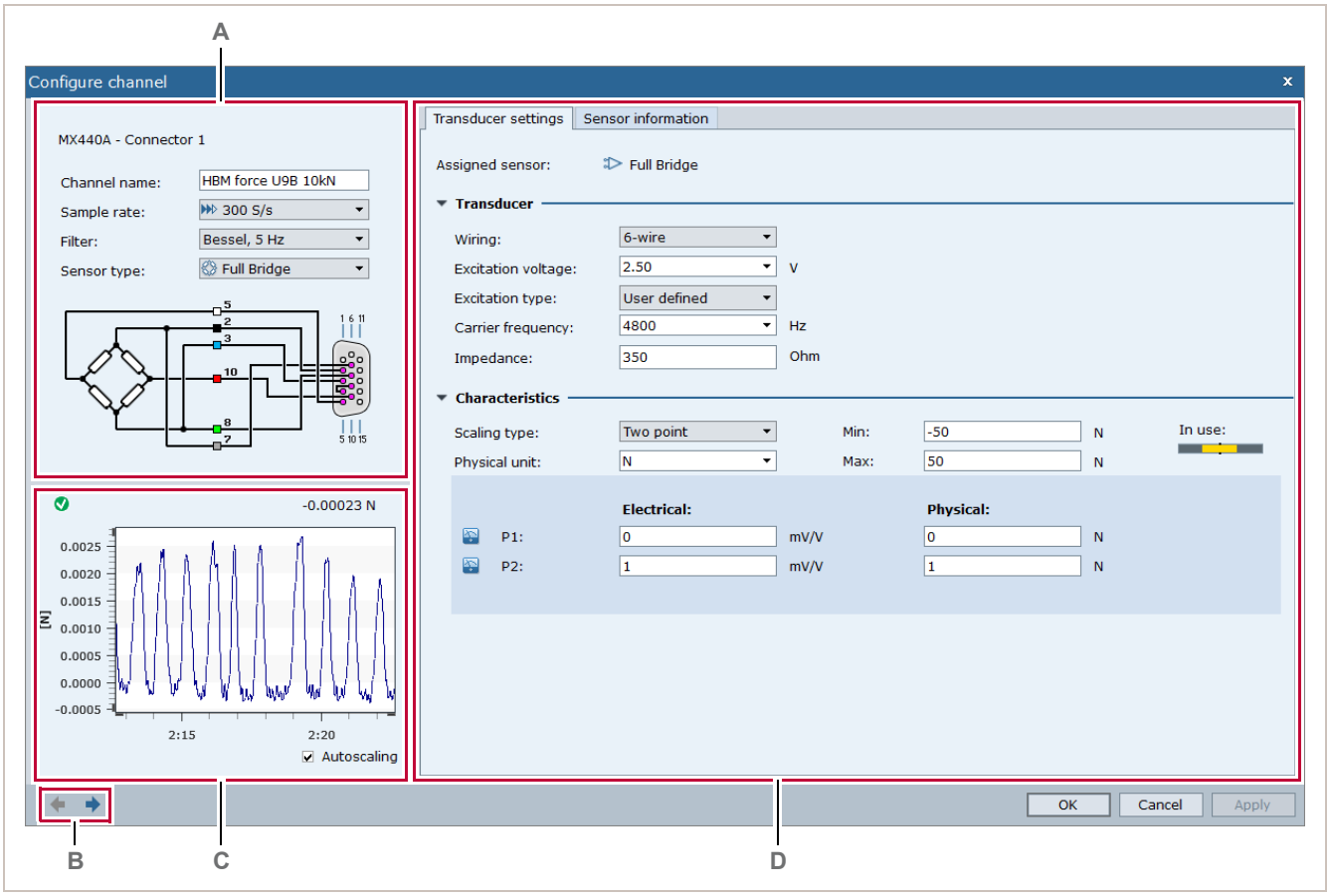
4.1.1 Channels command groups

Group	Description
Project channels	To select channels as project channels or to delete project channels from the **Project channels** panel (Project channels panel, 107↗).
Channel configuration	To configure project channels. Click a project channel, and then click **Configure** to display the **Configure channel** dialog box (Configure channel dialog box, 76↗). Click **TEDS** to display options for TEDS sensors (TEDS options, 88↗).
Sample rate	To define sample rate groups and sample rate domains. Click to display the **Sample rate groups** dialog box (Sample rate group and domain options, 89↗).
Filter	To define a filter for a project channel. Click a project channel, and then click **Filter** to display filter options for the channel (Filter options, 97↗).
Zero	To zero balance project channels. Hold down Ctrl while clicking the project channels you want to balance, and then click **Balance**. To display all commands related to zero balancing, click a channel, and then click **Balance** (Zero balance commands, 101↗). Click to define how zero values are determined (Zero balance options, 103↗).
Sensors	To display and edit sensor settings. Click **Manage** to display the **Sensor manager** tab (Appendix A: Sensor manager, 227↗).

⇨ Start / Stop button, 40↗

4.1.2 Configure channel dialog box

To display, click the **Channels** tab, click a project channel, and then click **Configure**.



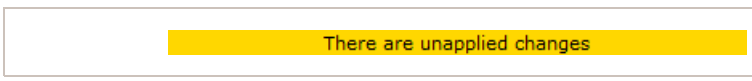
A Channel settings, 77

B Previous / Next buttons, 77

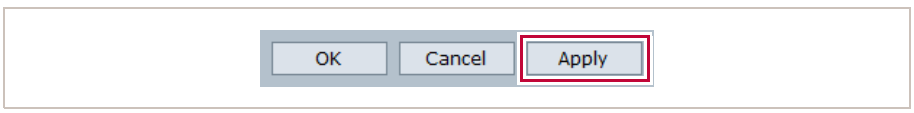
C Signal preview, 78

D Sensor settings, 79

In this dialog box, you can configure a single project channel. If you edit a setting, a notification bar is displayed.



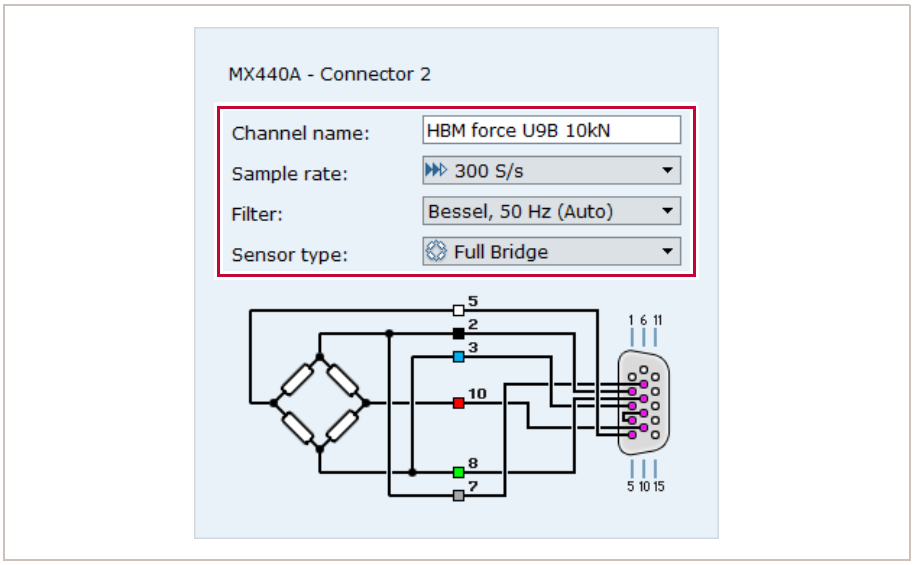
To apply a change, click **Apply**.



After changing a setting, you must rebalance the channel (Zero balance commands, 101).

Channel settings

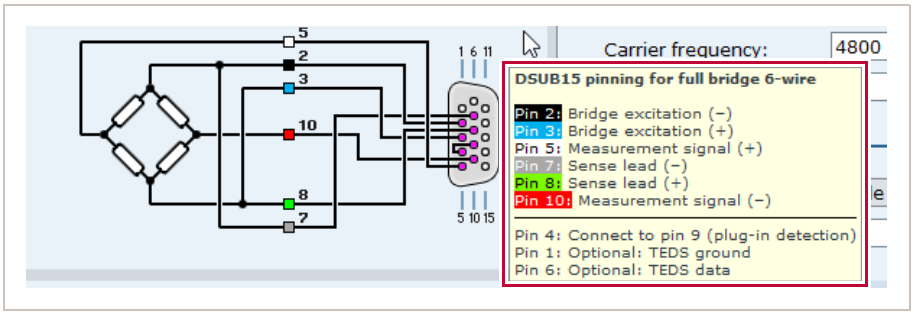
You can edit the channel name and change the sensor type, sample rate, and filter for the channel.



The wiring diagram shows which sensor wire should be soldered to which connector pin.

The colors displayed for the wires are the standard HBM wire colors.

For detailed pinning information, point to the diagram. A tooltip is displayed.



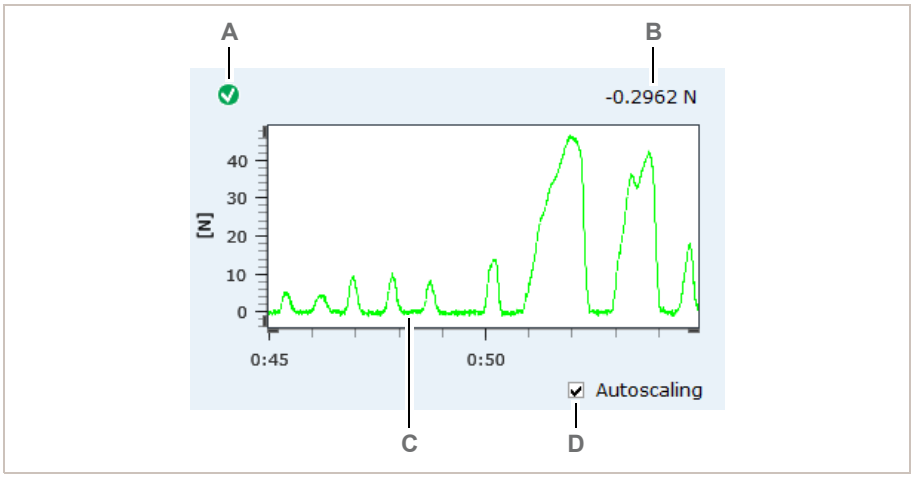
Make sure that the pinning of your sensor matches the wiring diagram.

Previous / Next buttons

Click or to display the previous or next project channel from the list on the **Project channels** panel.

Signal preview

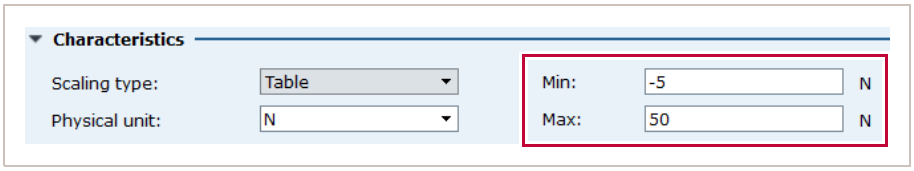
The signal preview displays a real-time trace of the signal without starting data acquisition.



A	Signal status	C	Real-time trace
B	Live measured value	D	Autoscaling check box

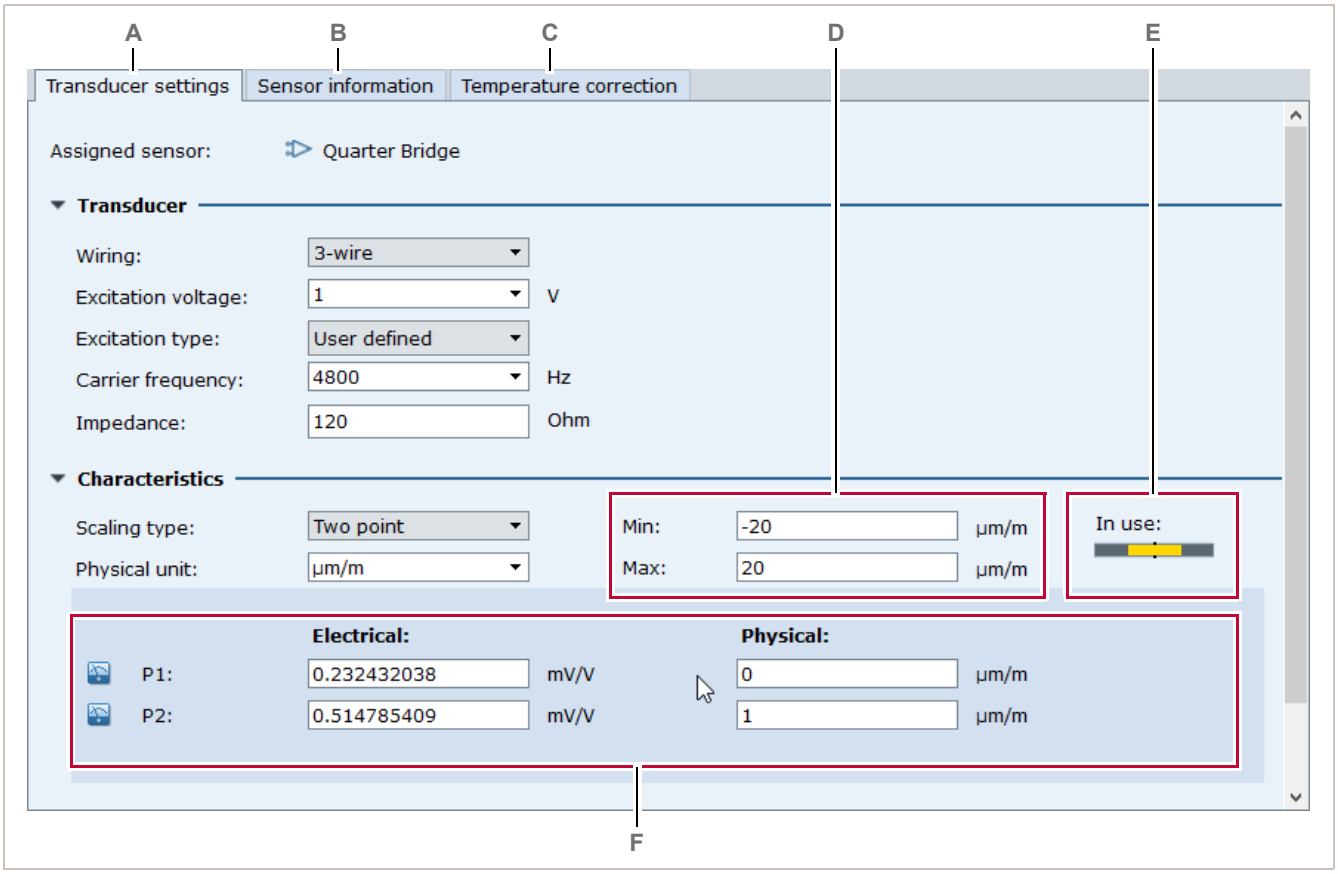
If you want the scale of the y-axis to adjust automatically to the range of measured values, select the **Autoscaling** check box.

If you want a fixed scale, clear the **Autoscaling** check box, and then define a range of physical values on the **Transducer settings** tab.



Sensor settings

On the following tabs, you can edit the sensor settings, create user-defined sensors, and enter temperature correction parameters.



A Transducer settings, 79↗

B Sensor information, 79↗

C Temperature correction, 79↗

D Measuring range, 80↗

E In use bar, 80↗

F Calibration parameters, 81↗

Transducer settings

To edit the sensor settings, e.g., to match the settings on the sensor data sheet (Transducer settings, 237↗).

Sensor information

To edit the metadata of the sensor, e.g., the sensor name and its calibration date. You can create user-defined sensors.

Temperature correction

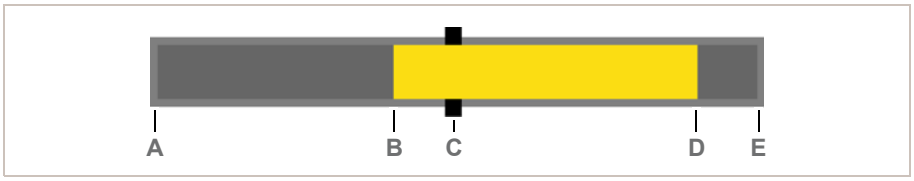
This tab is displayed for strain gauges (quarter bridge). You can compensate for unwanted temperature effects on the measured values (Temperature correction, 82↗).

Measuring range

The measuring range is defined by a minimum and a maximum physical value between which you expect the measured values to be. On the y(t) chart, the measuring range is displayed on the y-axis if you use the scaling type from the sensor settings (Y-axes tab for y(t) charts, 154).

In use bar

The **In use** bar displays the measuring range in relation to the total voltage range available from the amplifier.



- **A** Minimum value of total range
- **B** Minimum value of measuring range
- **C** Zero point
- **D** Maximum value of measuring range
- **E** Maximum value of total range

If your measuring range is too small in relation to the total available range, the measuring resolution might be insufficient and the measured values could turn out inaccurate.

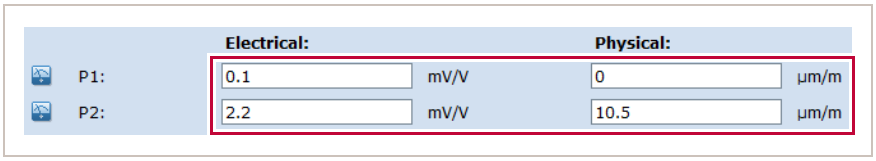
Calibration parameters

The calibration characteristic of a sensor is defined by a scaling type (Scaling type, 238) and a set of calibration parameters pertaining to the scaling type which together specify how a valid range of electrical values is mapped to the corresponding physical values.

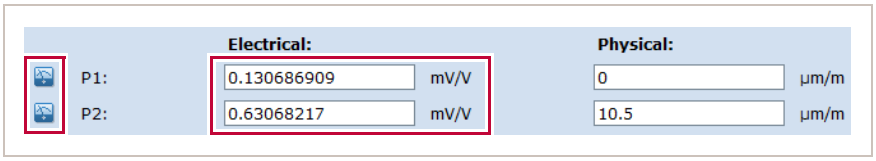
For linear calibration characteristics, i.e., **Zero span**, **Two-point**, and **Table**, you can either use the settings from the sensor data sheet or specify two or more calibration points **P1**, **P2**, etc.

Example: **Scaling type** = **Two point**

- **P1** = (**0.1** mV/V | **0** µm/m)
- **P2** = (**2.2** mV/V | **10.5** µm/m)



If you need the actual electrical values for **P1** and **P2** while the sensor is under a defined load, click **Measure electrical value**.

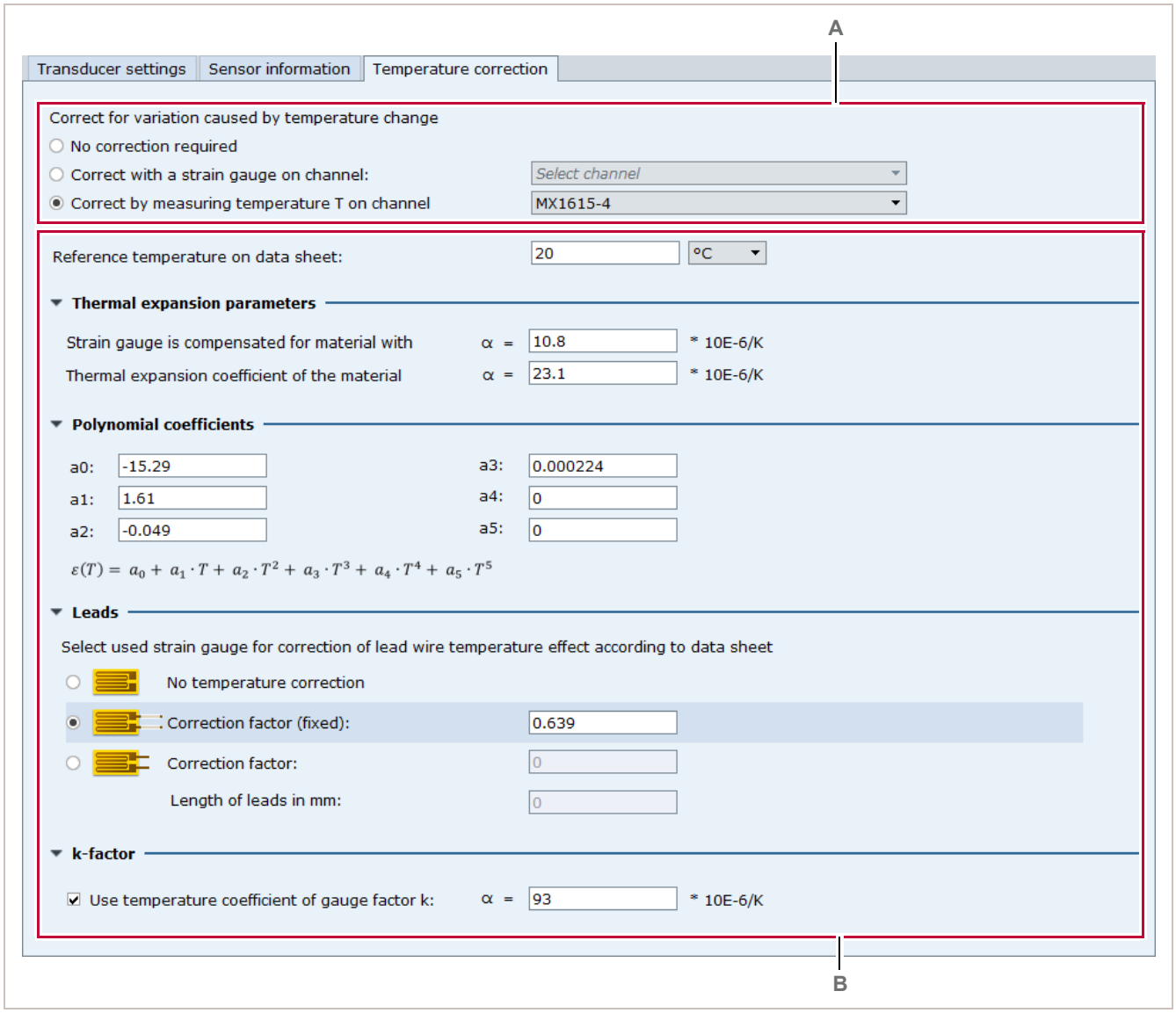


The **Measure electrical value** buttons are available for linear calibration characteristics, i.e., for the **Zero span**, **Two-point**, and **Table** scaling types (Scaling type, 238).

For non-linear calibration characteristics, e.g., **Polynomial**, you must enter the polynomial coefficients from the sensor data sheet (Transducer settings, 237).

4.1.3 Temperature correction

To display, click the **Channels** tab, click a project channel, click **Configure**, and then click the **Temperature correction** tab.



A Temperature correction options, 83↗ B Temperature correction parameters, 84↗

Strain gauges (quarter bridge) that are operated in an environment with a temperature that deviates from the usual reference temperature of 20°C may deliver skewed measured values due to thermal expansion or contraction of the materials involved.

On the **Temperature correction** tab, you can compensate for unwanted temperature effects on the measured values.

Temperature correction options

Option	Description
No correction required	The strain gauge is operated in an environment with a temperature of 20°C to 25°C, e.g., in a laboratory.
Correct with a strain gauge on channel	The strain gauge is operated in an environment with a temperature that deviates significantly from the reference temperature of 20°C, e.g., in a range from 0°C to 40°C. For each strain gauge used, you must specify a second identical strain gauge which is placed in the vicinity of the first one, without being exposed to strain. EVIDAS then automatically compensates for undesirable temperature effects on the measured values.
Correct by measuring temperature T on channel	The strain gauge is operated in an environment with a temperature that deviates significantly from the reference temperature of 20°C, e.g., in a range from 0°C to 40°C. You must specify a temperature channel that measures the ambient temperature. A single temperature channel is sufficient for all strain gauges involved. You must enter the temperature correction parameters (Temperature correction parameters, 84) from the respective data sheet (Strain gauge data sheet, 87).

Temperature correction parameters

If you use temperature correction with a temperature reference channel (Temperature correction options, 83), you must enter the following temperature correction parameters from the sensor data sheet:

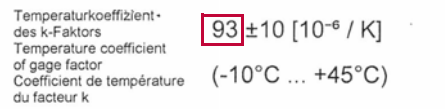
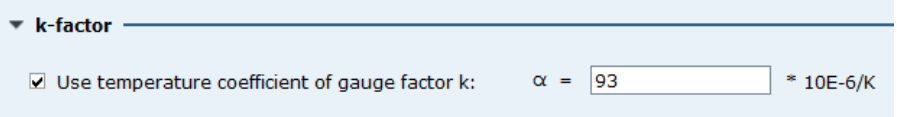
Temperature correction parameters when using temperature reference channel

Parameter	Description
Reference temperature on data sheet	The reference temperature is specified in the polynomial on the data sheet (Strain gauge data sheet, 87), e.g.: $\varepsilon_S(T) = -15.29 + 1.61 * T - 4.90E\text{-}02 * T^2 + 2.24E\text{-}04 * T^3 \pm (T-20) * 0.30\ [\mu m/m] + 0.639 * (T-20)\ [\mu m/m]$ You must enter the reference temperature on the **Temperature correction** tab. Reference temperature on data sheet: 20 °C At reference temperature, the measured values are not skewed, i.e., the temperature-dependent correction $\varepsilon_s \approx 0$.
Thermal expansion parameters	HBM determines the temperature correction parameters of the strain gauge while it is attached to a specific material, e.g., to steel. The thermal expansion of the material is specified by the thermal expansion coefficient α (Strain gauge data sheet, 87), e.g.: Temperaturkompensation: Ferritischer Stahl mit Temperature compensation: steel with Compensation de température: acier avec $\alpha = 10.8\ [10^{-6}\ /\ K]$ You must enter α into the first text box of the **Thermal expansion parameters** section, e.g. 10.8 for steel. If you use the same material as on the data sheet, you must enter α also into the second text box. **Thermal expansion parameters** Strain gauge is compensated for material with α = 10.8 * 10E-6/K Thermal expansion coefficient of the material α = 10.8 * 10E-6/K If the material on which you use the strain gauge is different, you must enter the respective thermal expansion coefficient α, e.g., 23.1 for aluminium. **Thermal expansion parameters** Strain gauge is compensated for material with α = 10.8 * 10E-6/K Thermal expansion coefficient of the material α = 23.1 * 10E-6/K

Temperature correction parameters when using temperature reference channel

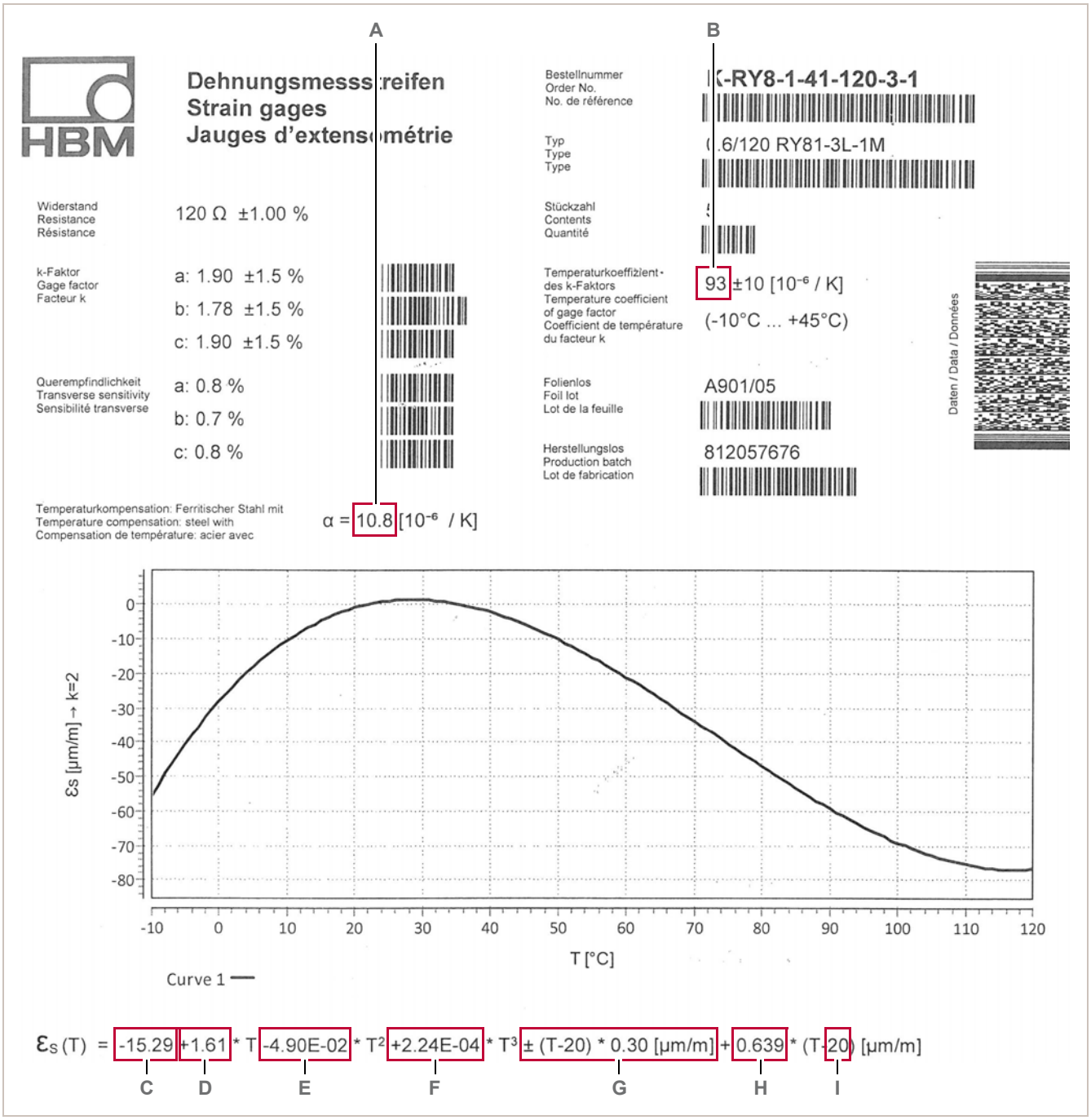
Parameter	Description
Polynomial coefficients	The polynomial coefficients are specified in the polynomial on the data sheet (Strain gauge data sheet, 87↗), e.g.: $\varepsilon_S(T)$ = -15.29 +1.61 * T -4.90E-02 * T² +2.24E-04 * T³ ± (T-20) * 0.30 [µm/m] + 0.639 * (T-20) [µm/m] You must enter as many polynomial coefficients as specified in the polynomial. **Polynomial coefficients** a0: -15.29 a3: 0.000224 a1: 1.61 a4: 0 a2: -0.049 a5: 0 $\varepsilon(T) = a_0 + a_1 \cdot T + a_2 \cdot T^2 + a_3 \cdot T^3 + a_4 \cdot T^4 + a_5 \cdot T^5$ If a polynomial coefficient uses exponential representation, e.g., -4.90E-02, type the exponential representation and EVIDAS will convert it to decimal. The polynomial describes the temperature-dependent correction ε_s.
Leads	The effect of temperature on the total electrical resistance is also influenced by the length of the lead wires connected to the strain gauge. The correction factor is specified in the polynomial on the data sheet (Strain gauge data sheet, 87↗), e.g.: $\varepsilon_S(T)$ = -15.29 +1.61 * T -4.90E-02 * T² +2.24E-04 * T³ ± (T-20) * 0.30 [µm/m] + 0.639 * (T-20) [µm/m] If you use a strain gauge that comes with fixed length lead wires, you must enter the specified correction factor. **Leads** Select used strain gauge for correction of lead wire temperature effect according to data sheet No temperature correction Correction factor (fixed): 0.639 Correction factor: 0 Length of leads in mm: 0 If you use a strain gauge and cut the existing lead wires, you must enter the specified correction factor as well as the length of the leads in millimeters.

Temperature correction parameters when using temperature reference channel

Parameter	Description
k-factor	The temperature coefficient of the gauge factor is specified on the data sheet (Strain gauge data sheet, 87), e.g.:  Typical temperature coefficients of the gauge factor are 0.01 %/K. Their effect on the measurement result is usually very small and therefore ignored. However, compensation for the temperature-dependency of the gauge factor is possible. 

Strain gauge data sheet

The following data sheet for a strain gauge (quarter bridge) illustrates where to find the parameters for temperature correction with a temperature reference channel (Temperature correction parameters, 84).

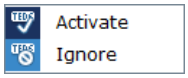


A	Thermal expansion parameter α	D	Polynomial coefficient a1	G	Range of error of correction polynomial
B	Temperature coefficient of the gauge factor	E	Polynomial coefficient a2	H	Correction factor for fixed length leads
C	Polynomial coefficient a0	F	Polynomial coefficient a3	I	Reference temperature

4.1.4 TEDS options

A TEDS sensor has a connector with a built-in TEDS (**T**ransducer **E**lectronic **D**ata **S**heet) chip that contains the sensor settings.

To display the TEDS options, click the **Channels** tab, click a project channel, and then click **TEDS**.



The TEDS option assigned to the channel is displayed in dark blue, e.g., **Activate**.

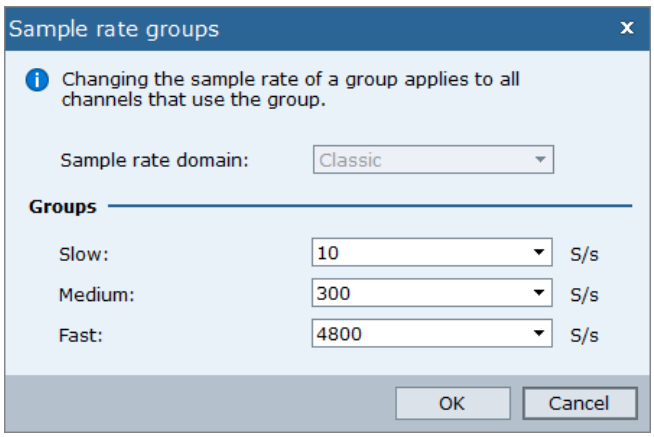
TEDS options

Option	Description
Activate	If the channel has a TEDS sensor, the settings from the TEDS chip are used to configure the channel. Even if you connect the TEDS sensor later, the settings from the TEDS chip will be used. If you select all project channels and assign this option, the settings of the connected TEDS sensors will be used to configure the respective channels. Even if you unplug a TEDS sensor and plug it into a different channel, EVIDAS will recognize the TEDS sensor and activate its settings. For project channels with this option, on the **Project channels** panel (Project channels panel, 107↗), you can change the channel name, sample rate, filter, and zero value. If you want to edit other settings, e.g., the calibration date, click **Configure** and then click the following message: This is a TEDS transducer. Click the lock to make changes. A message indicates that you are editing a TEDS sensor: You are editing a TEDS transducer. To again lock the TEDS sensor, click this message or click **Apply** to save your changes. The changes are saved on the TEDS chip.
Ignore	If the channel has a TEDS sensor, the settings from the TEDS chip are ignored. The channel is not configured automatically. Use this option if you want to use a TEDS sensor with your own settings. Click **Configure** to edit any setting (Sensor settings, 79↗). The settings are saved in the data acquisition device. The settings on the TEDS chip are not affected.

⇨ Configure with TEDS sensor, 61↗

4.1.5 Sample rate group and domain options

To display, click the **Channels** tab, and then click the Dialog Box Launcher of the **Sample rate** group.



In this dialog box, you define the sample rate domain (Sample rate domain, 90) and the sample rate groups (Sample rate groups, 91).

Sample rate domain

A data acquisition device has one or two basic sample rates, e.g., 192,000 Hz. The other sample rates available from the device are derived by dividing the basic sample rate, e.g., by 2, 4, 5, and 10, leading to sample rates such as 96,000 Hz, 48,000 Hz, 38,400 Hz, and 19,200 Hz.

The sample rate domain is a setting applied to data acquisition devices. It determines the set of sample rates available for a measuring project.

Sample rate domain	Description
Classic	Available with all HBM data acquisition devices. The following sample rates are part of this domain: …, 20, 25, 50, 75, 100, 200, 300, 600, 1,200, 2,400, 4,800, 9,600, 19,200, 38,400, 48,000, 96,000, and 192,000 Hz.
Decimal	Available with some devices only. The following sample rates are part of this domain: …, 20, 50, 100, 200, 500, 600, 1,000, 2,000, 2,500, 5,000, 10,000, 20,000, 25,000, 50,000, 100,000, and 200,000 Hz. Devices with a **Decimal** sample rate domain can be switched to the **Classic** sample rate domain (Changing the sample rate domain, 92).

The sample rate domain selected for a data acquisition device applies to all channels of the device.

The data acquisition devices of a measuring project must all use the same sample rate domain.

If you have one device that has only the **Classic** sample rate domain and another device that has both sample rate domains, you must use the **Classic** sample rate domain for both.

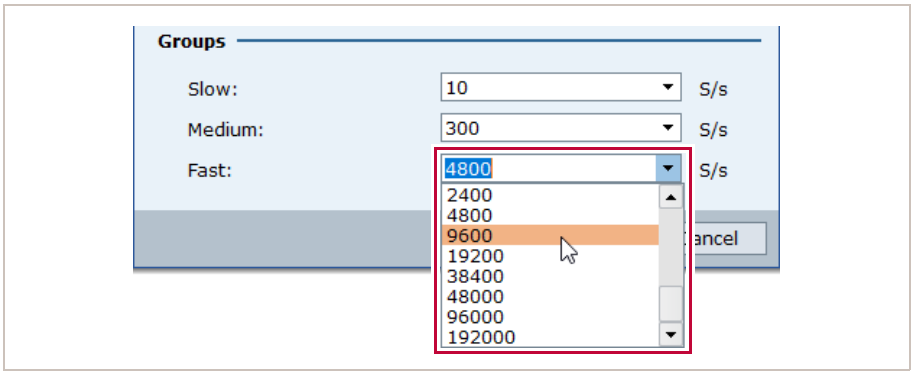
Changing the sample rate domain, 92

Sample rate groups

You can define three preferred sample rates for a measuring project. These sample rates are referred to as sample rate groups.

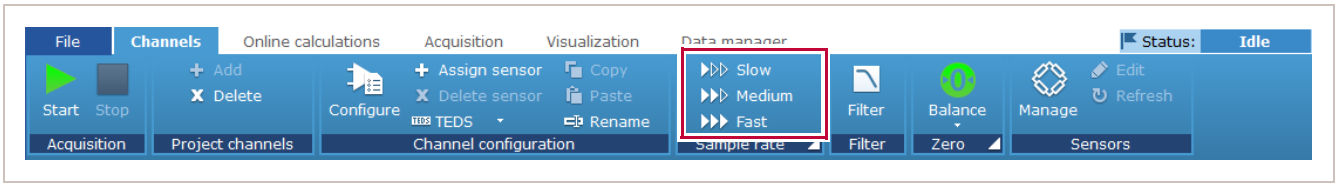
Sample rate group	Description
Slow	Slow sample rate.
Medium	Medium sample rate (default).
Fast	Fast sample rate.

To define a sample rate group, in the **Sample rate groups** dialog box (Sample rate group and domain options, 89), select a sample rate from the drop-down list.



The available sample rates are determined by the sample rate domain (Sample rate domain, 90).

To assign a project channel to a sample rate group, click the **Channels** tab, click a project channel, and then click **Slow**, **Medium**, or **Fast**.



If you change the sample rate of a group, the change applies to all project channels assigned to this group.

Not every project channel must be assigned to a sample rate group. You can assign sample rates without using a sample rate group (Assigning sample rates to project channels, 93). Each project channel can have its own sample rate.

4.1.6 Changing the sample rate domain

You can change the sample rate domain of a data acquisition device if the device supports both the **Classic** and **Decimal** sample rate domains.

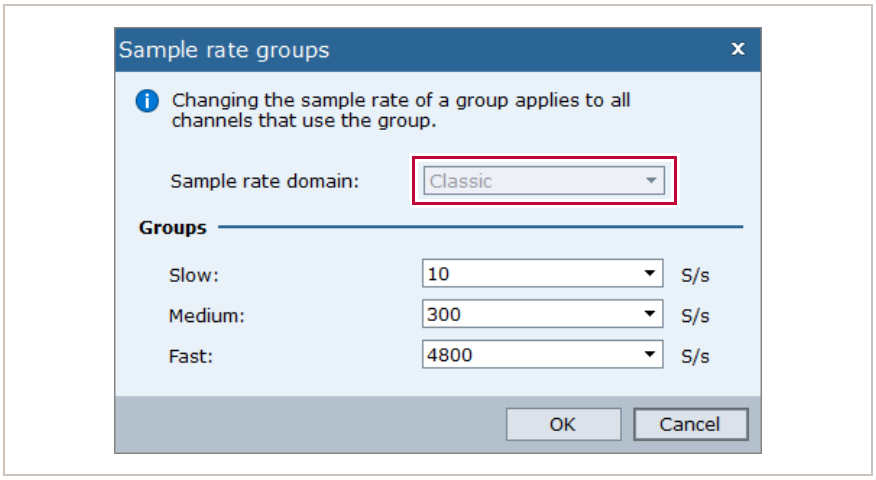
To change the sample rate domain

1 Click the **Channels** tab.

2 In the **Sample rate** group, click .

✓ The **Sample rate groups** dialog box is displayed.

3 Select a sample rate domain.



4 Click **OK**.

✓ A dialog box is displayed that asks you to confirm the new sample rate domain.

5 Click **Restart devices**.

✓ The data acquisition device is restarted. Wait until the LEDs on the device indicate that it is ready.

6 Adjust the sample rate groups and filters to the project.

⇨ Sample rate groups, 91
Filter options, 97

4.1.7 Assigning sample rates to project channels

The following procedure illustrates your options to assign sample rates to project channels.

Prerequisites

- Data acquisition device and sensors are connected.
- Project channels are selected (To select project channels, 59).

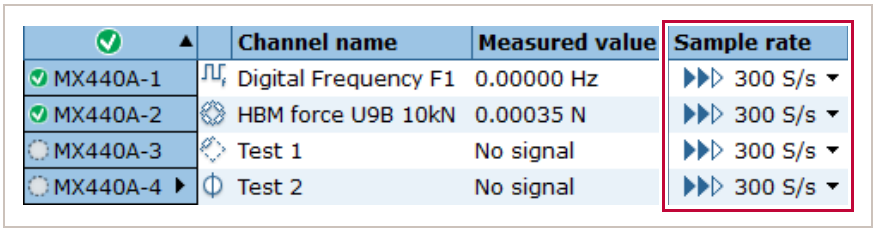
You have the following options to assign a sample rate:

Option	Description
Automatically	On the **Options** panel (Configuring the sample rate and filter of project channels automatically, 100), if you selected **Configure sample rate and filter automatically**, new project channels are assigned to the ▶▶▷ **Medium** sample rate group. If you did not select automatic channel configuration, the sample rate that is stored in the data acquisition device is used.
Via sample rate group	Assign a project channel to a different sample rate group or change the sample rate of a sample rate group (see below).
Manually	Manually assign a sample rate to a project channel via the sample rate drop-down menu. The project channel is no longer assigned to a sample rate group (see below).

To assign sample rates

1 Click the **Channels** tab.

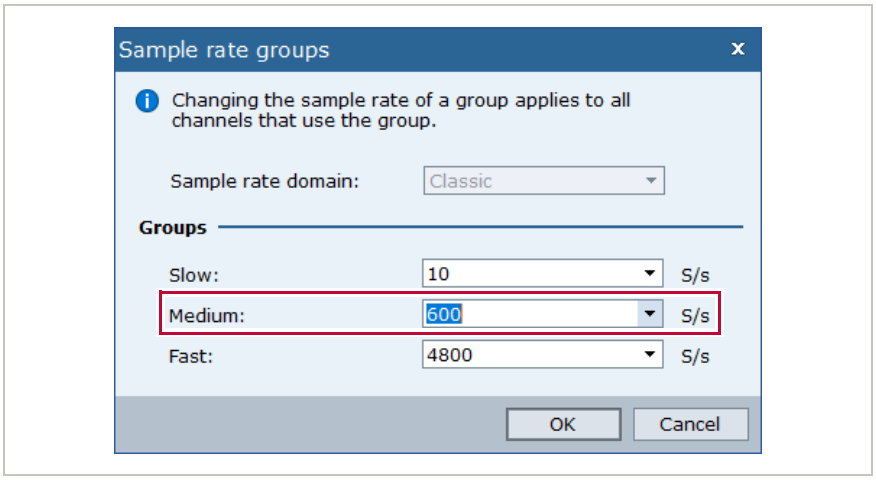
✓ The **Project channels** panel is displayed.



2 On the **Channels** tab, in the **Sample rate** group, click .

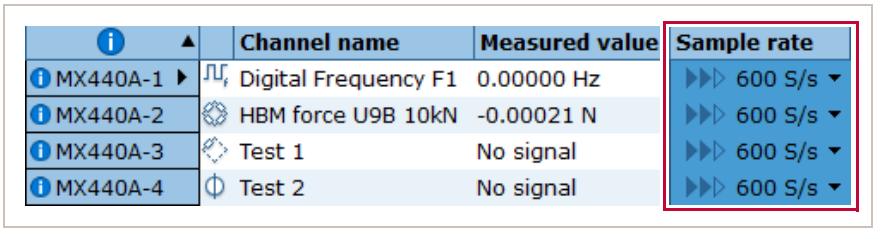
✓ The **Sample rate groups** dialog box is displayed.

3 Change the sample rate of the **Medium** sample rate group to **600** S/s.



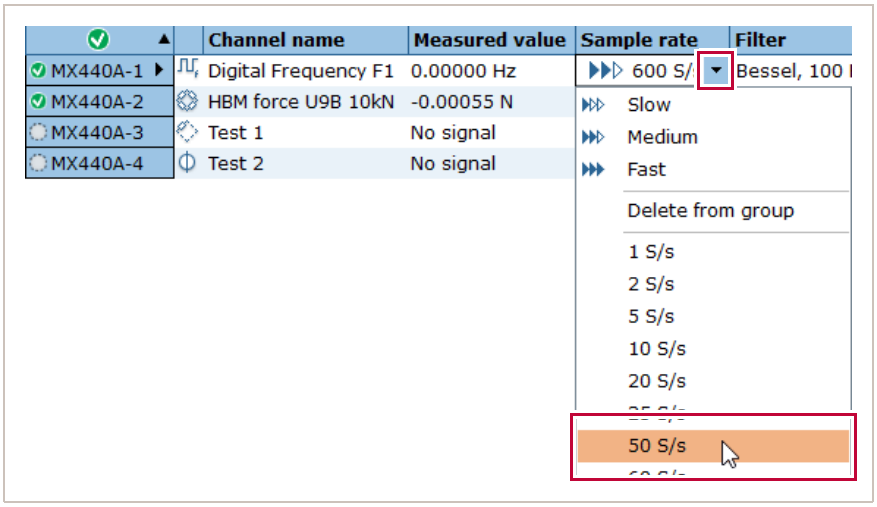
4 Click **OK**.

✓ The sample rate of all project channels is changed to 600 S/s.

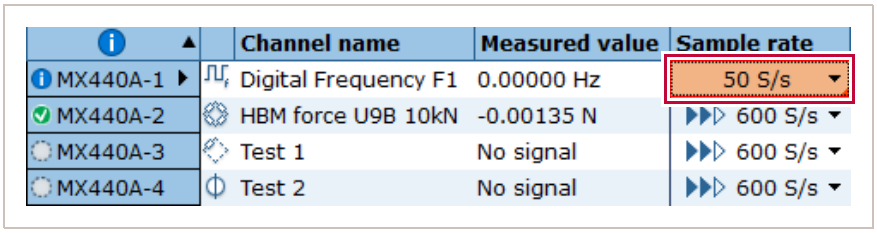


ⓘ On the Options panel (Options panel, 224), if you selected **Show information about automatically adapted settings**, the background of the changed settings is highlighed and an ⓘ is displayed.
Point to ⓘ to display information on the adapted settings.
To clear an info, right-click ⓘ, and then, on the context menu, click **Clear all info notifications**.

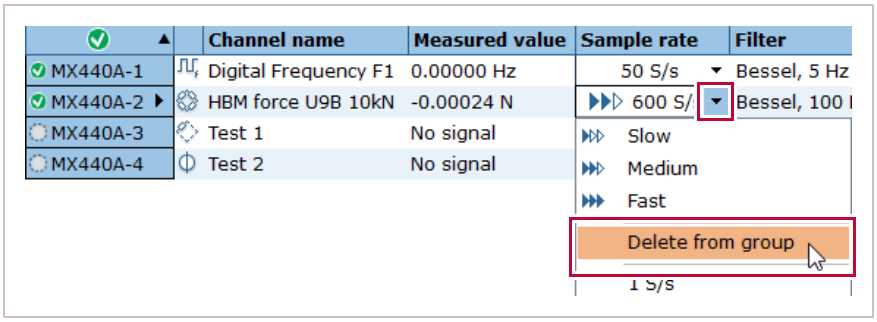
5 For the first project channel, in the **Sample rate** column, click , and then click **50 S/s**.



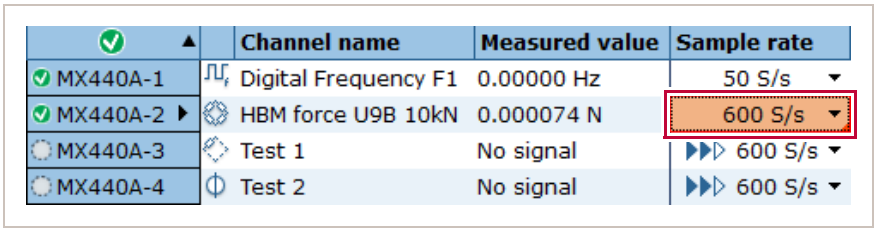
✓ The sample rate of the first channel is changed and the channel is no longer assigned to a sample rate group.



6 For the second project channel, in the **Sample rate** column, click , and then click **Delete from group**.

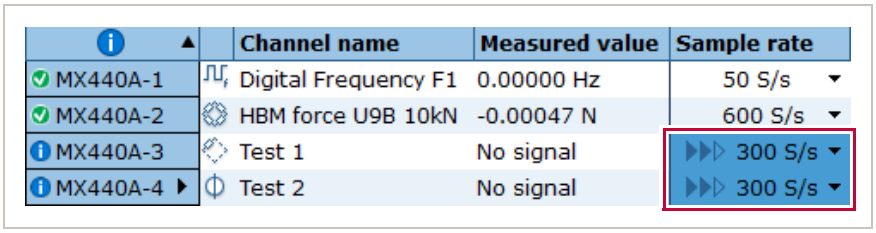


✓ The sample rate of the second channel is unchanged, but the channel is no longer assigned to the ▶▶▷ **Medium** sample rate group.



7 Change the sample rate of the ▶▶▷ **Medium** sample rate group back to **300** S/s.

✓ Only the last two channels are affected by the change.

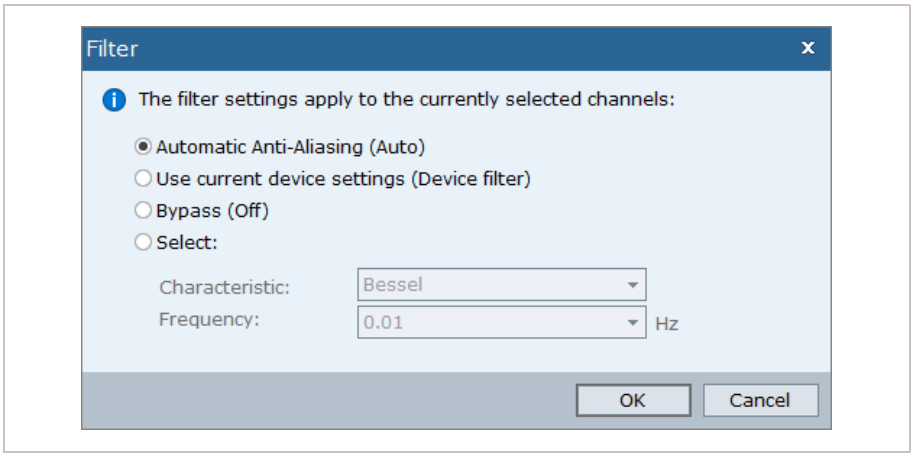


	Channel name	Measured value	Sample rate
MX440A-1	Digital Frequency F1	0.00000 Hz	50 S/s
MX440A-2	HBM force U9B 10kN	-0.00047 N	600 S/s
MX440A-3	Test 1	No signal	▶▶▷ 300 S/s
MX440A-4	Test 2	No signal	▶▶▷ 300 S/s

⇨ Sample rate groups, 91

4.1.8 Filter options

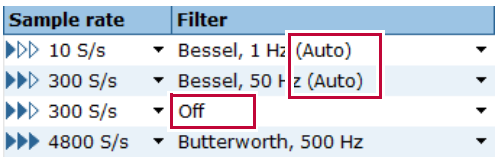
To display, click the **Channels** tab, click a project channel, and then click **Filter**.



Filters are used to prevent noise (Noise, 98) and aliasing (Aliasing, 99).

Filter option	Abbreviation	Description
Automatic Anti-Aliasing	**(Auto)**	Automatic selection of a low-pass Bessel filter that uses about 15% of the sample rate as filter frequency, e.g., a 50 Hz Bessel filter for a sample rate of 300 S/s.
Use current device settings	**(Device filter)**	If EVIDAS does not configure the sample rate and filter automatically (Configuring the sample rate and filter of project channels automatically, 100), your last selection is remembered by the data acquisition device. The device settings are used.
Bypass	**(Off)**	No filter is used.
Select	**-**	You can select a filter, e.g., a 50 Hz Butterworth filter.

The abbreviations, e.g., **(Auto)**, are displayed in the **Filter** column of the **Project channels** panel.

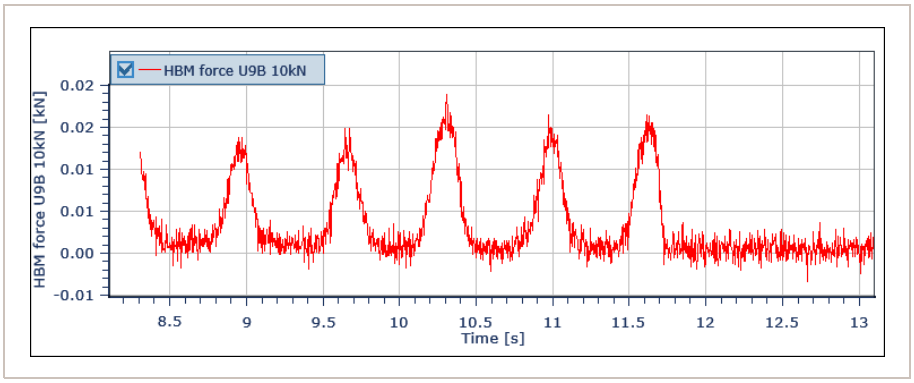


You want to apply a filter option to several project channels at the same time? On the **Project channels** panel, hold down Ctrl while clicking the channels you need, click **Filter**, and then select a filter option.

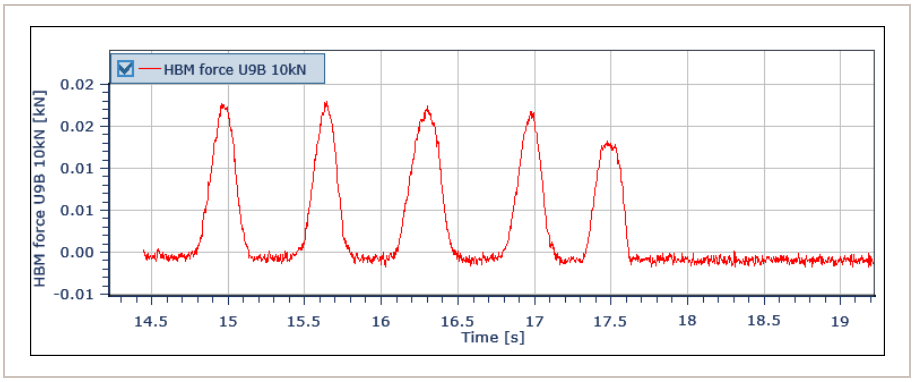
Noise

Noise is the entirety of random frequencies that disturb the perception of the expected signal.

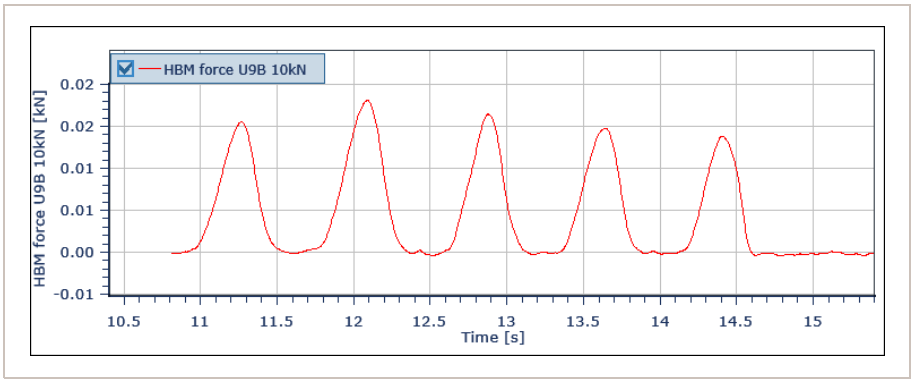
You can eliminate noise with low-pass filters that let frequencies below a specified threshold pass and eliminate the frequencies above it.



☐ Signal with low-pass Bessel filter, 500 Hz



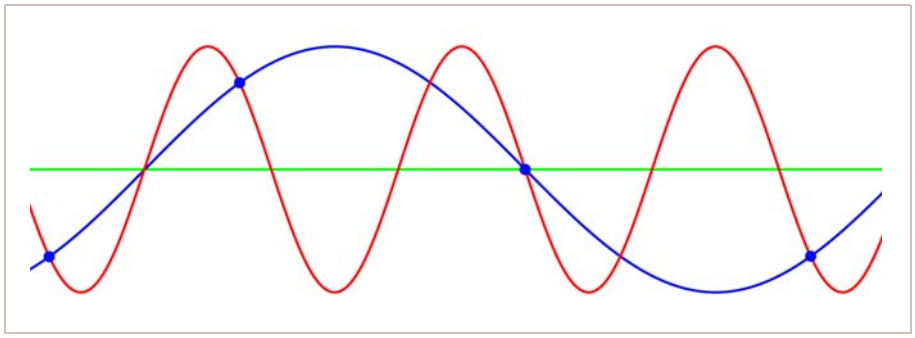
☐ Signal with low-pass Bessel filter, 50 Hz



☐ Signal with low-pass Bessel filter, 5 Hz

Aliasing

An alias is a trace (blue) that does not represent the signal (red) because the measured values are too few.



☐ Aliasing

Theoretically, if a function y(t) contains no frequencies higher than B Hertz, it is completely determined by giving its ordinates at a series of points spaced 1/(2B) seconds apart (Nyquist-Shannon sampling theorem).

For example, a 50 Hz signal is completely determined by measured values taken at points spaced 1/100 second apart, i.e., you need a sample rate of 100 S/s or higher.

HBM recommends to assign a sample rate about ten times the maximum frequency that you want to measure, i.e., for a 50 Hz signal to measure with 300 S/s or 600 S/s.

Also, the Nyquist-Shannon sampling theorem assumes that y(t) contains no frequencies higher than B Hertz. You must therefore eliminate with a low-pass filter any frequencies higher than the maximum frequency that you want to measure.

If EVIDAS configures the filter automatically, it uses a filter threshold of about 15% of the sample rate, e.g., a 50 Hz Bessel filter for a sample rate of 300 S/s.

4.1.9 Configuring the sample rate and filter of project channels automatically

You can define that the sample rate and filter of new project channels are configured automatically.

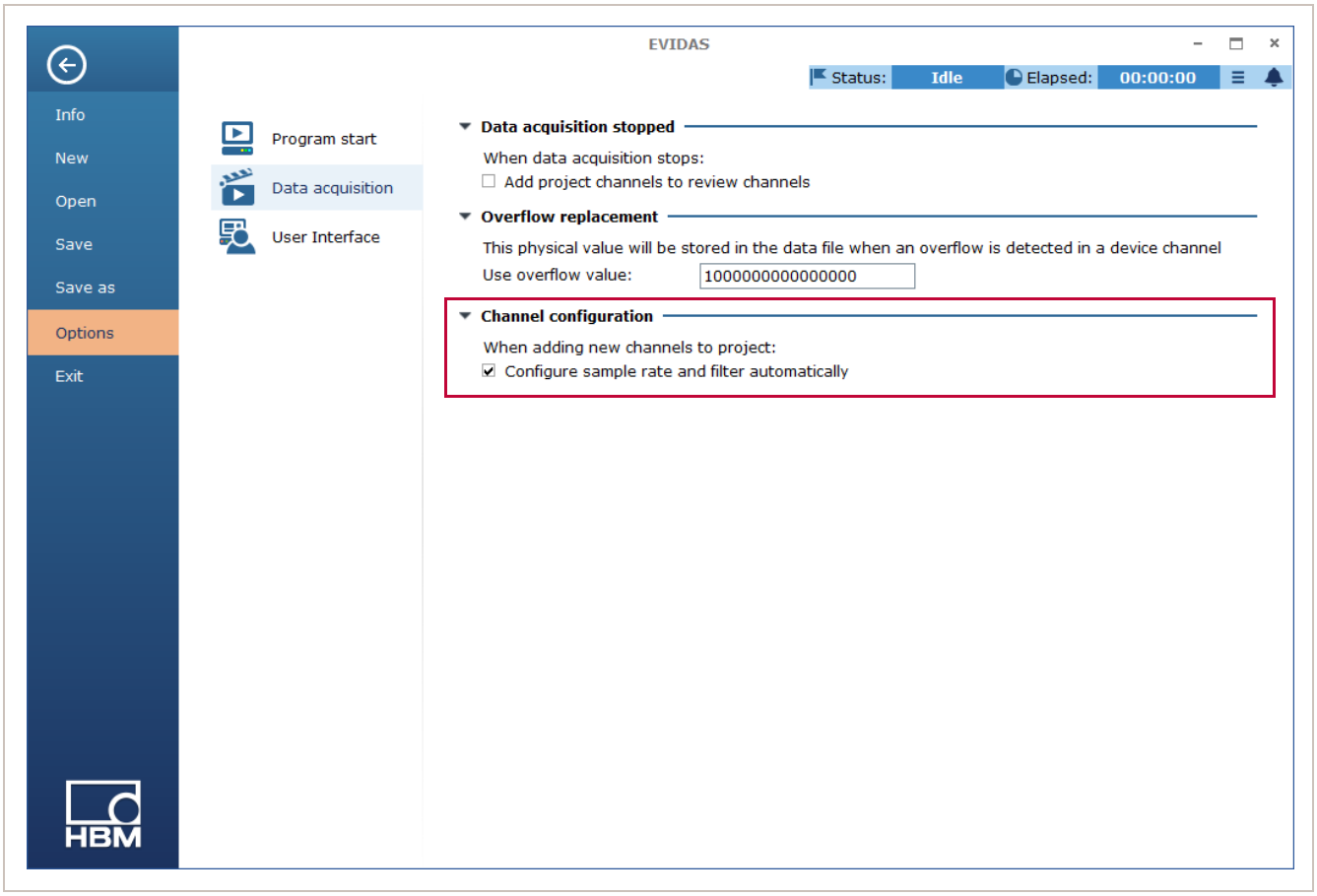
To configure the sample rate and filter of project channels automatically

1 Click the **File** tab.

2 Click **Options**.

3 Click **Data acquisition**.

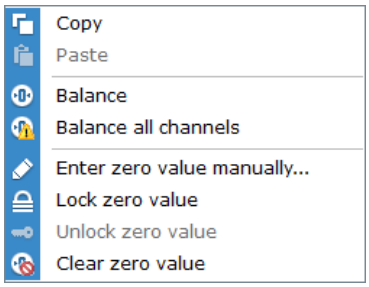
4 In **Channel configuration**, select the **Configure sample rate and filter automatically** checkbox.



ⓘ On the **Project channels** panel, the sample rate and filter of new project channels will be configured automatically.

4.1.10 Zero balance commands

To display, click the **Channels** tab, and then click **Balance** in the **Zero** group.



Before recording, you must zero balance the project channels because the measured values from the unloaded sensors are usually different from zero.

To eliminate this offset, you must determine a new zero value for each project channel either with a single-point reading or with a multi-point reading with averaging (Zero balance options, 103).

The zero value is displayed in the **Zero value** column and subtracted from the measured values.

Zero balance commands

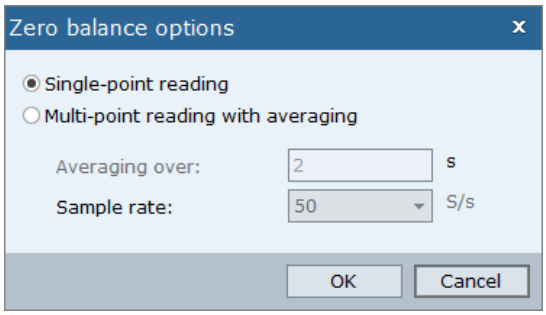
Command	Description
Copy	To copy a zero value.
Paste	To paste a zero value.
Balance	To zero balance a project channel. While balancing, make sure the sensor is unloaded. You can define how the zero value is determined (Zero balance options, 103). If you want to zero balance a selection of project channels, hold down Ctrl while clicking the project channels you need, and then click **Balance**.
Balance all channels	To zero balance all project channels. Project channels with locked zero values are not affected. While balancing, make sure the sensors are unloaded. You can define how the zero value is determined (Zero balance options, 103).
Enter zero value manually	To enter a zero value manually. Alternatively, click the cell in the **Zero value** column, type the zero value into the cell, and then press Enter.

Zero balance commands

Command	Description
Lock zero value	To lock a zero value. A is displayed. You cannot zero balance project channels with locked zero values. Use this function to secure the zero values you want to keep. You can use this command together with **Balance all channels** to zero balance a selection of project channels.
Unlock zero value	To unlock a zero value.
Clear zero value	To clear a zero value, i.e., to reset the zero value to **0**.

4.1.11 Zero balance options

To display, click the **Channels** tab, and then click the Dialog Box Launcher of the **Zero** group.



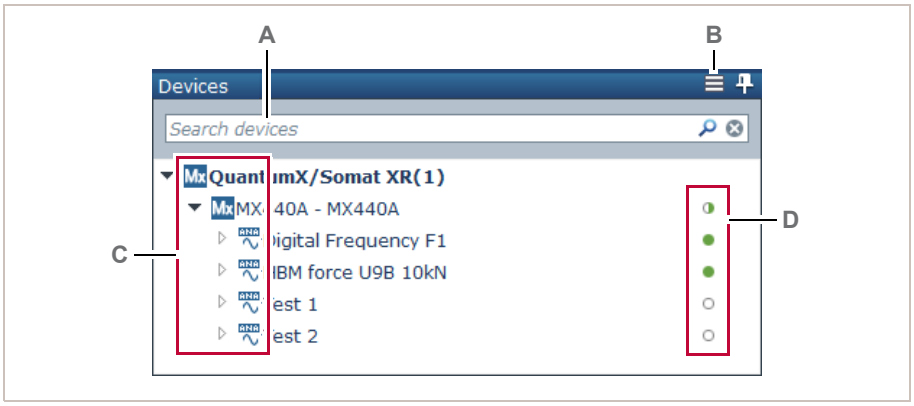
In this dialog box, you define how the zero values are determined.

Option	Description
Single-point reading	A single measured value from the project channel is taken as zero value.
Multi-point reading with averaging	The arithmetic mean of a defined number **n** of measured values $\mathbf{x}_i$ from the project channel is taken as zero value. $\bar{x} = \frac{1}{n}(x_1 + x_2 + ... + x_n) = \frac{1}{n}\sum_{i=1}^{n} x_i$ Use this option if the signal is slightly unstable. The number of measured values **n** used with this option is determined by the **Averaging over** time and the **Sample rate**, e.g., 2 s * 50 S/s = 100 S, i.e.,**n** = 100.

⇨ Zero balance commands, 101

4.2 Devices panel

To display, click the **Channels** tab.



A Search box, 104↗
B Burger menu, 104↗
C Device icons, 106↗
D Channel status icons, 106↗

The **Devices** panel displays the available data acquisition devices.

Search box

The search box enables you to filter the list of data acquisition devices (Search box, 47↗).

Burger menu

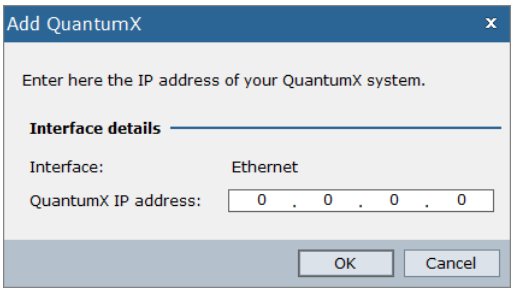
To display, click a data acquisition device, and then click .

The options displayed on the burger menu depend on whether you selected a channel or a module and on the type of channels you use, e.g., CAN channels.

Burger menu commands on Devices panel

Command	Description
Blink module LEDs on / off	To blink on / off the LEDs on the selected data acquisition device to support module identification.
Blink channel LED on / off	To blink on / off the LED of the selected channel to support channel identification.

Burger menu commands on Devices panel

Command	Description
Add device	To add a data acquisition device that is not recognized automatically, e.g., because the device is hidden behind a firewall. You must specify the IP address of the device, e.g.: 
Edit CAN Bus settings	To edit the settings of the CAN bus, e.g., the bit rate. This command is only displayed for CAN channels.
Change connector to analog input	To switch the first connector of a QuantumX MX840A or MX840B module from CAN mode to analog input. This command is only displayed for CAN channels.
Change connector to CAN bus	To switch the first connector of a QuantumX MX840A or MX840B module from analog input to CAN mode, i.e., to have 128 CAN channels available (Appendix B: CAN channels, 241). This command is only displayed for the first connector of the mentioned data acquisition devices.
Restore factory settings	To restore the factory settings of a data acquisition device, e.g., when an error occurs and you do not know which setting is causing it.
Rename device	To edit the name of a data acquisition device.
Change IP settings	To bypass the Dynamic Host Configuration Protocol (DHCP) that automatically assigns an IP address to each data acquisition device. You must specify an IP address for the device.
Update firmware	To update the firmware of the data acquisition device. You must update the firmware if the channel status icons turn orange ●.
Help on devices	To access context-sensitive help.

Device icons

	MGCplus data acquisition device.
	PMX data acquisition device.
	QuantumX or SomatXR data acquisition device.
	Analog channel.
	CAN channel.
	Digital channel in input mode.
	Digital channel in output mode.

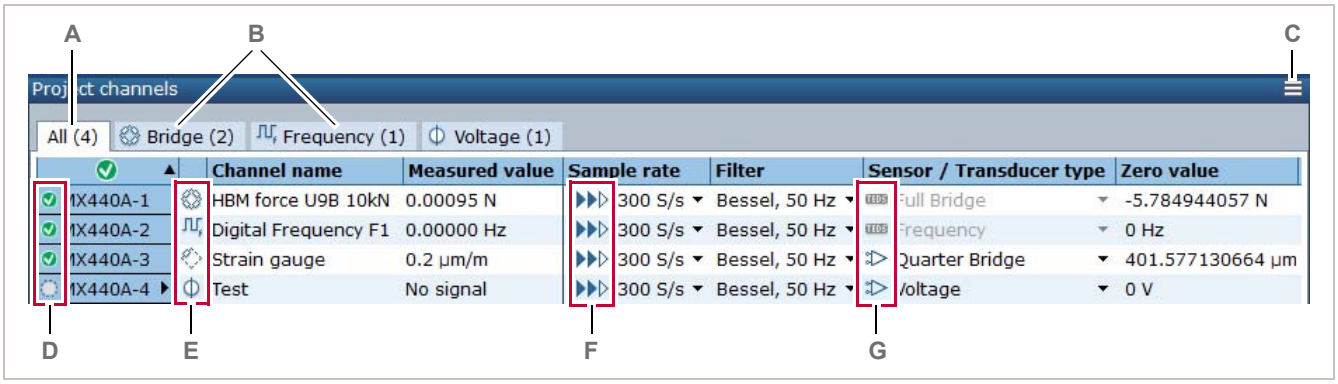
Channel status icons

	Some channels of the data acquisition device are used as project channels.
	Channel is not used as project channel.
	Channel is used as project channel.
	Firmware update is required.

⇨ Appendix B: CAN channels, 241
Appendix C: Digital channels, 253

4.3 Project channels panel

To display, click the **Channels** tab.



A All project channels, 108↗

B Subgroups of project channels, 108↗

C Burger menu, 108↗

D Signal status icons, 109↗

E Sensor icons, 231↗

F Sample rate icons, 110↗

G Sensor status icons, 110↗

This panel displays all project channels, i.e., the live project channels you selected in **Devices** (Devices panel, 104↗), and the online calculation channels you defined in **Online calculations** (Online calculations, 111↗).

Editing options

To edit the settings of a live project channel, use one of the following options:

Option	Description
Type	Click a cell without ▾ and edit the setting.
Select from drop-down list	Click ▾ in a cell and select a setting from the drop-down list.
Copy / Paste	Right-click a cell, click **Copy**, right-click the target cell, and then click **Paste**.
Fill command	Click a cell and point to the right lower corner. A red triangle is displayed. Point to the red triangle. When the pointer turns into a ↕+, drag the pointer to copy the setting to other project channels (Configure on Project channels panel, 67↗).

Sorting

To sort the **Project channels** table, click a header cell, and then click ▼ to sort in normal order or ▲ to sort in reverse order.

All project channels

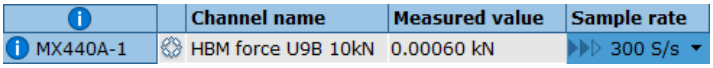
The **All** tab contains all project channels and displays basic information on each channel.

Column	Description
Channel type	The channel type string is composed of the data acquisition device type and the channel number on the device, e.g., **MX440A-1**.
Channel name	User-defined channel name.
Measured value	Live measured value. To change the notation of the measured value, right-click the value and select a notation.
Sample rate	Number of measured values transferred per second from the data acquisition device to EVIDAS (Sample rate group and domain options, 89).
Filter	Filter option (Filter options, 97).
Sensor / Transducer type	Sensor type (Sensors panel, 230).
Zero value	Zero value (Zero balance commands, 101).

Subgroups of project channels

A subgroup contains project channels with the same sensor type.

Burger menu

Command	Description
Clear all info notifications	To acknowledge and clear an info about a changed project channel setting. indicates that EVIDAS or a device has changed a project channel setting.  The background of the changed setting is highlighed. Info icons are displayed if the **Error handling** option is selected (Options panel, 224).
Show / hide columns	To define which columns are displayed. If you add a **Comment** column, any comment typed into this column will be displayed after data acquisition with the traceability data (Traceability data, 207) of the respective channel. If you have dragged a column from its default position, click **Show / hide columns**, and then click **Reset to default** to restore the default order.
Autosize columns	To display the full content of all columns.
Blink channel LED on / off	To blink on / off the LED of the selected channel to support channel identification.
Help on channel settings	To access context-sensitive help.

Signal status icons


Signal ok.


No signal.


Overflow (Options panel, 224).


A project channel setting has been changed by a device or by EVIDAS.

Point to for a report. Info icons are displayed if the **Error handling** option is selected (Options panel, 224).


No connection to data acquisition device.


Error. Point to for a report.

Sample rate icons

Slow sample rate.

Medium sample rate (default).

Fast sample rate.

Sensor status icons

Settings from database.

Settings have been edited (not saved yet).

Settings from device.

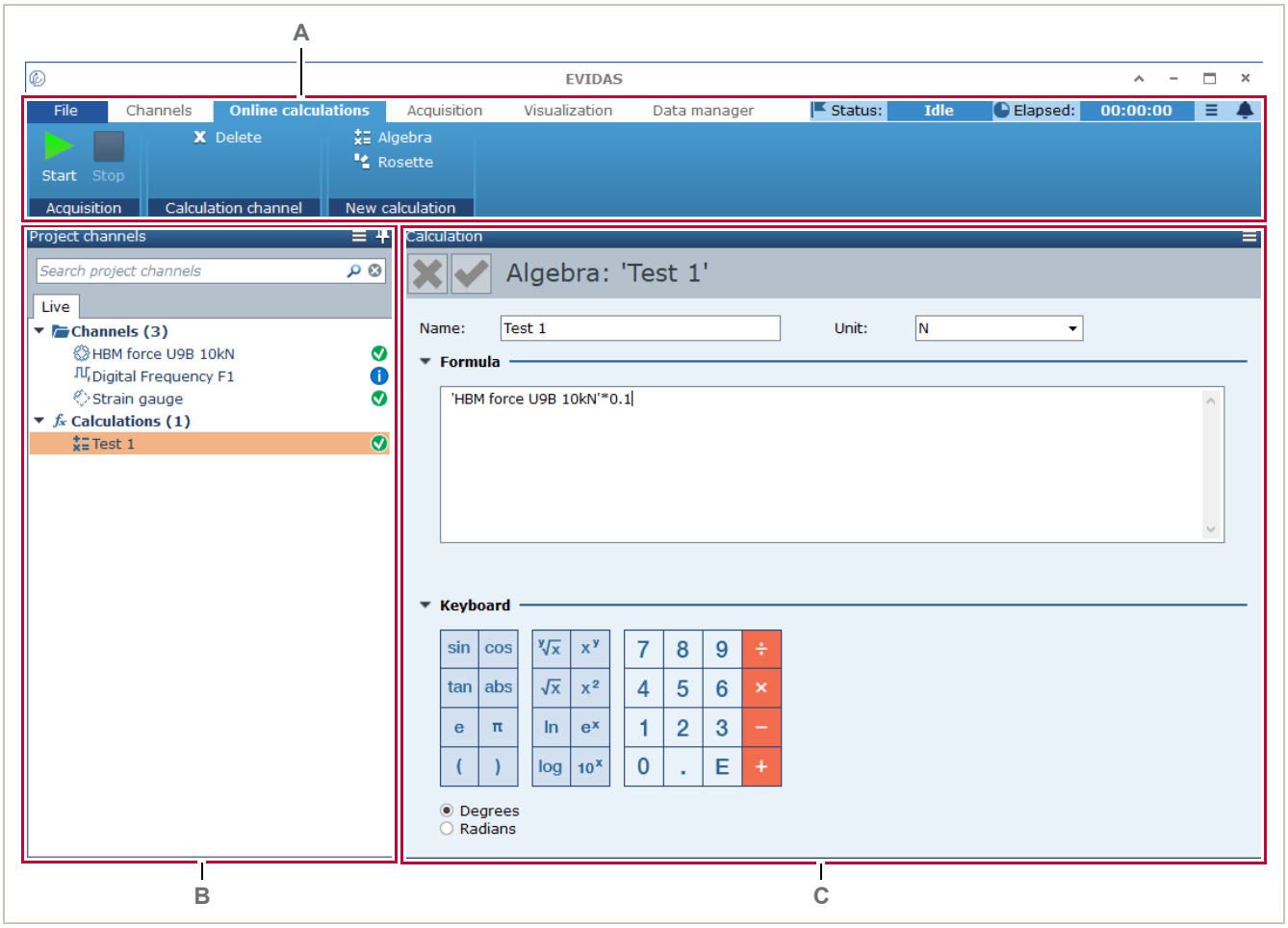
Settings from TEDS sensor.

⇨ Project channels panel, 113
Project channels panel in Visualization, 176
Project channels panel in Data manager, 210

5 Online calculations

On the **Online calculations** tab and its panels, you can define the following types of calculation channels:

- Algebra calculation channels (Algebra calculation panel, 116).
- Rosette calculation channels (Rosette calculation panel, 118).



A Online calculations tab, 112

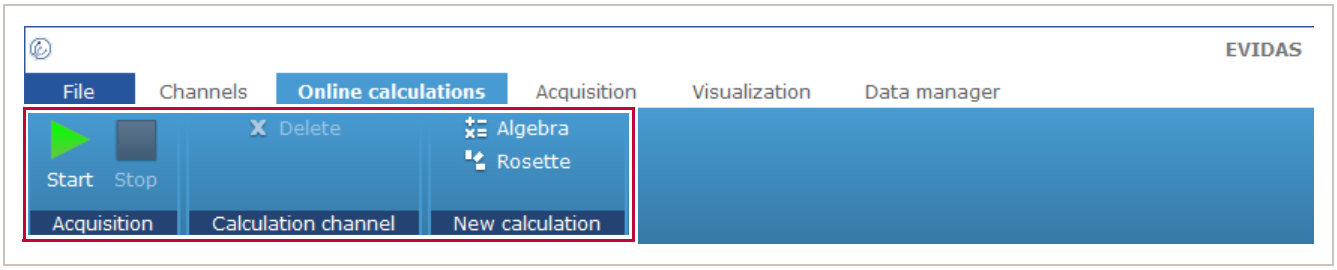
B Project channels panel, 113

C Calculation panel, 115

A calculation channel uses a formula to calculate in real time new values from values on other channels, e.g., Work **W** = Force **F** * displacement **s**.

Calculation channels are project channels.

5.1 Online calculations tab



You can define new calculation channels using various calculation methods.

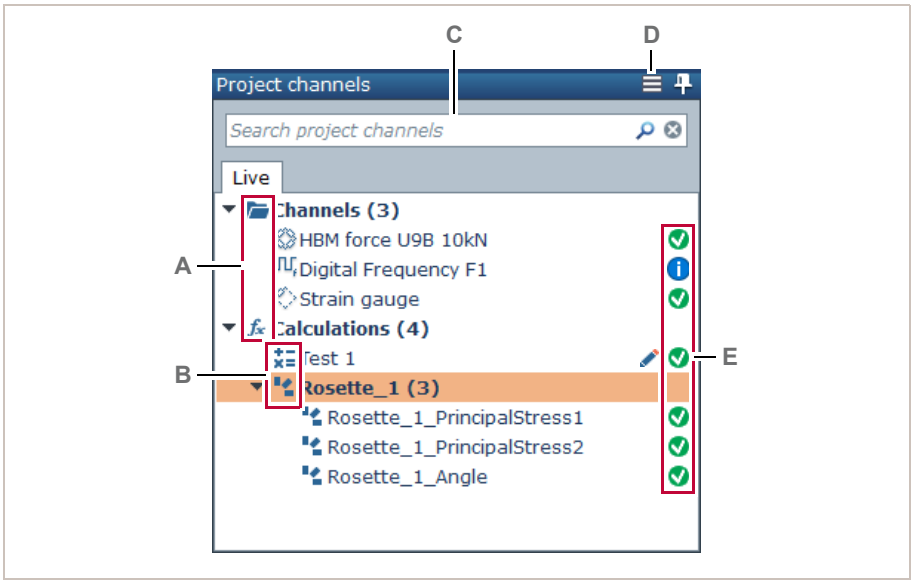
Online calculations commands

Group	Command	Description
Calculation channel	**Delete**	To delete a calculation channel. In **Project channels**, click the calculation channel you want to delete, and then click **Delete**.
New calculation	**Algebra**	To display the **Algebra** calculation panel (Algebra calculation panel, 116↗). On the **Algebra** calculation panel, you can define an algebra calculation channel (Defining an algebra calculation channel, 122↗).
	Rosette	To display the **Rosette** calculation panel (Rosette calculation panel, 118↗). On the **Rosette** calculation panel, you can define rosette calculation channels (Defining rosette calculation channels, 123↗).

⇨ Start / Stop button, 40↗

5.2 Project channels panel

To display, click the **Online calculations** tab.



A Folder icons, 113↗
B Calculation channel icons, 113↗
C Search box, 47↗
D Burger menu, 114↗
E Signal status icons, 109↗

The **Project channels** panel displays the live channels you selected as project channels and the calculation channels you defined.

Click a calculation channel to display its definition on the **Calculation** panel (Calculation panel, 115↗).

Click a calculation channel, and then press the Up Arrow key / Down Arrow key to subsequently display the definitions of the other calculation channels.

If you edit the definition of a calculation channel, a is displayed on the **Project channels** panel until you apply the changes by clicking **Apply** (Calculation panel header, 115↗).

Folder icons

Folder with live channels (collapsed).

Folder with live channels (expanded).

Folder with calculation channels.

Calculation channel icons

Algebra calculation channel.

Rosette calculation channel / Group of rosette calculation channels.

Burger menu

Command	Description
Help on project channels	To access context-sensitive help.

5.3 Calculation panel

On the **Calculation** panel, you can define the following types of calculation channels:

- Algebra calculation channels (Algebra calculation panel, 116).
- Rosette calculation channels (Rosette calculation panel, 118).

To display the respective calculation panel, click its icon on the **Online calculations** tab, e.g., **Rosette**.

5.3.1 Calculation panel header

The **Calculation** panel header is the same for all calculation panels.

A Cancel button, 115

B Apply button, 115

C Burger menu, 115

The buttons to the left are active, i.e., displayed in blue, if you edit the definition of a calculation channel. Otherwise, the buttons are displayed in grey, e.g., .

Cancel button

To cancel any changes on the **Calculation** panel since you last saved the definition of the calculation channel.

If you want to define a new calculation channel, click the respective icon on the **Online calculations** tab, e.g., **Rosette**.

Apply button

To save the definition of a calculation channel.

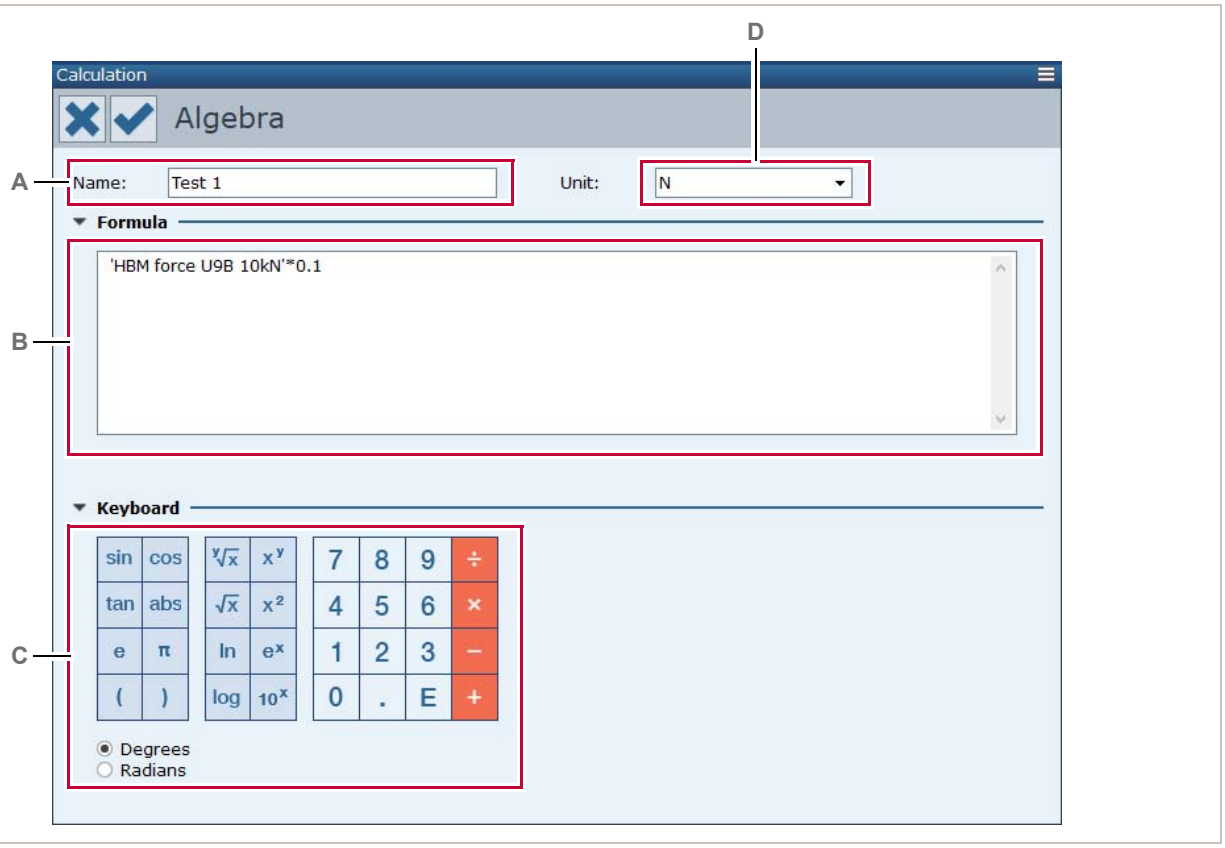
If you defined a new calculation channel, it is now displayed on the **Project channels** panel (Project channels panel, 113).

Burger menu

Command	Description
Help on online calculations	To access context-sensitive help.

5.3.2 Algebra calculation panel

To display, click the **Online calculations** tab, and then click **Algebra**.



A Channel name, 116

B Formula, 116

C Algebra keyboard, 117

D Formula output unit, 117

On the **Algebra** calculation panel, you can create and edit algebra calculation channels, i.e., calculation channels with user-defined formulas.

Channel name

For a new algebra calculation channel, enter a unique name. The default names are **Formula_1**, **Formula_2**, **Formula_3**, etc.

Formula

The formula determines how the output values of the algebra calculation channel are calculated from the values on its input channels.

The formula must contain at least one live channel or calculation channel as input channel.

To insert a channel, drag it from the **Project channels** panel (Project channels panel, 113) to the **Formula** text box. The channel is represented in apostrophes.

If you use several input channels, the channels must have the same sample rate.

Algebra keyboard

Use the algebra keyboard for support with algebraic expressions, e.g., **sqrt((‘Channel A’ ^ 2) + (‘Channel B’ ^ 2))**.

Place the cursor at the desired position in the formula, and then click the respective button to insert a number, an operator, or a function.

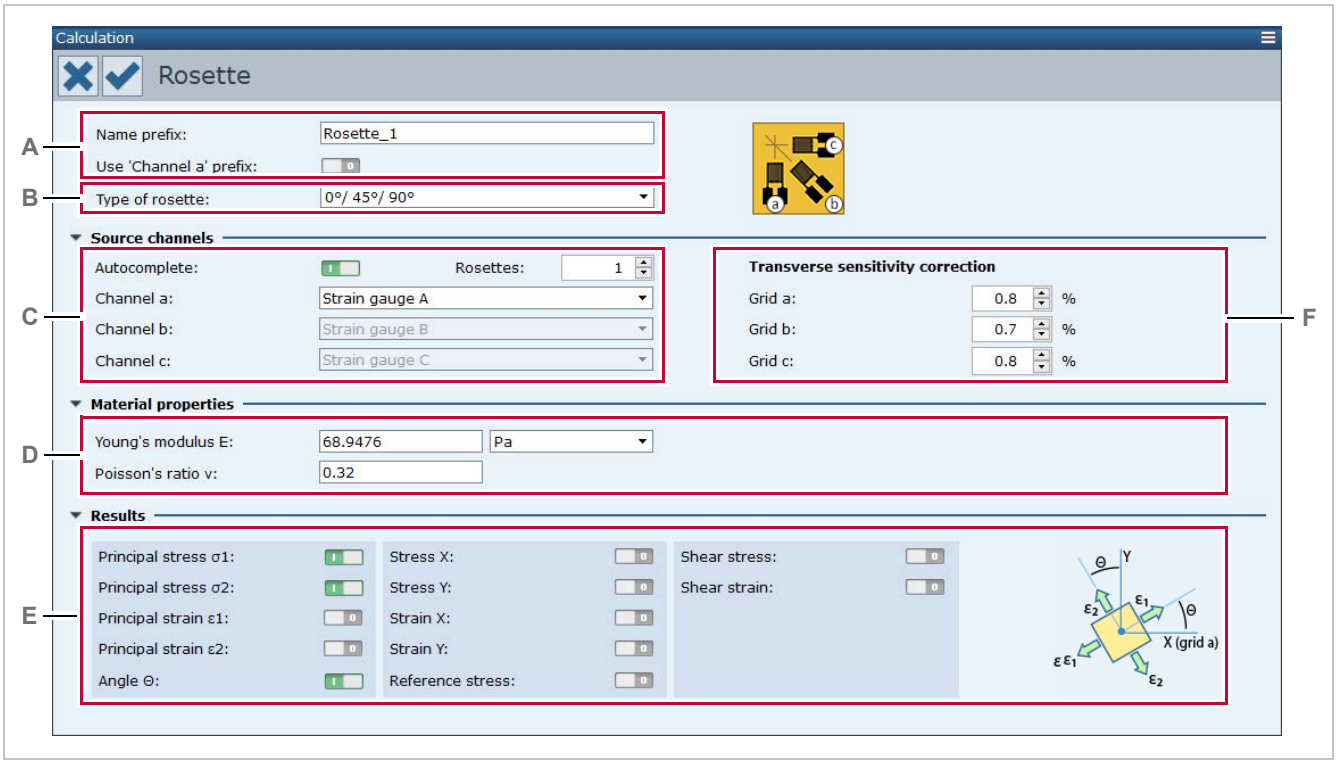
Formula output unit

Use this combo box to specify the output unit of the algebra calculation channel.

⇨ Defining an algebra calculation channel, 122

5.3.3 Rosette calculation panel

To display, click the **Online calculations** tab, and then click **Rosette**.



A Name prefix, 119↗

B Type of rosette, 119↗

C Source channels, 120↗

D Material properties, 120↗

E Results, 121↗

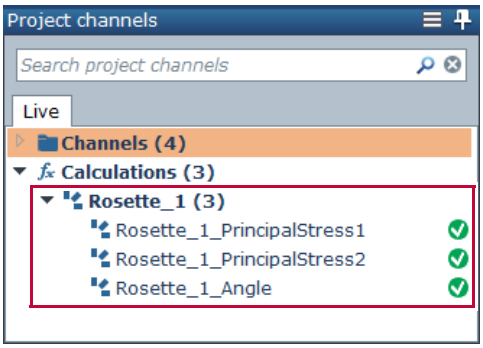
F Transverse sensitivity correction, 121↗

On the **Rosette** calculation panel, you can create and edit rosette calculation channels. The rosette formulas are predefined.

Name prefix

For each rosette, enter a unique name. The default names are **Rosette_1**, **Rosette_2**, **Rosette_3**, etc.

This name is then used as group name for a set of rosette calculations, e.g., **Rosette_1**, and as prefix for the rosette calculations included in the group, e.g., **Rosette_1_PrincipalStress1**.



If you select **Use 'Channel a' prefix** = , the channel name specified in **Channel a** is used as rosette name and prefix.

If you select **Use 'Channel a' prefix** = , **Autocomplete** = , and **Rosettes** > 1, the channel name specified in **Channel a** of *each* rosette is used as rosette name and prefix for the respective rosette (Source channels, 120).

Type of rosette

Select the type of rosette you use.

The drop-down list includes general types of rosettes, e.g., **0°/45°/90 a-b-c**, as well as specific HBM rosettes, e.g., **HBM RY1 series**.

The diagram to the right displays the type of rosette you selected.

Source channels

In **Channel a**, **Channel b**, and **Channel c**, you assign the project channels that correspond to the strain gauges on the rosette.

You have the following options to assign project channels to rosettes:

Options for assigning project channels to rosettes

Option	Autocomplete toggle switch	Rosettes spin box	Description
Assign manually		1	Select the project channels from the **Channel a**, **Channel b**, and **Channel c** drop-down lists. Alternatively, assign the project channels via drag-and-drop from the **Project channels** panel.
Autocomplete one rosette		1	Assign the project channel with the signal from strain gauge **a** on the rosette to **Channel a**. The next two project channels on the **Project channels** panel are filled in automatically to **Channel b** and **Channel c**.
Autocomplete several rosettes		n	Assign the project channel with the signal from strain gauge **a** on the *first* rosette to **Channel a**. The next two project channels on the **Project channels** panel are filled in automatically to **Channel b** and **Channel c**. Also, the next (n-1)*3 project channels on the **Project channels** panel are filled in automatically for the next n-1 rosettes.

For T-rosettes, you assign two project channels, one to **Channel a** and one to **Channel b**.

A project channel can only be assigned to a rosette if its sensor is a strain gauge.

If you autocomplete several rosettes, you can check the project channels assigned to each rosette after clicking  **Apply**.

Material properties

Type in Young's modulus **E** and Poisson's ratio **v** for the material to which the rosette is attached to.

Results

List of available rosette calculations.

Select to include the respective rosette calculation, select to exclude it.

Once you click **Apply**, for each rosette calculation you selected, one rosette calculation channel is created .

Transverse sensitivity correction

The transverse sensitivity parameters specify for each strain gauge on the rosette a percentage as correction for the angular deviation of the strain gauge in relation to the exact intended angle.

To take into account this angular deviation, enter the transverse sensitivity parameters from the rosette data sheet, e.g.:

A Transverse sensitivity parameters on a rosette data sheet

⇨ Defining rosette calculation channels, 123

5.4 Defining calculation channels

5.4.1 Defining an algebra calculation channel

Prerequisites

- Data acquisition device and sensors are connected.
- Project channels are selected (To select project channels, 59).

To define a calculation channel

1 Click the **Online calculations** tab, and then click **Algebra**.

2 In **Name**, type a name for the new calculation channel.

3 In **Unit**, select the output unit of the formula.

4 In **Formula**, enter the formula that calculates the output values.

ⓘ The formula must contain at least one live channel or calculation channel as input channel.

ⓘ To insert an input channel, drag it from the **Project channels** panel into the **Formula** text box.

ⓘ Use the algebra keyboard for support with algebraic expressions (Algebra keyboard, 117).

ⓘ If you use several input channels, make sure the channels have the same sample rate (Formula, 116).

5 Click **Apply**.

✓ The formula is saved and the new algebra calculation channel is displayed on the **Project channels** panel.

5.4.2 Defining rosette calculation channels

Prerequisites

- Data acquisition device and rosette are connected.
- Rosette channels are selected as project channels (To select project channels, 59).

To define rosette calculation channels

1 Click the **Online calculations** tab, and then click **Rosette**.

ⓘ To minimize your work when defining new rosette calculation channels, on the **Project channels** panel, click a similar rosette calculation channel in order to display its definition on the **Calculation** panel. Then click **Rosette**. The definition of the previous rosette is kept on the **Calculation** panel, except for the name prefix and the source channels.

2 In **Name prefix**, enter a name for the group of rosette calculations.

ⓘ This name is also used as prefix for the rosette calculation channels, e.g., **Rosette_1_PrincipalStress1**.

ⓘ If you want to use the name of **Channel a** instead, select **Use 'Channel a' prefix** = .

3 In **Type of rosette**, select the type of rosette you use, e.g., **0°/45°/90° a-b-c**.

4 In **Channel a**, **Channel b**, and **Channel c**, assign the project channels that correspond to the strain gauges on the rosette.

ⓘ If you want to manually assign the project channels, select **Autocomplete** = , and then select the project channels from the **Channel a**, **Channel b**, and **Channel c** drop-down lists. Alternatively, assign the project channels via drag-and-drop from the **Project channels** panel.

ⓘ If the rosette project channels are listed subsequently on the **Project channels** panel, you can autocomplete all but the first source channel: Select **Autocomplete** = and in **Rosettes** the number of rosettes you want to create calculation channels for, e.g., **3**, and then, in **Channel a**, assign the project channel with the signal from strain gauge **a** on the first rosette.
The project channels for the first rosette and all subsequent rosettes are filled in automatically (Source channels, 120).

5 In **Young's modulus E** and **Poisson's ratio v**, type in the respective physical constants of the material to which the rosette is attached.

6 Under **Results**, select to include a rosette calculation, or to exclude it.

7 In **Grid a**, **Grid b**, and **Grid c**, enter the transverse sensitivity parameters of the rosette from the rosette data sheet (Transverse sensitivity correction, 121).

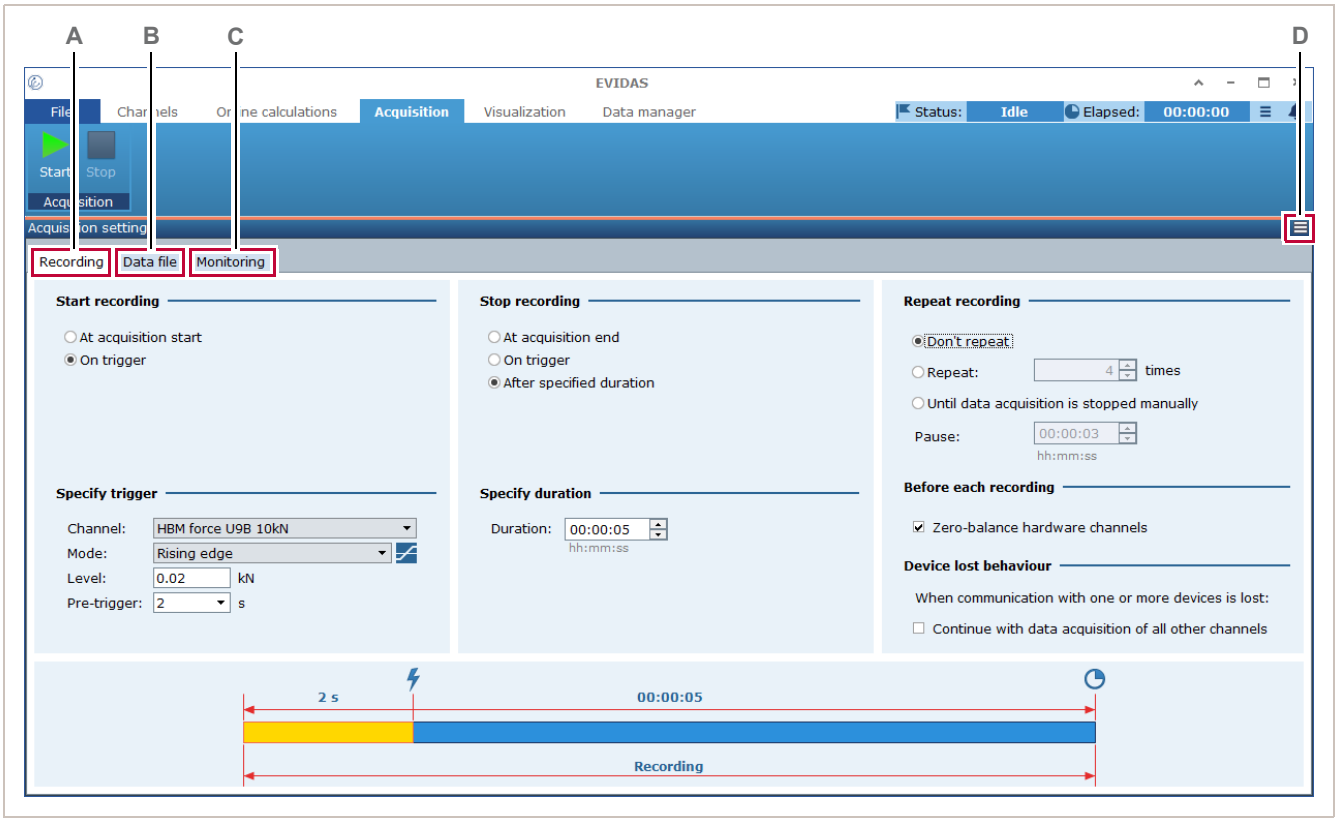
8 Click **Apply**.

✓ A group of rosette calculation channels is created and displayed on the **Project channels** panel.

6 Acquisition

On the **Acquisition** tab and its panels, you define the following:

- Start and stop conditions of a recording.
- Location where the measured values are stored.
- Optionally, a connection to an IoT platform to which measured values of selected project channels are streamed at defined intervals.



A Recording options, 126

B Data file options, 136

C Monitoring options, 141

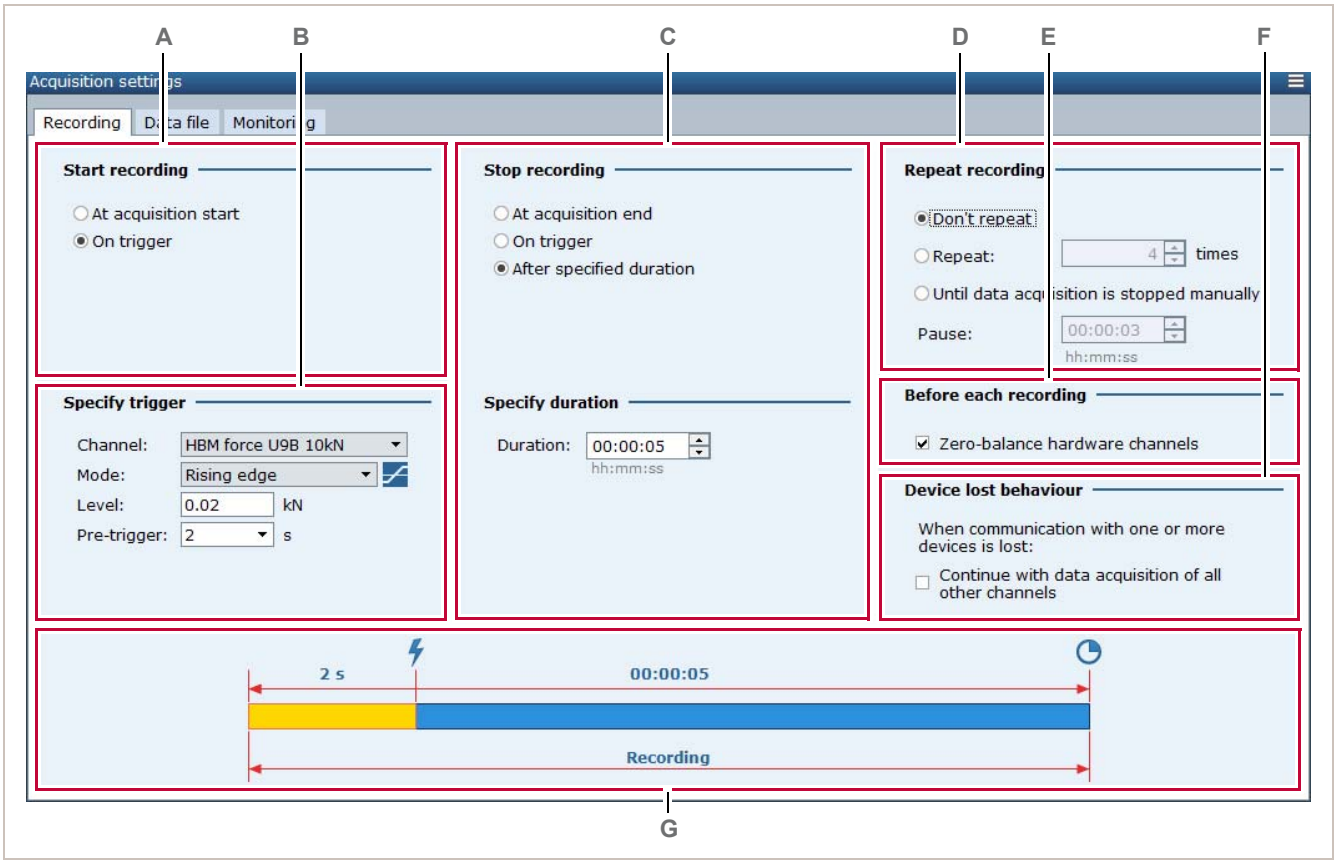
D Burger menu, 125

Burger menu

Command	Description
Help on recording	To access context-sensitive help.
Help on data file	To access context-sensitive help.
Help on monitoring	To access context-sensitive help.

6.1 Recording options

To display, click the **Acquisition** tab, and then click the **Recording** tab.



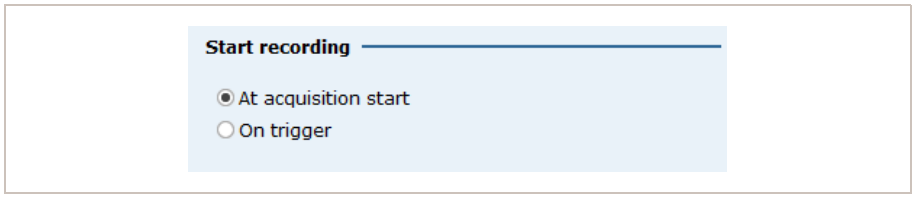
- **A** Start options, 127
- **B** Trigger options, 129
- **C** Stop options, 128
- **D** Repeat recording options, 133
- **E** Zero-balance option, 133
- **F** Device lost behavior, 134
- **G** Time line, 134

On the **Recording** tab, you define how the recording is started and stopped, whether it is repeated, and whether the channels are zero-balanced before each recording.

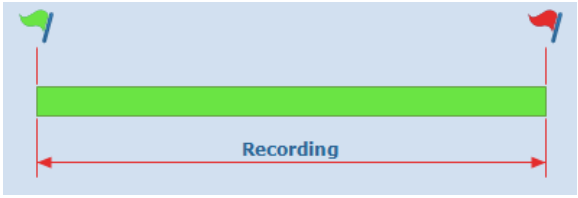
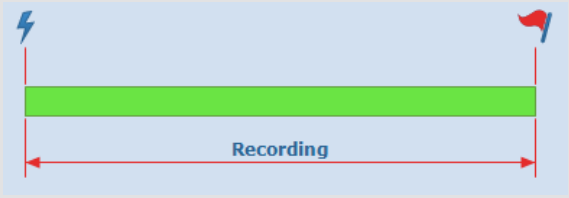
A time line visualizes the start / stop events you defined and the composition of the total recording time.

6.1.1 Start options

To display, click the **Acquisition** tab, and then click the **Recording** tab.



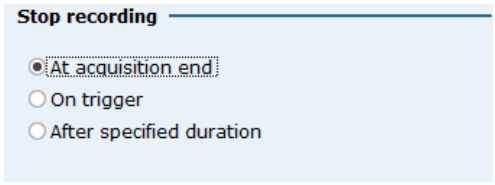
Start options define how the recording is started.

Option	Description
At acquisition start	To manually start recording by clicking **Start**. 
On trigger	To automatically start recording as soon as a trigger condition is fulfilled (Trigger options, 129↗).  You must still click **Start** to start monitoring the trigger channel until the trigger condition is fulfilled.

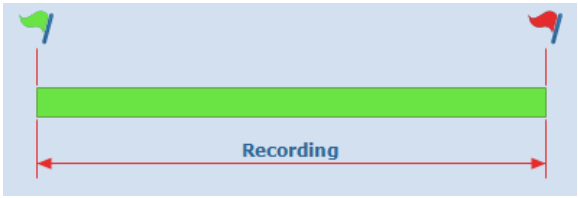
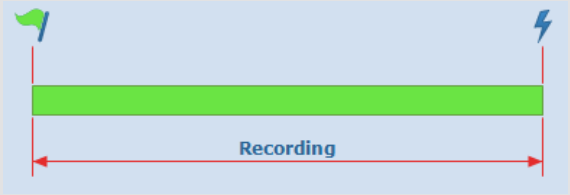
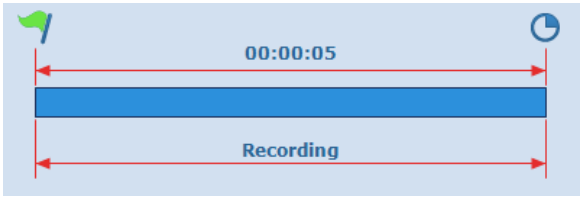
⇨ Time line icons, 135↗

6.1.2 Stop options

To display, click the **Acquisition** tab, and then click the **Recording** tab.



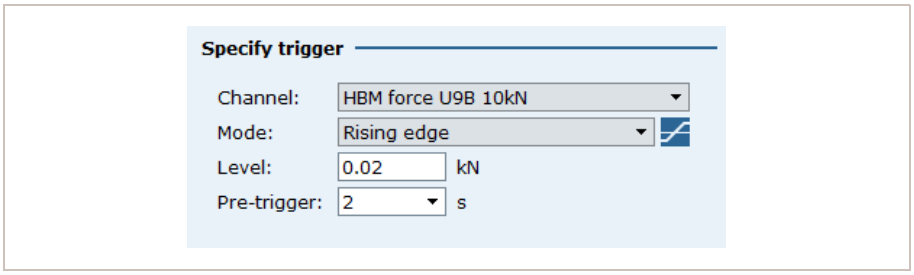
Stop options define how the recording is stopped.

Option	Description
At acquisition end	To manually stop recording by clicking **Stop**. 
On trigger	To automatically stop recording as soon as a trigger condition is fulfilled (Trigger options, 129). 
After specified duration	To stop recording after a specified time. 

⇨ Time line icons, 135

6.1.3 Trigger options

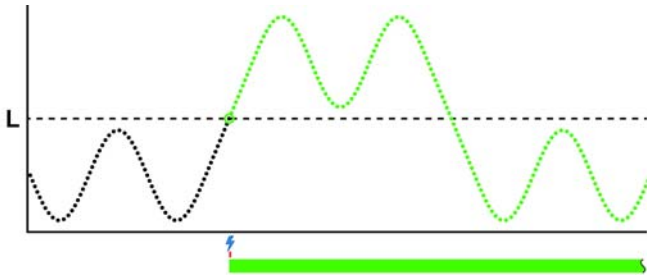
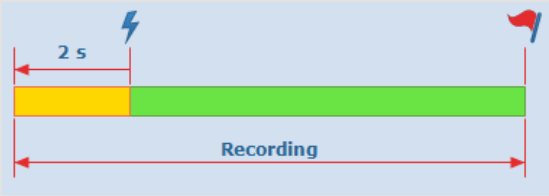
To display, click the **Acquisition** tab, and then click the **Recording** tab.



You can define start and stop triggers.

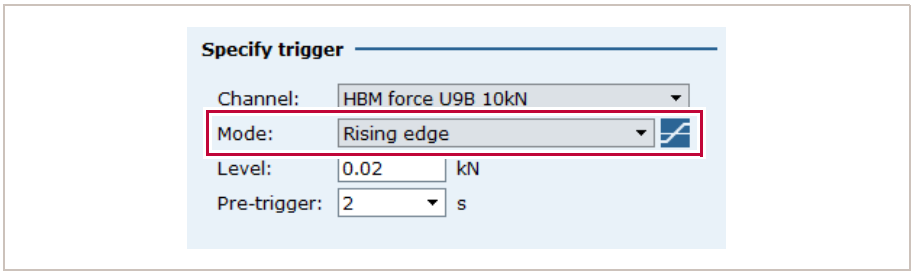
If you define a trigger to start or stop recording, EVIDAS monitors the trigger channel and checks whether the trigger condition is fulfilled.

Once the trigger condition is fulfilled, EVIDAS starts or, respectively, stops recording on all project channels.

Trigger option	Mode	Description
Channel		The channel to be monitored to check whether the trigger condition is fulfilled.
Mode	**Above level**	Recording is started as soon as a measured value **Y** ≥ trigger level **L** (Trigger modes, 131↗).
	Below level	Recording is started as soon as a measured value **Y** ≤ trigger level **L** (Trigger modes, 131↗).
	Rising edge	Recording is started if the measured values have been below the trigger level **L** for some time and then a measured value **Y** ≥ **L** (Trigger modes, 131↗).
	Falling edge	Recording is started if the measured values have been above the trigger level **L** for some time and then a measured value **Y** ≤ **L** (Trigger modes, 131↗).
Level		In the **Level** box, you define the trigger level **L**. 
Pre-Trigger		To define a time period in which data is already recorded prior to the start trigger condition being fulfilled.  This option is displayed if you specify a start trigger (Start options, 127↗).
Post-Trigger		To define a time period in which data is recorded after the stop trigger condition has been fulfilled.  This option is displayed if you specify a stop trigger (Stop options, 128↗).

Trigger modes

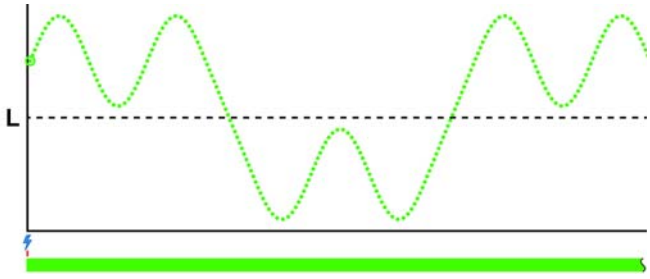
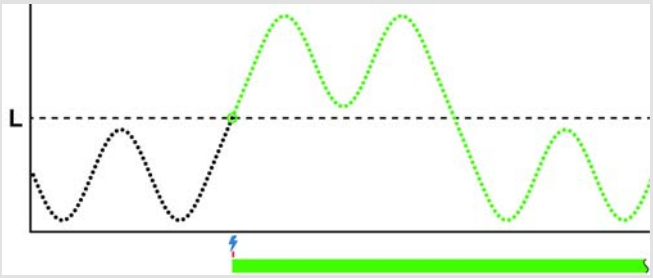
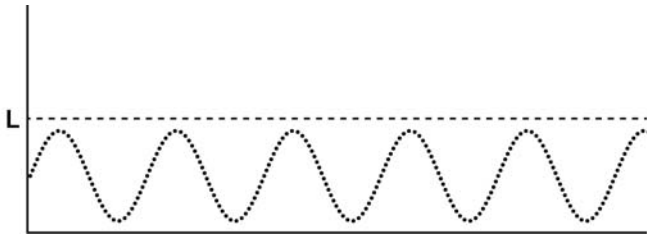
To display, click the **Acquisition** tab, and then click the **Recording** tab.



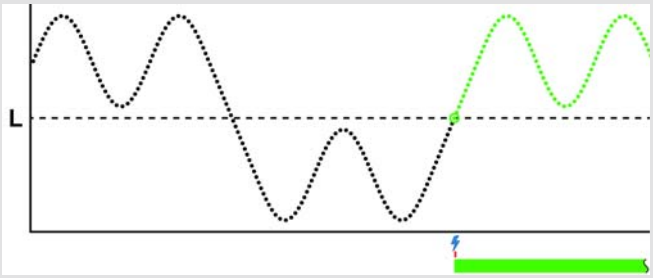
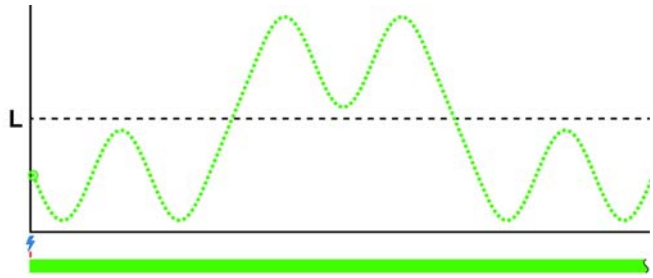
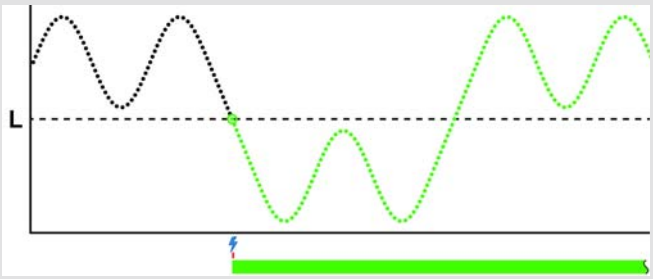
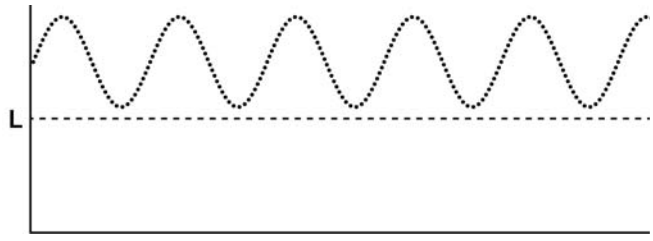
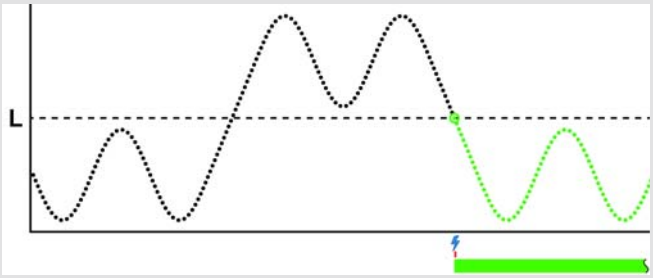
The trigger mode and trigger level **L** (Trigger options, 129) define an event on the trigger channel that starts or stops recording on all project channels.

The following table displays how trigger modes affect the start of the recording. To stop recording, the trigger modes work analogously.

Trigger modes

Trigger mode	Description	Illustration
Above level	If you click **Start** and the first measured value **Y** ≥ **L**, recording is started immediately.	
	If the measured values are below **L** initially, recording is started as soon as a measured value **Y** ≥ **L**.	
	If the measured values are continously below **L**, recording is not started.	

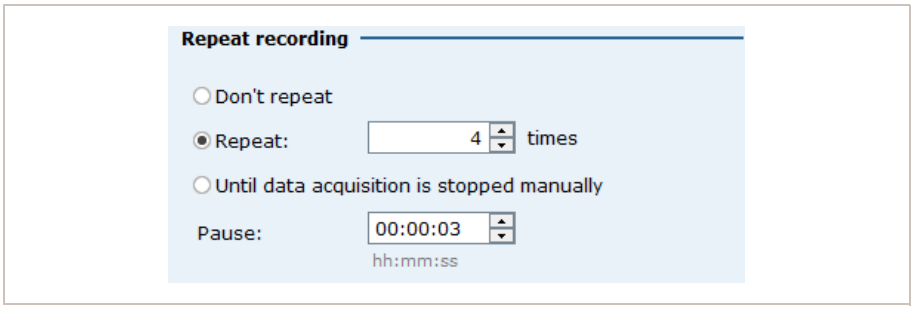
Trigger modes

Trigger mode	Description	Illustration
Rising edge	If the measured values are below **L** for some time and then a measured value **Y** ≥ **L**, recording is started.	
Below level	If you click **Start** and the first measured value **Y** ≤ **L**, recording is started immediately.	
	If the measured values are above **L** initially, recording is started as soon as a measured value **Y** ≤ **L**.	
	If the measured values are continously above **L**, recording is not started.	
Falling edge	If the measured values are above **L** for some time and then a measured value **Y** ≤ **L**, recording is started.	

⇨ Time line icons, 135

6.1.4 Repeat recording options

To display, click the **Acquisition** tab, and then click the **Recording** tab.



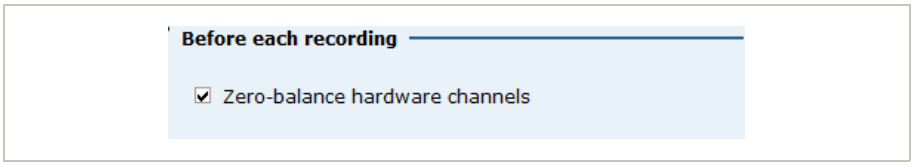
The **Repeat recording** options are active if the recording stops automatically, i.e., either after a specified duration, or when a stop trigger condition is fullfilled.

Option	Description
Don't repeat	The recording is performed once and is not repeated.
Repeat	The recording is performed once and is then repeated n times. You can stop recording at any time by clicking **Stop**.
Until data acquisition is stopped manually	The recording is performed once and is then repeated endlessly until data acquisition is stopped by clicking **Stop**.
Pause	Delay between repeated recordings.

Each recording is stored in a separate EVIDAS data file (*.pnrf) and optionally in other formats as well (File formats, 139).

6.1.5 Zero-balance option

To display, click the **Acquisition** tab, and then click the **Recording** tab.

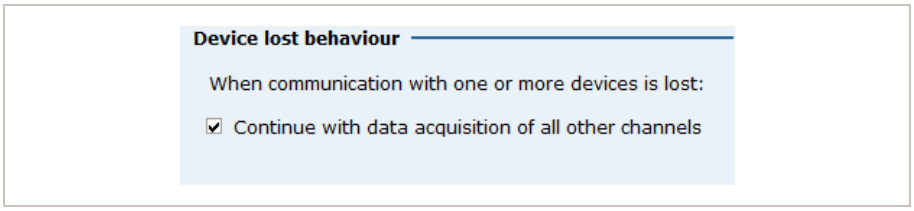


Option	Description
Zero-balance hardware channels	The project channels are zero-balanced automatically before each recording.

⇨ Zero balance commands, 101
Zero balance options, 103

6.1.6 Device lost behavior

To display, click the **Acquisition** tab, and then click the **Recording** tab.



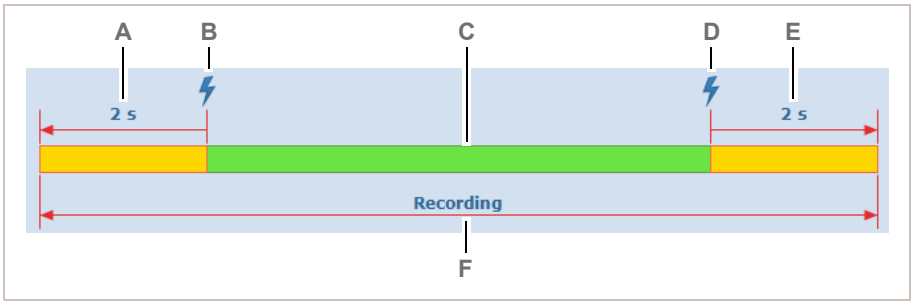
Option	Description
Continue with data acquisition of all other channels	Data acquisition continues for all active channels even if communication with one or more devices is lost.

6.1.7 Time line

To display, click the **Acquisition** tab, and then click the **Recording** tab.

Example 1: Time line with triggers

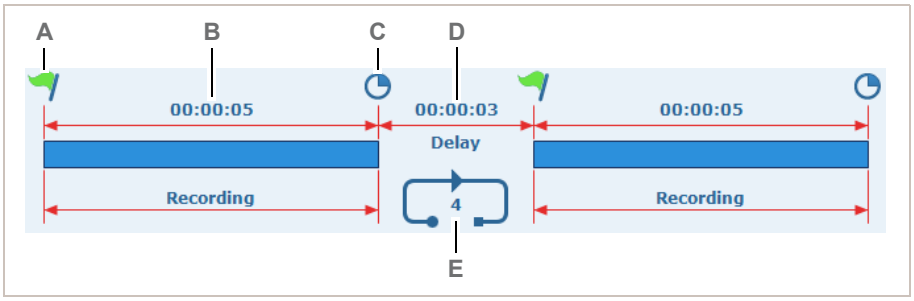
The time line visualizes the start / stop events you defined and the composition of the total recording time.



A Pre-trigger recording time
B Start trigger
C Main recording time
D Stop trigger
E Post-trigger recording time
F Total recording time

Example 2: Time line with repeated recordings

In addition to the time line of the recording, the delay between recordings and the number of repetitions are displayed.



A Start of recording	D Delay between recordings
B Specified duration of recording	E Number of repetitions
C Stop of recording	

Time line icons

Recording is started manually by clicking **Start**.

For repeated recordings, also indicates the start of a repeated recording.

Recording is started / stopped automatically with a trigger condition (Trigger options, 129).

Recording is stopped after a specified duration.

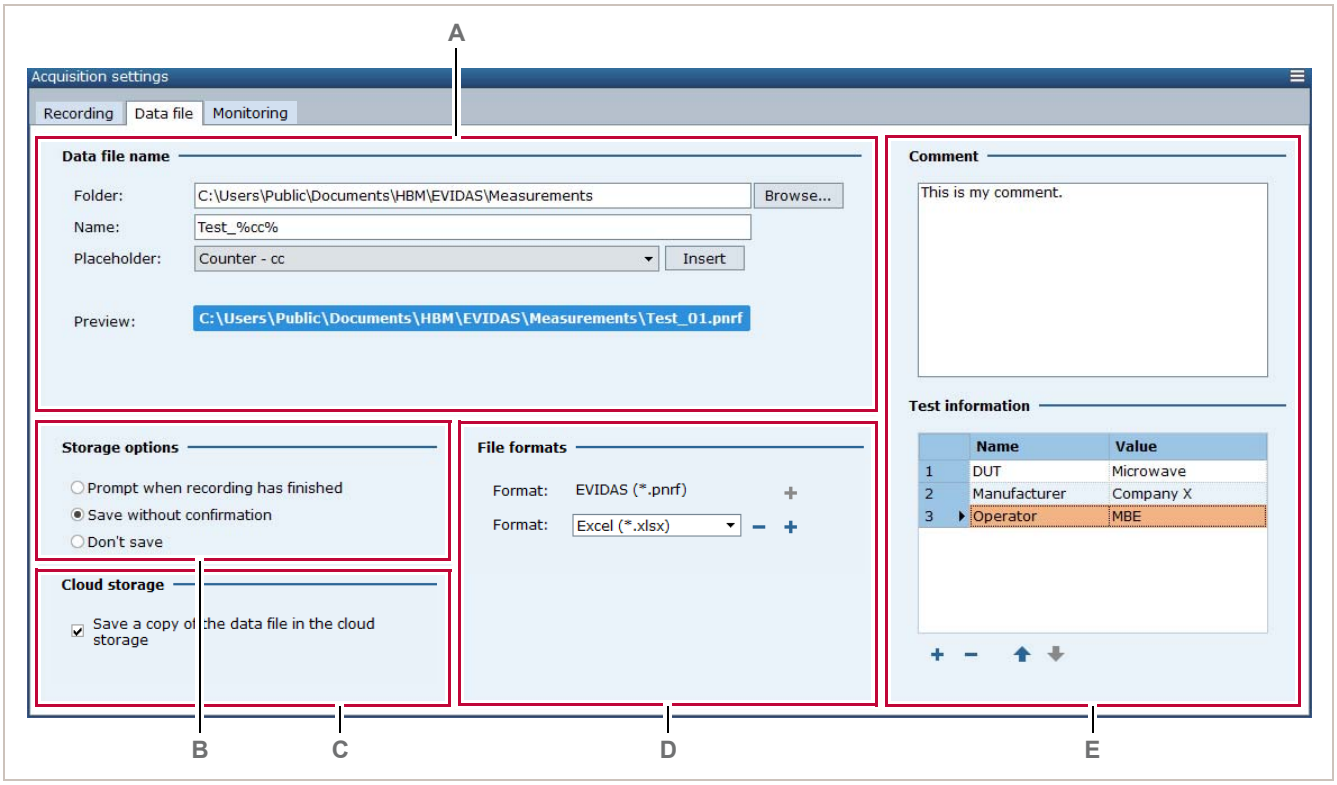
Recording is stopped manually by clicking **Stop**.

Recording is repeated automatically, e.g., 4 times.

Recording is repeated endlessly until data acquisition is stopped manually by clicking **Stop**.

6.2 Data file options

To display, click the **Acquisition** tab, and then click the **Data file** tab.



A Data default folder and file name, 136

B Storage options, 138

C Cloud storage, 138

D File formats, 139

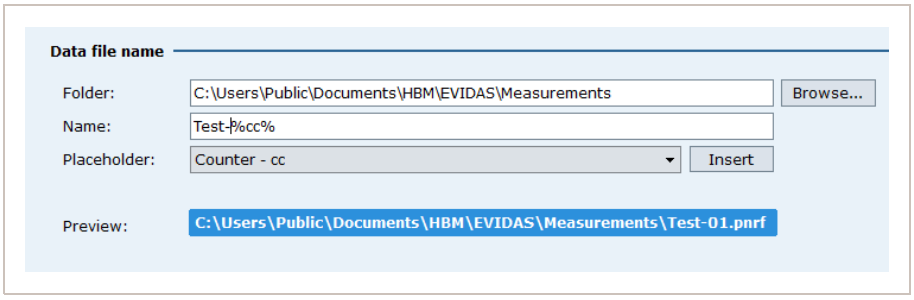
E Metadata, 140

On the **Data file** tab, you define where the measured values are stored.

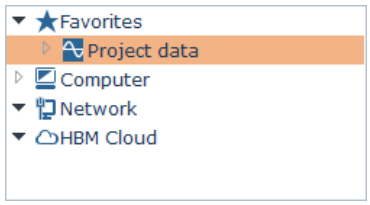
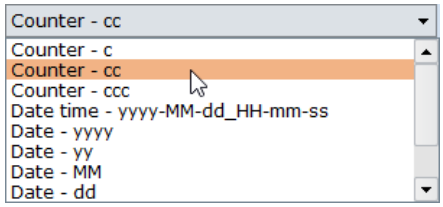
You can save data in several file formats simultanously and add metadata to each file.

6.2.1 Data default folder and file name

To display, click the **Acquisition** tab, and then click the **Data file** tab.

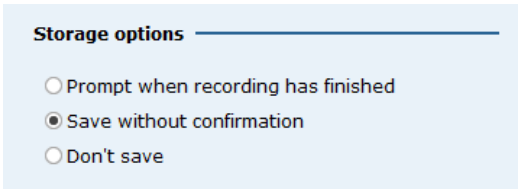


After recording, if you want to save the data without further confirmation (Storage options, 138), you can define a default folder and file names for the data files.

Option	Description
Folder	You can define a default folder for data files, e.g.: D:\Project_A If you do not define a default folder, EVIDAS uses the following default: C:\Users\Public\Documents\HBM\EVIDAS\Measurements After recording, you can access the data files in the **Project data** folder in **Data manager** (File browser, 205). 
Name	In the filename, you can include counters and date / time information via placeholders. Test_%yyyy%-%MM%-%dd%_%cc%
Placeholder	Placeholders make sure that each recording has a unique filename, e.g., **Test_%cc%** generates **Test_01**, **Test_02**, **Test_03**, etc. for subsequent recordings. Name / Creation date / Size Test_03.pnrf / 2018-10-31 08:36 / 1.4 MB Test_02.pnrf / 2018-10-31 08:36 / 1.4 MB Test_01.pnrf / 2018-10-31 08:36 / 1.4 MB If you do not insert a placeholder, EVIDAS inserts a counter automatically, e.g., **Test**, **Test(2)**, **Test(3)**, etc. To insert a placeholder, click on a position in the filename, e.g., at the end, select a placeholder from the **Placeholder** drop-down list, and then click **Insert**.  You can also use placeholders in the folder name, e.g., to automatically generate a new default folder on the first, second, third, etc. day of measuring.
Preview	Displays the default folder, the filename, and an example string for the placeholders you inserted, e.g.: D:\Project_A\Test_2017-10-05_01.pnrf

6.2.2 Storage options

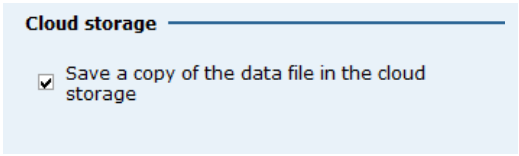
To display, click the **Acquisition** tab, and then click the **Data file** tab.



Option	Description
Prompt when recording has finished	At the end of recording, a dialog box is displayed. You must decide whether to save the data or not. If you want to save the data, you must specify a folder and a filename.
Save without confirmation	The data files are stored in the default folder for data files without further notice or confirmation (Data default folder and file name, 136).
Don't save	The data files are not saved.

6.2.3 Cloud storage

To display, click the **Acquisition** tab, and then click the **Data file** tab.

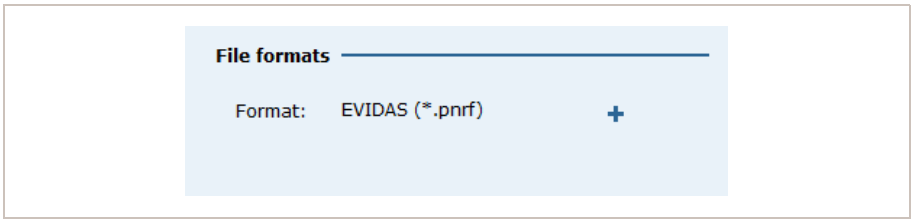


You can select this option if the HBM cloud is activated (Activating the HBM cloud, 23).

Option	Description
Save a copy of the data file in the cloud	At the end of each recording, a copy of the *.pnrf data file is uploaded to the HBM cloud (HBM cloud, 208).

6.2.4 File formats

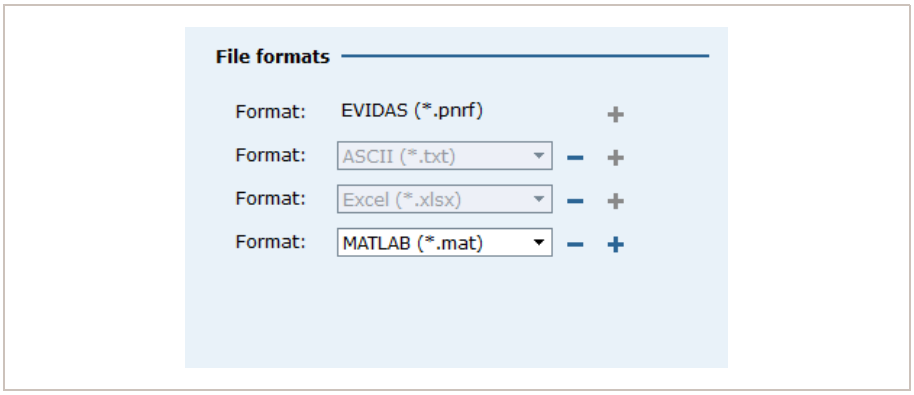
To display, click the **Acquisition** tab, and then click the **Data file** tab.



All measured values are saved in the EVIDAS data file format *.pnrf (**P**rimary **N**ative **R**ecording **F**ormat).

In addition, you can select other file formats. The measured values are then saved simultanously in the selected formats.

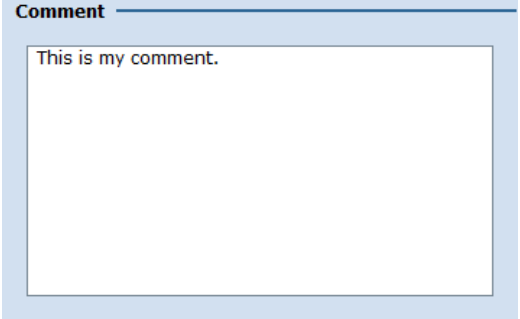
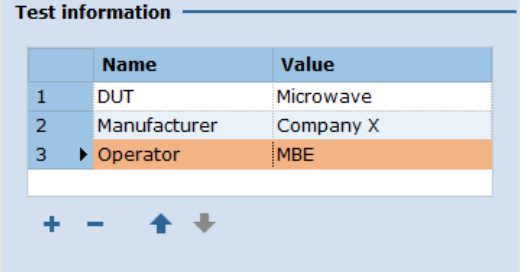
Click + to add a file format to the list, e.g., **ASCII (*.txt)**, **Excel (*.xlsx)**, or **MATLAB (*.mat)**.



Click − to remove a file format from the list.

6.2.5 Metadata

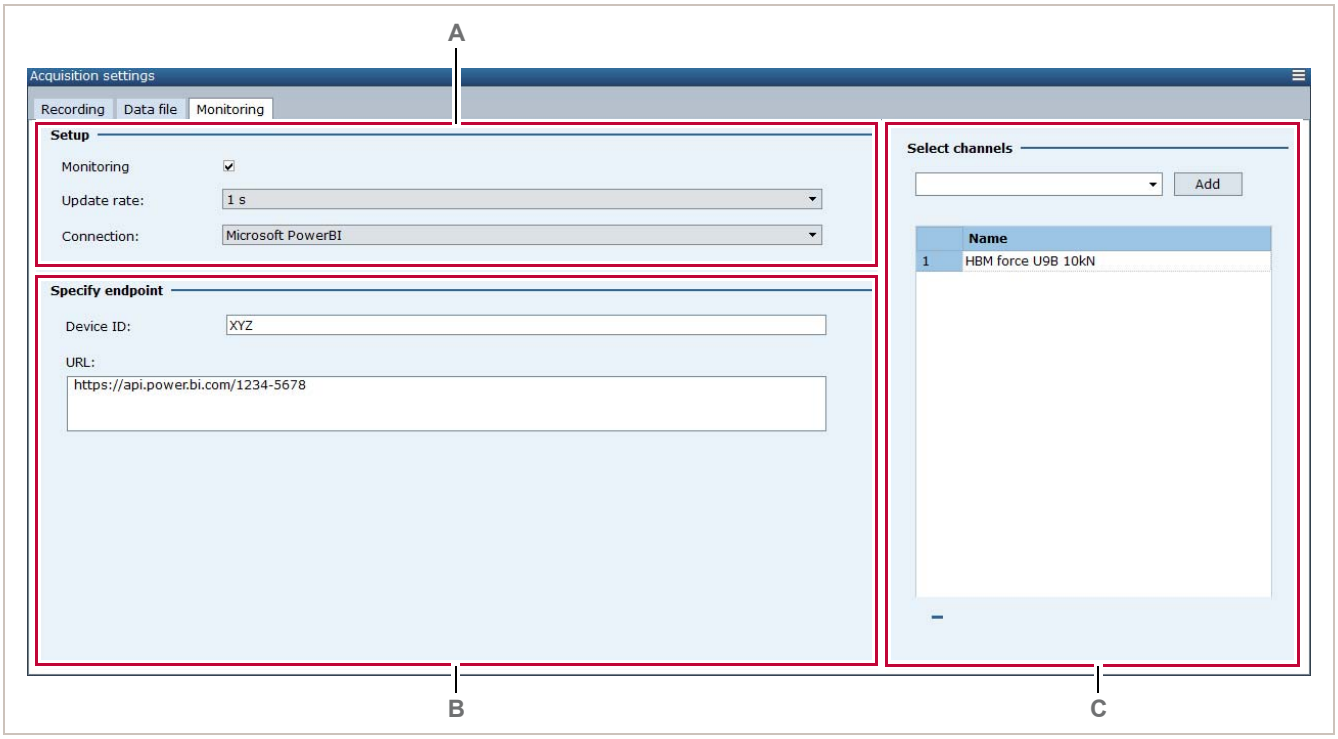
To display, click the **Acquisition** tab, and then click the **Data file** tab.

Data type	Description
Comment	To add information not specified under **Test information**. 
Test information	To define test information, e.g., the device under test (DUT), the manufacturer, and the operator.  For each item, you must define a name, a value, and a position in the list. Click + to add an item to the list. Click − to remove the selected item from the list. Click ↑ to move up a selected item in the list. Click ↓ to move down a selected item in the list.

The metadata is recorded in all subsequent data files and is displayed in the **Data manager** (Traceability data, 207).

6.3 Monitoring options

To display, click the **Acquisition** tab, and then click the **Monitoring** tab.



A Setup options, 142

B Endpoint specifications, 143

C Channel selection, 144

On the **Monitoring** tab, you can set up a data streaming connection between EVIDAS and an IoT platform in the cloud, e.g., to monitor a road bridge.

EVIDAS supports data streaming to different IoT platforms, e.g., to Microsoft Power BI.

Prerequisites

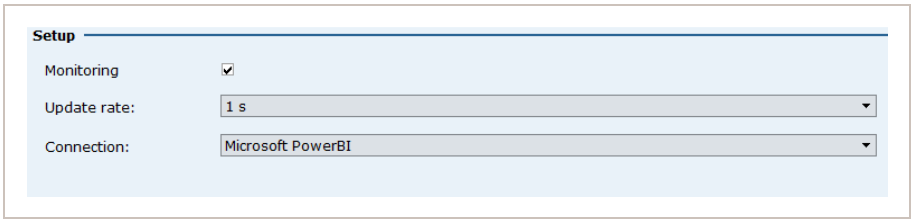
Before setting up EVIDAS for data streaming, you must do the following:

- Open an account on the IoT platform.
- Decide which project channels you want to stream.
- On your IoT account, create a streaming dataset and specify the exact names of the project channels you want to stream.
- Copy the Push URL provided by the IoT platform when creating the streaming dataset.
- On your IoT account, create a dashboard. The dashboard defines how the streaming dataset is displayed.

For detailed information how to set up a streaming dataset and a dashboard, please refer to the respective HBM Tech Notes on **www.hbm.com**.

6.3.1 Setup options

To display, click the **Acquisition** tab, and then click the **Monitoring** tab.

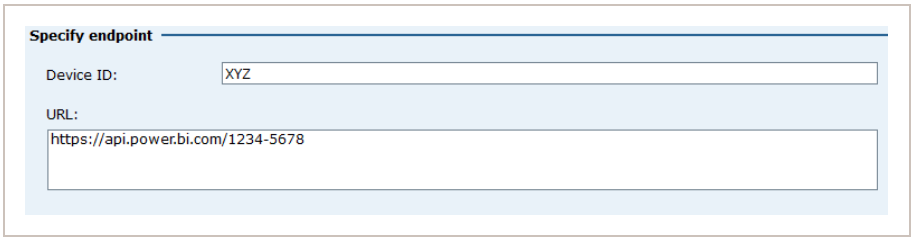


In **Setup** you define the update rate and the IoT platform you are connecting to.

Option	Description
Monitoring	Select this checkbox if you want to stream data to an IoT platform.
Update rate	The update rate is the interval at which measured values from selected project channels (Channel selection, 144) are streamed to the IoT platform. The update rate is normally less than 1 Hz in order not to exceed the data streaming volume.
Connection	From the drop-down list, select the IoT platform.

6.3.2 Endpoint specifications

To display, click the **Acquisition** tab, and then click the **Monitoring** tab.



An endpoint defines the web address where the data is streamed to.

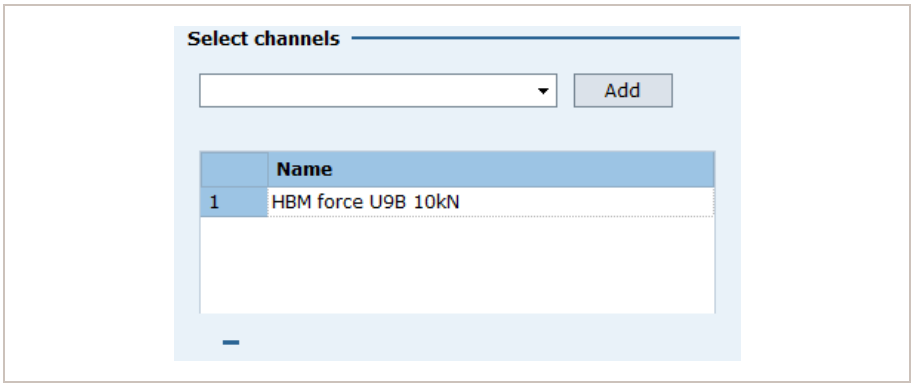
Option	Description
Device ID	Enter the device ID you specified when creating the streaming dataset on your IoT account. Specifying a device ID prevents measured values from different devices being mixed in the same streaming dataset.
URL	Paste the Push URL provided by the IoT platform when you created the streaming dataset.

In addition, some IoT platforms request that you specify the username and password of your IoT account, as well as an SSL certificate.

For detailed information how to specify the endpoint that connects EVIDAS to the IoT platform, please refer to the respective HBM Tech Notes on **www.hbm.com**.

6.3.3 Channel selection

To display, click the **Acquisition** tab, and then click the **Monitoring** tab.



In **Select channels** you define the project channels that are streamed to the IoT platform.

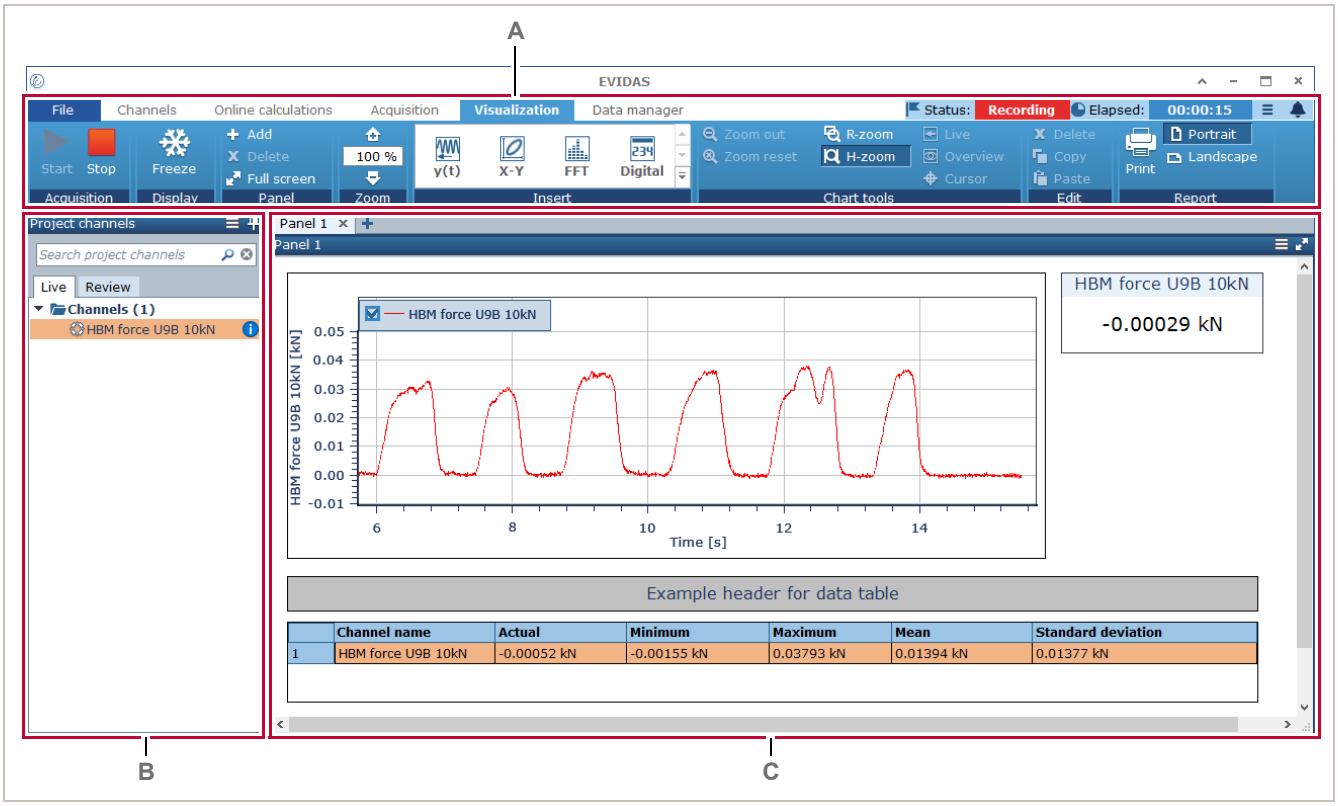
The channel names specified in the streaming dataset on the IoT platform must be identical with the project channel names in EVIDAS.

To add a channel to the list, select a project channel from the drop-down list, and then click **Add**.

To remove a channel from the list, select the channel, and then click −.

7 Visualization

On the **Visualization** tab and its panels, you define how the project channels are visualized.



A Visualization tab, 146↗

B Project channels panel in Visualization, 176↗

C Visualization panel, 177↗

Project channels can be live project channels, live calculation channels, or review channels.

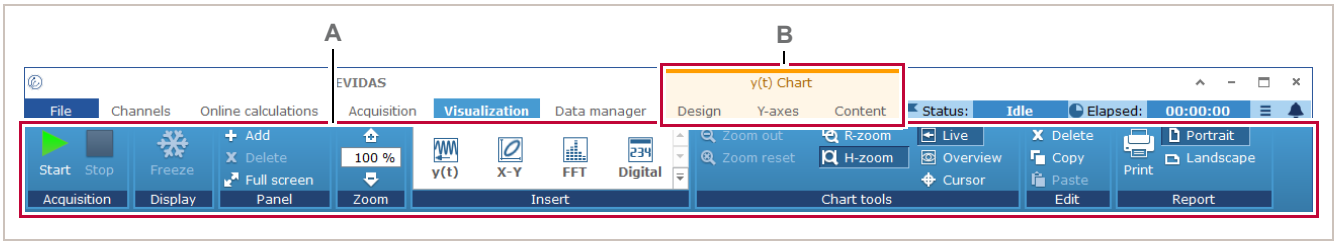
To visualize a channel, click a visualization object in the **Insert** group, e.g., **y(t)**, and then drag a channel from **Project channels** to the visualization object (Visualizing signals in a y(t) chart, 179↗).

To configure a visualization object, click the object, and then define its design and content on the contextual tab set displayed on the ribbon (Contextual tabs for visualization objects, 150↗).

If you do not define any visualization objects, EVIDAS will automatically visualize the first four active project channels in y(t) charts, digital meters, and a data table as soon as you click **Start**.

For examples on working with visualization objects, see: Working with visualization objects, 179↗.

7.1 Visualization tab



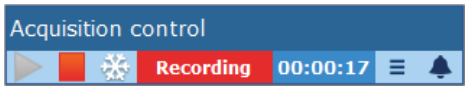
A Visualization commands, 146

B Contextual tabs for visualization objects, 150

On the **Visualization** tab, you select the visualization objects for your project channels, e.g., a y(t) chart. The visualization objects are displayed on the visualization panel (Visualization panel, 177).

7.2 Visualization commands

Visualization commands

Group	Command	Description
Panel	**Add**	To add a panel, e.g., to display visualization objects on another screen. To configure the new panel, see: Contextual tab for visualization panels, 175.
	Delete	To delete a panel. You can only delete a panel if at least one other panel remains. Click the panel you want to delete, and then click **Delete**.
	Full screen	To switch all visualization panels to full screen. The **Acquisition control** panel is displayed (Status panel, 41).  To exit full screen mode, click on the header of the visualization panel or press Esc.
Zoom	**Zoom**	To zoom in on all visualization objects on the active panel by 10% per click. The zoom factor is displayed.
	Zoom out	To zoom out on all visualization objects on the active panel by 10% per click. The zoom factor is displayed.

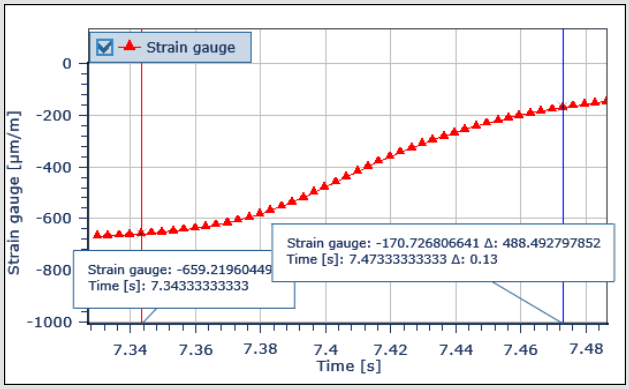
Visualization commands

Group	Command	Description
Insert	**y(t)**	To insert a y(t) chart on the visualization panel (Visualizing signals in a y(t) chart, 179↗). To assign a channel to a y(t) chart, drag the channel from **Project channels** to the y-axis of the y(t) chart. To configure the y(t) chart, see: Contextual tabs for y(t) charts, 151↗.
	X-Y	To insert an X-Y chart on the visualization panel (Visualizing signals in an X-Y chart, 182↗). To assign channels to an X-Y chart, drag them from **Project channels** to the y-axis and to the x-axis of the X-Y chart. To configure the X-Y chart, see: Contextual tabs for X-Y charts, 156↗
	FFT	To insert an FFT chart on the visualization panel (Visualizing the frequency spectrum of a signal in an FFT chart, 186↗). To assign a channel to an FFT chart, drag the channel from **Project channels** to the y-axis of the FFT chart. To configure the FFT chart, see: Contextual tabs for FFT charts, 161↗
	Digital	To insert a digital meter on the visualization panel. To assign a channel to a digital meter, drag the channel from **Project channels** to the digital meter. To configure the digital meter, see: Contextual tab for digital meters, 169↗.
	Table	To insert a data table on the visualization panel. To assign a channel to a data table, drag the channel from **Project channels** to the table. To configure the data table, see: Contextual tab for data tables, 171↗.
	Text	To insert a text box on the visualization panel. To configure the text box, see: Contextual tab for text boxes, 173↗

Visualization commands

Group	Command	Description
Chart tools	**Zoom out**	To zoom out one zoom step at a time in the selected y(t) chart.
	Zoom reset	To zoom out all prior zoom steps at once in the selected y(t) chart.
	R-zoom	To zoom in on any rectangular section of the selected y(t) chart. To zoom, click **R-zoom**, click in the trace area, and hold down the pointer. When the pointer turns into a +, drag the pointer to select a rectangular section, and then release the pointer (Zooming a trace, 193).
	H-zoom	To zoom in horizontally, i.e., on a time window of the selected y(t) chart. To zoom, click **H-zoom**, click in the trace area, and hold down the pointer. When the pointer turns into a +, drag the pointer to select a time window, and then release the pointer (Zooming a trace, 193).

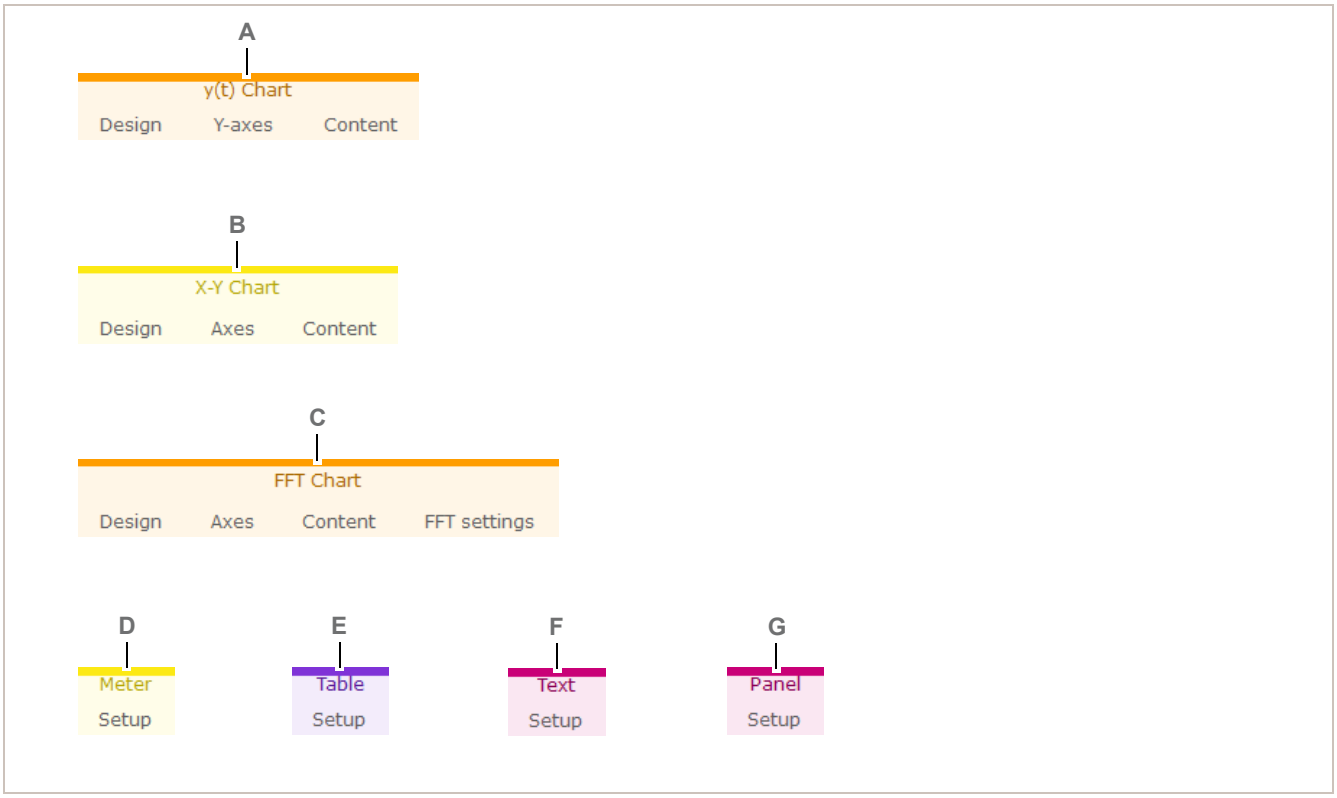
Visualization commands

Group	Command	Description
	Live	To display all traces in the selected y(t) chart in real time within a defined time window (Design tab for y(t) charts, 152), e.g., within the last 10 seconds. The trace is moved from right to left through the time window. You can switch from **Live** mode to **Overview** mode and back during the measurement.
	Overview	To display an overview of all traces in the selected y(t) chart since the start of the measurement. The y(t) chart keeps its size and the trace is compressed each time after a defined time increment (Design tab for y(t) charts, 152). You can switch from **Overview** mode to **Live** mode and back during the measurement.
	Cursor	To display information on measured values in a y(t) chart.  You can drag the cursors to the measured values you want to investigate. For the second measured value, the difference Δ to the first value is displayed.
Edit	**Delete**	To delete the selected visualization object.
	Copy	To copy the selected visualization object to the clipboard.
	Paste	To paste the content of the clipboard to the selected visualization panel.
Report	**Print**	To print the content of the selected visualization panel to the default printer.
	Portrait	To select portrait printing.
	Landscape	To select landscape printing.

⇨ Start / Stop button, 40
Freeze button, 40

7.3 Contextual tabs for visualization objects

On the contextual tabs (Contextual tabs, 43) you define the design and content of the visualization objects.



A Contextual tabs for y(t) charts, 151

B Contextual tabs for X-Y charts, 156

C Contextual tabs for FFT charts, 161

D Contextual tab for digital meters, 169

E Contextual tab for data tables, 171

F Contextual tab for text boxes, 173

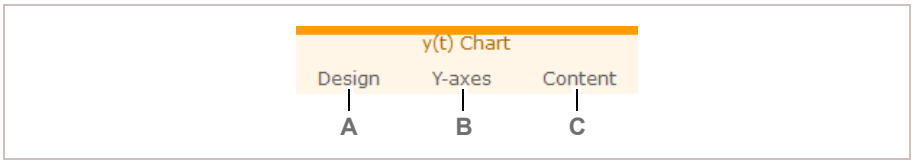
G Contextual tab for visualization panels, 175

The contextual tabs are displayed if a visualization object is selected, e.g., a y(t) chart.

The contextual tabs disappear if you click away from the visualization panel.

7.3.1 Contextual tabs for y(t) charts

To display, click the **Visualization** tab, and then click **y(t)**, or click a y(t) chart on the visualization panel.

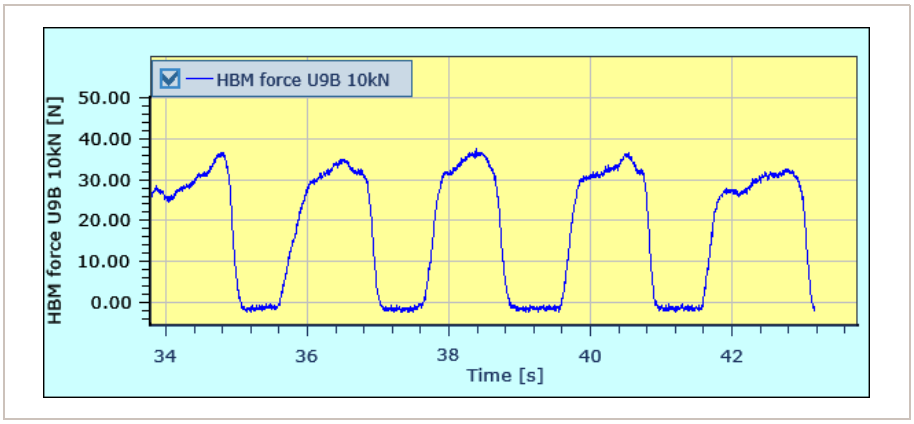


A Design tab for y(t) charts, 152

B Y-axes tab for y(t) charts, 154

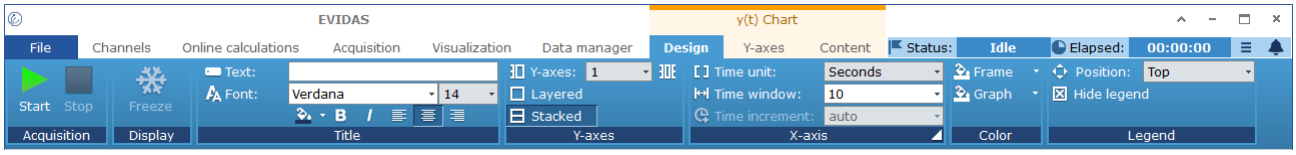
C Content tab for y(t) charts, 155

A y(t) chart visualizes measured values over time.



To assign a channel to a y(t) chart, drag a live or review channel from the **Project channels** panel to the y(t) chart (Visualizing signals in a y(t) chart, 179).

Design tab for y(t) charts



On the **Design** tab for y(t) charts, you define the title, the number and arrangement of y-axes, the time settings, the background colors, and the position of the legend.

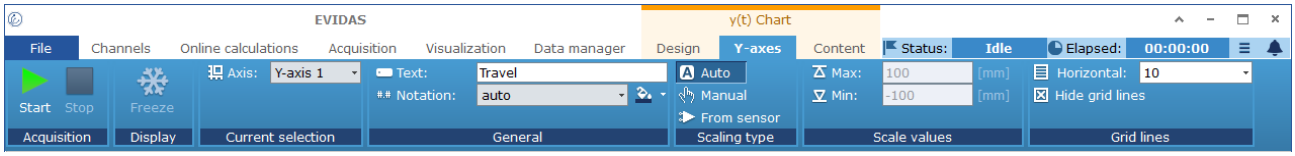
Commands on the Design tab for y(t) charts

Group	Command	Description
Title	**Text**	To enter the title of the y(t) chart.
	Font	To select the font and font size of the title.
		To define the title color, select a color from the color palette. 
		Bold.
		Italic.
		Align left.
		Center.
		Align right.
Y-axes	**Y-axes**	To define the number of coordinate systems in the y(t) chart (Visualizing signals in a y(t) chart, 179).
	Layered	To display the coordinate systems layered, i.e., one on top of the other. The scales of the y-axes are all displayed to the left. The traces are displayed on top of each other.
	Stacked	To display the coordinate systems stacked. The traces are displayed separately.
	Dual	For layered systems, to display the scales of the y-axes to the left and to the right.

Commands on the Design tab for y(t) charts

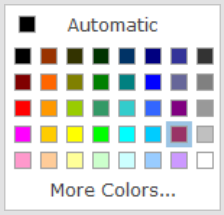
Group	Command	Description
X-Axis	**Time unit**	To define the time unit on the x-axis, e.g., seconds [s].
	Time window	To define the time window displayed on the y(t) chart. This option is available in **Live** mode (Visualization commands, 146).
	Time increment	To define the time after which the display of the x-axis and of the trace is updated to fit the expanding time window in **Overview** mode (Visualization commands, 146).
Color	**Frame**	To define the background color outside the coordinate system.
	Graph	To define the background color of the coordinate system.
Legend	**Position**	To define the position of the legend. The legend is displayed outside the trace area.
	Hide legend	To hide or display the legend.

Y-axes tab for y(t) charts

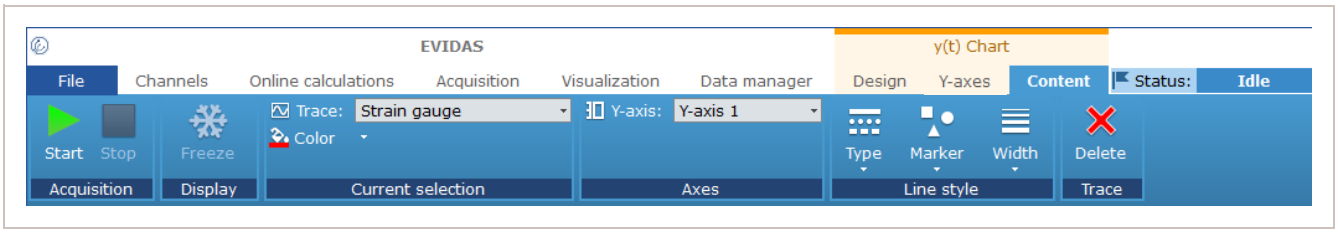


On the **Y-axes** tab for y(t) charts, you define the scaling of the y-axes.

Commands on the Y-axes tab for y(t) charts

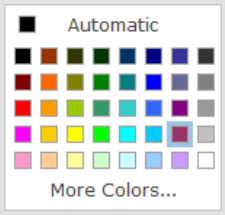
Group	Command	Description
Current selection	**Axis**	To select a y-axis. Alternatively, click a legend in the y(t) chart.
General	**Text**	To edit the name of the y-axis.
	Notation	To define the notation of the scale values on the y-axis.
		To define the scale color of the y-axis, select a color from the color palette. 
Scaling type	**Auto**	To use autoscaling. The scaling on the y-axis is adjusted automatically to the signal range.
	Manual	To manually define the scaling range on the y-axis. You must define a minimum and maximum value in the **Scale values** group.
	From sensor	To apply the scaling range defined in the sensor settings (Sensor settings, 79).
Scale values	**Max**	To define the maximum scale value on the y-axis.
	Min	To define the minimum scale value on the y-axis.
Grid lines	**Horizontal**	To define the number of horizontal grid lines in the trace area.
	Hide grid lines	To hide or display the horizontal grid lines.

Content tab for y(t) charts



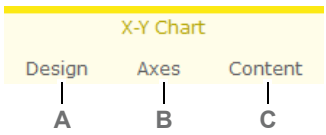
On the **Content** tab for y(t) charts, you assign traces to coordinate systems and define how the traces are displayed.

Commands on the Content tab for y(t) charts

Group	Command	Description
Current selection	**Trace**	To select a trace for editing. Alternatively, click a legend in the y(t) chart.
	Color	To define the trace color, select a color from the color palette. 
Axes	**Y-axis**	To select the coordinate system in which the trace is displayed.
Line style	**Type**	To define the line style of the trace, e.g., a dashed line (Changing the line style of a trace, 196↗).
	Marker	To define the markers used for the measured values (Changing the line style of a trace, 196↗).
	Width	To define the line width of the trace (Changing the line style of a trace, 196↗).
Trace	**Delete**	To delete a trace from the y(t) chart (Deleting a trace, 192↗). Alternatively, right-click the legend of the trace, and then click **Delete trace from chart**.

7.3.2 Contextual tabs for X-Y charts

To display, click the **Visualization** tab, and then click **X-Y**, or click an X-Y chart on the visualization panel.



A Design tab for X-Y charts, 157

C Content tab for X-Y charts, 160

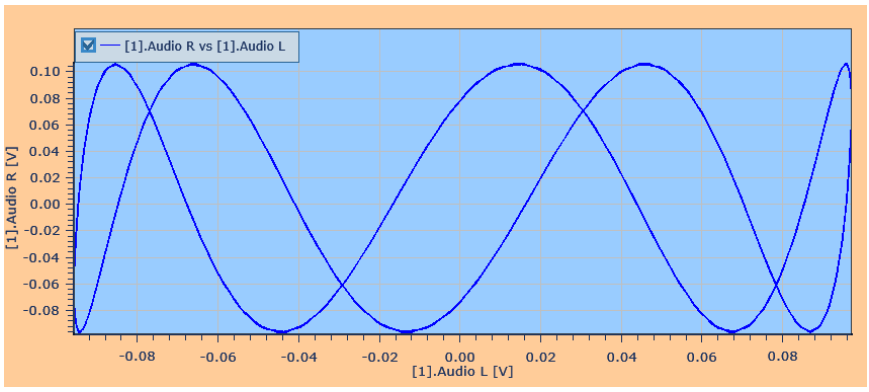
B Axes tab for X-Y charts, 159

X-Y charts are used to visualize the relationship between two signals. One signal is assigned to the x-axis, the other to the y-axis. The signal on the x-axis is called the X-channel, the signal on the y-axis is the Y-channel.

Example 1: Force-displacement diagram to determine mechanical work.

Example 2: Pressure-volume diagram to examine the efficiency of an engine cycle.

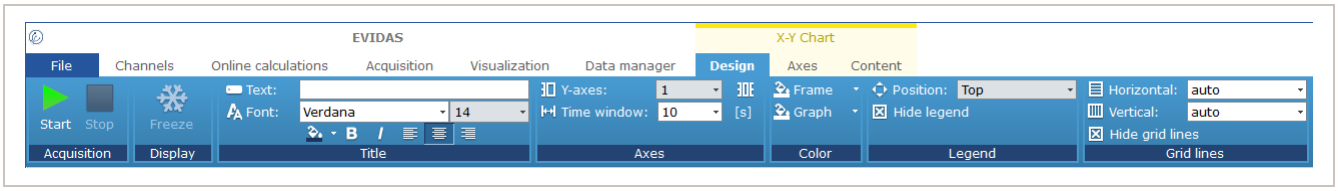
Example 3: Lissajous curves. These traces are generated by a sinewave each on the X-channel and on the Y-channel. The frequency ratio (a fraction with an integer numerator and denominator) and phase shift of the sinewaves determine the form of the Lissajous curve.



A Lissajous curve on an X-Y chart

To assign channels to an X-Y chart, drag the live or review channels from the **Project channels** panel to the respective axes on the X-Y chart (Visualizing signals in an X-Y chart, 182).

Design tab for X-Y charts



On the **Design** tab for X-Y charts, you define the title, the number and arrangement of y-axes, the background colors, the position of the legend, and the number of horizontal and vertical grid lines.

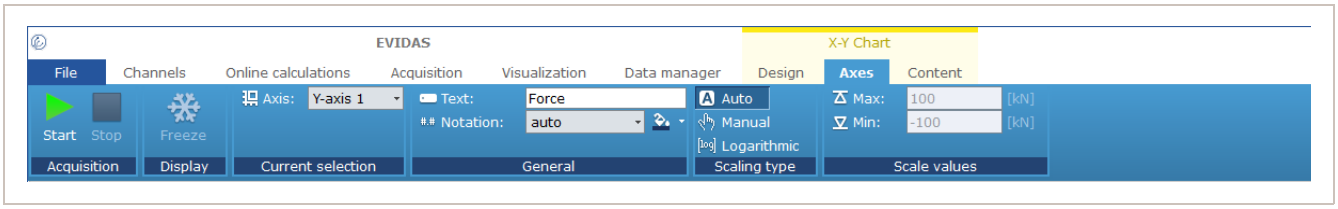
Commands on the Design tab for X-Y charts

Group	Command	Description
Title	**Text**	To enter the title of the X-Y chart.
	Font	To select the font and font size of the title.
		To define the title color, select a color from the color palette. 
		Bold.
		Italic.
		Align left.
		Center.
		Align right.
Axes	**Y-axes**	To define the number of y-axes on the X-Y chart. You can assign different project channels to the y-axes, but only one project channel to the x-axis. The coordinate systems are layered, i.e., they are displayed one on top of the other. Click **Dual** to display the scales of the y-axes on both sides of the layered coordinate systems.
	Time window	To define a time window for the X-Y chart. Measured values that are older than the time window, e.g., older than 10 seconds, are not displayed when the display of the chart is refreshed.
	Dual	To display the scales of the y-axes to the left and to the right.
Color	**Frame**	To define the background color outside the coordinate system.
	Graph	To define the background color of the coordinate system.

Commands on the Design tab for X-Y charts

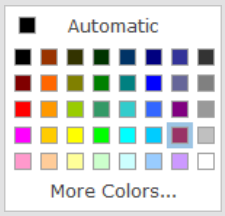
Group	Command	Description
Legend	**Position**	To define the position of the legend. The legend is displayed outside the trace area.
	Hide legend	To hide or display the legend.
Grid lines	**Horizontal**	To define the number of horizontal grid lines in the trace area.
	Vertical	To define the number of vertical grid lines in the trace area.
	Hide grid lines	To hide or display the grid lines.

Axes tab for X-Y charts

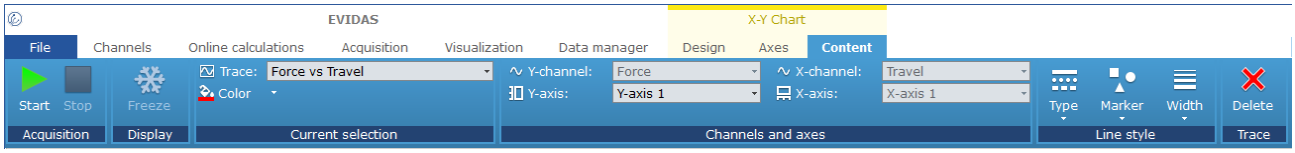


On the **Axes** tab for X-Y charts, you define the scaling of the axes.

Commands on the Axes tab for X-Y charts

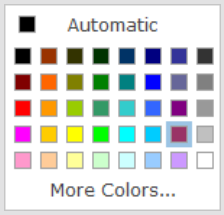
Group	Command	Description
Current selection	**Axis**	To select an axis. This can be a y-axis or the x-axis.
General	**Text**	To enter the name of the axis.
	Notation	To define the notation of the scale values on the axis.
		To define the scale color, select a color from the color palette. 
Scaling type	**Auto**	To use autoscaling. The scaling on the axis is adjusted automatically to the signal range.
	Manual	To manually define the scaling range. You must define a minimum and a maximum value in the **Scale values** group.
	Logarithmic	To use logarithmic scaling on the selected axis.
Scale values	**Max**	To define the maximum scale value on the axis.
	Min	To define the minimum scale value on the axis.

Content tab for X-Y charts



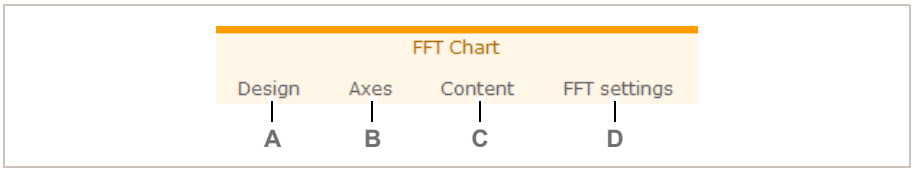
On the **Content** tab for X-Y charts, you assign traces to coordinate systems and define how the traces are displayed.

Commands on the Content tab for X-Y charts

Group	Command	Description
Current selection	**Trace**	To select a trace. Alternatively, click a legend on the X-Y chart.
	Color	To define the trace color, select a color from the color palette. 
Channels and axes	**Y-channel**	The Y-channel of the selected trace is displayed.
	Y-axis	The y-axis of the Y-channel is displayed. To assign the selected trace to another coordinate system, select a different y-axis.
	X-channel	The X-channel of the selected trace is displayed.
	X-axis	All traces use the same x-axis.
Line style	**Type**	To define the line style of the trace, e.g., a dashed line (Changing the line style of a trace, 196↗).
	Marker	To define the markers used for the measured values (Changing the line style of a trace, 196↗).
	Width	To define the line width of the trace (Changing the line style of a trace, 196↗).
Trace	**Delete**	To delete a trace from the X-Y chart (Deleting a trace, 192↗). Alternatively, right-click the legend of the trace, and then click **Delete trace from chart**.

7.3.3 Contextual tabs for FFT charts

To display, click the **Visualization** tab, and then click **FFT**, or click an FFT chart on the visualization panel.



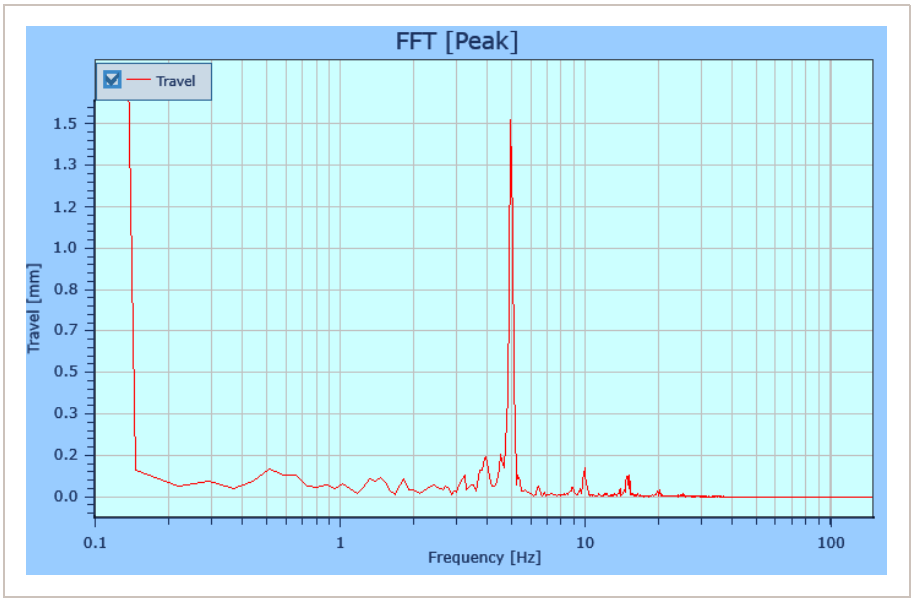
A Design tab for FFT charts, 162↗

B Axes tab for FFT charts, 164↗

C Content tab for FFT charts, 166↗

D FFT settings tab, 167↗

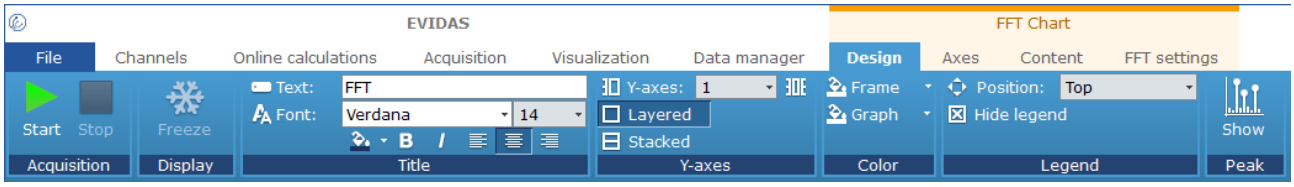
An FFT chart visualizes the frequency components of a signal. The amplitudes of the frequency components represent their respective contribution to the signal (Appendix D: FFT Introduction, 257↗).



☐ An FFT representation of a signal with a main frequency component of 5 Hz

To assign a channel to an FFT chart, drag a live or review channel from the **Project channels** panel to the y-axis of the FFT chart (Visualizing the frequency spectrum of a signal in an FFT chart, 186↗).

Design tab for FFT charts



On the **Design** tab for FFT charts, you define the title, the number and arrangement of y-axes, the background colors, the position of the legend, and whether the peak markers are displayed.

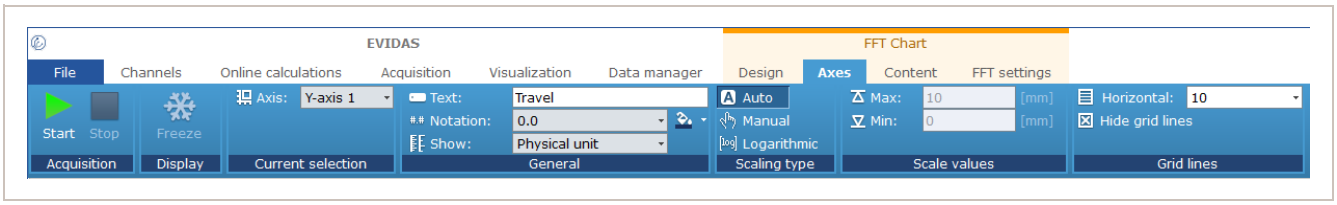
Commands on the Design tab for FFT charts

Group	Command	Description
Title	**Text**	To edit the title of the FFT chart.
	Font	To select the font and font size of the title.
		To define the title color, select a color from the color palette. 
		Bold.
		Italic.
		Align left.
		Center.
		Align right.
Y-axes	**Y-axes**	To define the number of coordinate systems on the FFT chart.
	Layered	To display the coordinate systems layered, i.e., one on top of the other. The scales of the y-axes are all displayed to the left. The traces are displayed on top of each other.
	Stacked	To display the coordinate systems stacked. The traces are displayed separately.
	Dual	For layered systems, to display the scales of the y-axes to the left and to the right.
Color	**Frame**	To define the background color outside the coordinate system.
	Graph	To define the background color of the coordinate system.

Commands on the Design tab for FFT charts

Group	Command	Description
Legend	**Position**	To define the position of the legend. The legend is displayed outside the trace area.
	Hide legend	To hide or display the legend.
Peak	**Show**	To display or hide the peak markers.

Axes tab for FFT charts



On the **Axes** tab for FFT charts, you define the scaling of the axes.

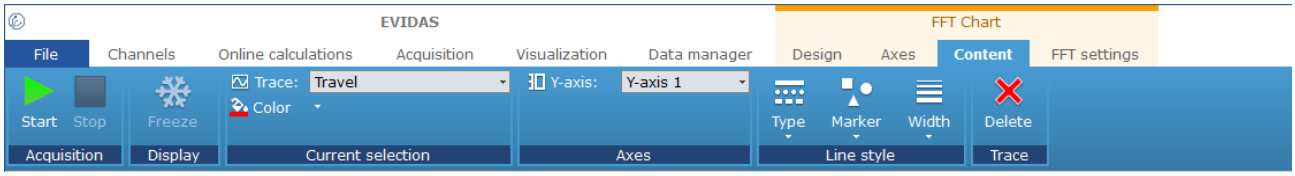
Commands on the Axes tab for FFT charts

Group	Command	Description
Current selection	**Axis**	To select an axis. This can be a y-axis or the frequency axis.
General	**Text**	To edit the name of a y-axis (The name of the x-axis is always **Frequency**).
	Notation	To define the notation of the scale values on the axis.
	Show	To define the unit of a y-axis either as the physical unit of the project channel or as decibel [dB]. For the x-axis the unit is Hz.
		To define the scale color of the axis, select a color from the color palette. 
Scaling type	**Auto**	To use autoscaling. The scaling on the axis is adjusted automatically to the signal range.
	Manual	To manually define the scaling range. You must define a minimum and maximum value in the **Scale values** group.
	Logarithmic	To use logarithmic scaling on the selected axis. Use logarithmic scaling to fit a vast spectrum of frequencies or amplitudes into the chart (Visualizing the frequency spectrum of a signal in an FFT chart, 186).
Scale values	**Max**	To define the maximum scale value on the y-axis.
	Min	To define the minimum scale value on the y-axis.
	Max	To define the maximum scale value on the x-axis.
	Min	To define the minimum scale value on the x-axis.

Commands on the Axes tab for FFT charts

Group	Command	Description
Grid lines	**Horizontal**	This icon is displayed if you select a y-axis. Use this icon to define the number of horizontal grid lines in the trace area. If you use logarithmic scaling on the y-axis, the number of horizontal grid lines is defined by the program.
	Vertical	This icon is displayed if you select the x-axis. Use this icon to define the number of vertical grid lines in the trace area. If you use logarithmic scaling on the x-axis, the number of vertical grid lines is defined by the program.
	Hide grid lines	To hide or display the grid lines.

Content tab for FFT charts

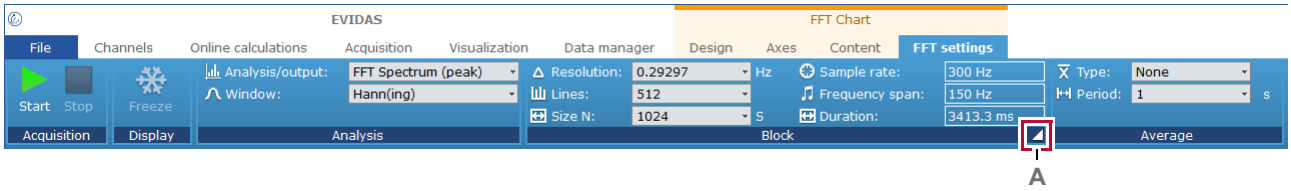


On the **Content** tab for FFT charts, you assign traces to coordinate systems and define how the traces are displayed.

Commands on the Content tab for FFT charts

Group	Command	Description
Current selection	**Trace**	To select a trace. Alternatively, click a legend on the FFT chart.
	Color	To define the trace color, select a color from the color palette. 
Axes	**Y-axis**	To select the coordinate system in which the trace is displayed.
Line style	**Type**	To define the line style of the trace, e.g., a dashed line (Changing the line style of a trace, 196↗).
	Marker	To define the markers used for the measured values (Changing the line style of a trace, 196↗).
	Width	To define the line width of the trace (Changing the line style of a trace, 196↗).
Trace	**Delete**	To delete a trace from the FFT chart (Deleting a trace, 192↗). Alternatively, right-click the legend of the trace, and then click **Delete trace from chart**.

FFT settings tab



A Dialog Box Launcher to display list of block settings (List of block settings, 168)

On the **FFT settings** tab, you define the following settings:

- Frequency analysis option.
- Window function.
- Block settings.
- Averaging algorithm.

Commands on the FFT settings tab

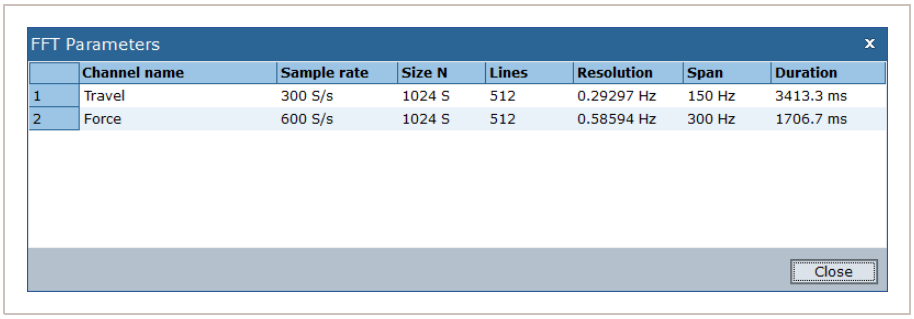
Group	Command	Description
Analysis	**Analysis/output**	To select a frequency analysis option (Frequency analysis options, 268).
	Window	To select a window function for minimizing spectral leakage (Windowing, 264).
Block	**Resolution**	To select the frequency resolution Δf (Frequency resolution Δf, 260). If the sample rate f_s and the frequency resolution Δf are given, the number of lines L and the block size N are determined (Dependencies between block settings, 262).
	Lines	To select the number of lines L (Number of lines L, 261). If the sample rate f_s and the number of lines L are given, the frequency resolution Δf and the block size N are determined (Dependencies between block settings, 262).
	Size N	To select the block size N (Block size N, 259). The larger the block size N, the longer the FFT calculation will take and the more accurate the frequency peaks will be displayed on the FFT chart. If the sample rate f_s and the block size N are given, the frequency resolution Δf and the number of lines L are determined (Dependencies between block settings, 262).
	Sample rate	The sample rate f_s of the project channel is displayed (Sample rate f_s, 259). You cannot change the sample rate in the frequency domain. If there is more than one project channel assigned to the FFT chart and those project channels use different sample rates, the sample rate of the first project channel assigned to the FFT chart is taken for the FFT calculation.

Commands on the FFT settings tab

Group	Command	Description
	Frequency span	The frequency span *F* is displayed (Frequency span *F*, 261↗). The frequency span *F* is determined by the sample rate f_s (Dependencies between block settings, 262↗).
	Duration	The block duration *T* is displayed (Block duration *T*, 260↗). With the sample rate f_s fixed, the block duration *T* is determined by the block size *N* (Dependencies between block settings, 262↗).
Average	**Type**	To select an averaging algorithm (Averaging, 267↗).
	Period	To select the time period in seconds for which the FFT calculations are averaged.

List of block settings

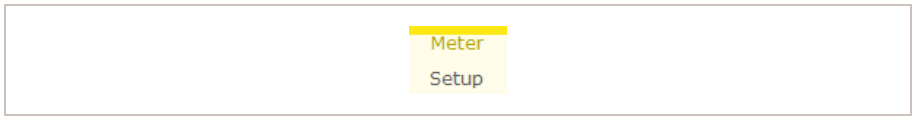
To display, on the **FFT settings** tab, click the Dialog Box Launcher of the **Block** group.



The block settings of the project channels that are assigned to the FFT chart are displayed (Block settings, 259↗).

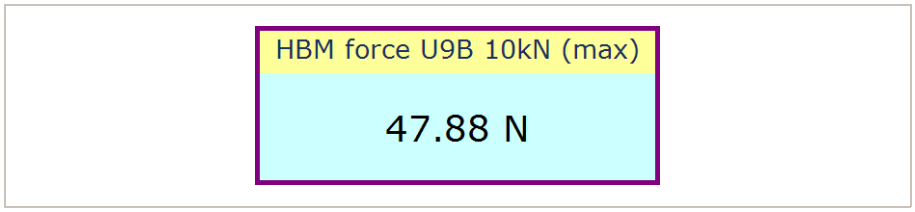
7.3.4 Contextual tab for digital meters

To display, click the **Visualization** tab, and then click **Digital** or a digital meter on the visualization panel.



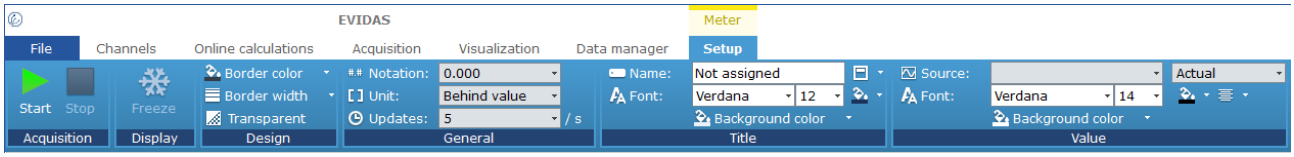
To assign a channel to a digital meter, drag a live or review channel from the **Project channels** panel to the digital meter.

A digital meter shows the digital representation of the measured values for a single project channel.



By default, the current value is displayed. Alternatively, you can display the minimum value, maximum value, mean value, or standard deviation since the start of the measurement.

Meter Setup tab



On the **Meter Setup** tab, you define the design of the digital meter, the notation of the measured values, the update rate, and the type of value displayed.

Commands on the Meter Setup tab

Group	Command	Description
Design	**Border color**	To define a color for the border of the digital meter.
	Border width	To define a line width for the border of the digital meter.
	Transparent	To make the digital meter transparent, i.e, only the title of the digital meter and the measured values are displayed. The border and background color definitions are inactive.
General	**Notation**	To define the notation of the measured values.
	Unit	To define the position of the physical unit.
	Updates	To define the update rate per second.
Title	**Name**	To enter a title for the digital meter.
	Font	To define the font and font size of the title.
	Background color	To define a background color for the title.
		To define the position of the title.
		To define the title color.
Value	**Source**	To assign a project channel to the digital meter and to select the type of value displayed. By default, the current value is displayed. Alternatively, you can display the minimum value, maximum value, mean value, or standard deviation since the start of the measurement.
	Font	To define a font and font size for the measured values.
	Background color	To define a background color for the measured values.
		To define a color for the measured values.
		To align the measured values on the meter.

⇨ Start / Stop button, 40⇗
Freeze button, 40⇗

7.3.5 Contextual tab for data tables

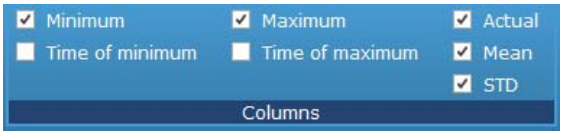
To display, click the **Visualization** tab, and then click **Table** or a table on the visualization panel.

To assign a channel to a data table, drag a live or review channel from the **Project channels** panel to the data table.

A data table displays the measured values and basic statistical parameters for one or more project channels.

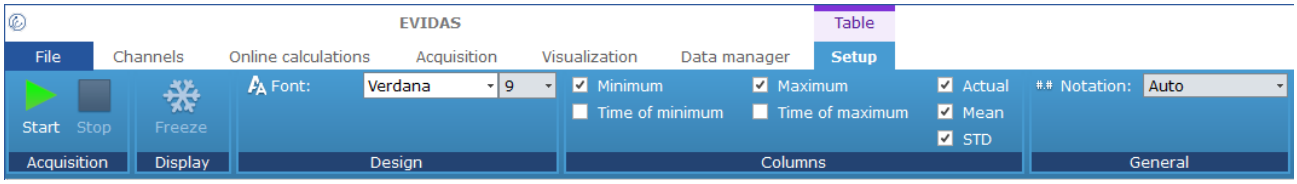
	Channel name	Actual	Minimum	Time of minimum	Maximum	Time of maximum	Mean	Standard deviation
1	HBM force U9B 10kN	-0.00033 kN	-0.00134 kN	15.74 s	0.05335 kN	12.21 s	0.01183 kN	0.01788 kN
2	Digital Frequency F1	0.00000 Hz	0.00000 Hz	0.00000 s	0.00000 Hz	0.00000 s	0.00000 Hz	0.00000 Hz

You can select the statistical parameters displayed in the data table.



The parameters are calculated over all measured values since the start of the measurement.

Table Setup tab



On the **Table Setup** tab, you define the design of the table header, the content of the columns, and the notation of the measured values.

Commands on the Table Setup tab

Group	Command	Description
Design	**Font**	To define a font and font size for the table header.
Columns	**Minimum**	To display the minimum value since the start of the measurement.
	Time of minimum	To display the time at which the minimum value occured.
	Maximum	To display the maximum value since the start of the measurement.
	Time of maximum	To display the time at which the maximum value occured.
	Actual	To display the current value.
	Mean	To display the mean value of all measured values since the start of the measurement.
	STD	To display the standard deviation of all measured values since the start of the measurement.
General	**Notation**	To define the notation of the measured values. Click the drop-down menu button and select a notation. Auto 0 0.0 0.00 0.000 0.0000 0.00000 0.000000 Scientific

⇨ Start / Stop button, 40
Freeze button, 40

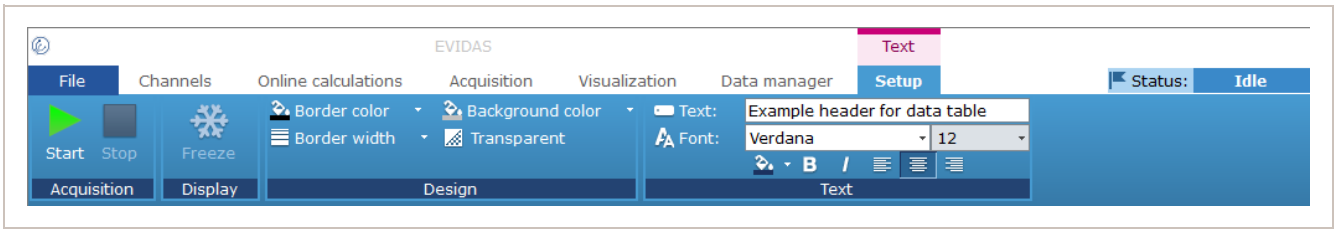
7.3.6 Contextual tab for text boxes

To display, click the **Visualization** tab, and then click **Text** or a text box on the visualization panel.

You can insert a text box, e.g., to create a header for a data table.

You cannot assign a channel to a text box.

Text Setup tab



On the **Text Setup** tab, you define the design of the text box, its content and how the text is formated.

You can move the text box on the visualization panel and adjust its size.

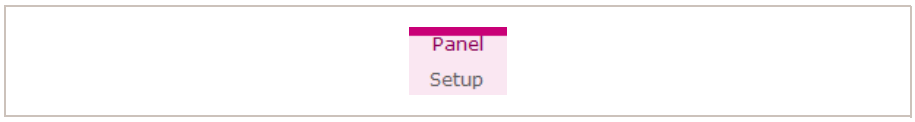
Commands on the Text Setup tab

Group	Command	Description
Design	**Border color**	To define a color for the border of the text box.
	Border width	To define a line width for the border of the text box.
	Background color	To define a background color for the text box.
	Transparent	To make the text box transparent, i.e, only the text and the border of the text box are displayed. The background color is inactive.
Text	**Text**	To enter text into the text box.
	Font	To select a font and font size for the text.
		To define the color of the text.
		Bold.
		Italic.
		Align left.
		Center.
		Align right.

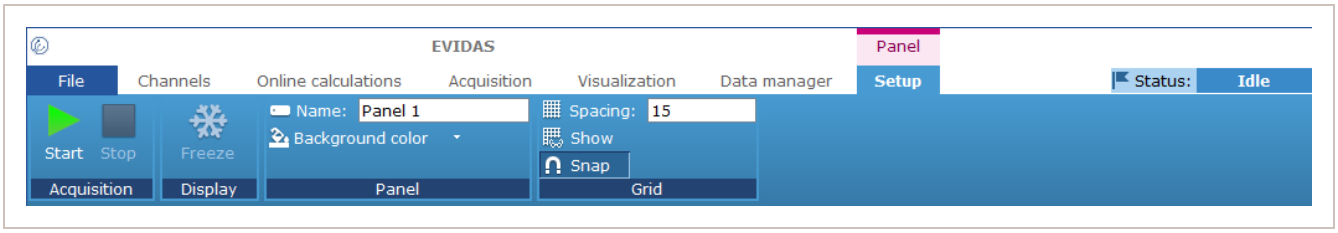
⇨ Start / Stop button, 40
Freeze button, 40

7.3.7 Contextual tab for visualization panels

To display, click the **Visualization** tab, and then click the visualization panel, but not any visualization object.

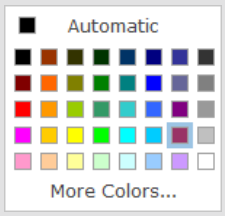
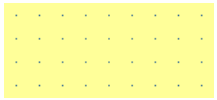


Panel Setup tab



On the **Panel Setup** tab, you can edit the panel name, change the background color, and define a snap grid.

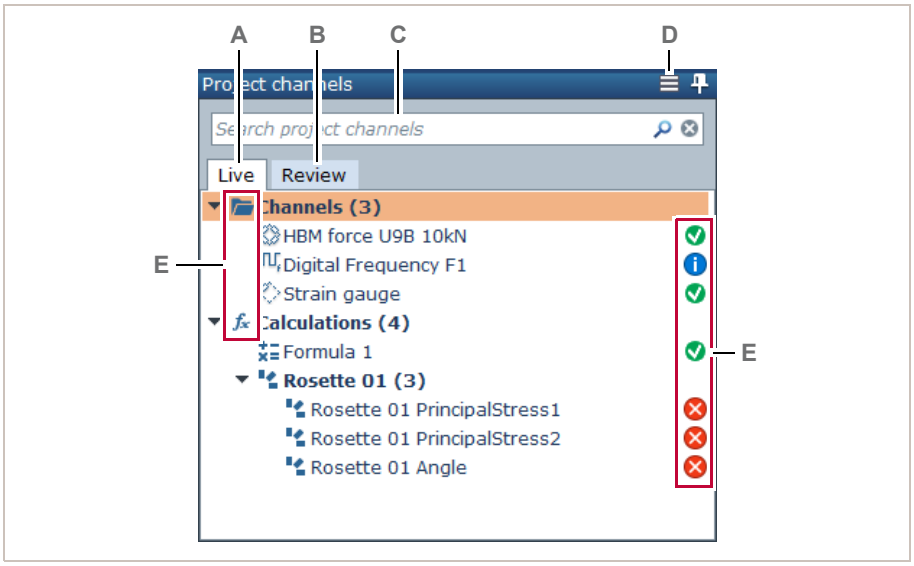
Commands on the Panel Setup tab

Group	Command	Description
Panel	**Name**	To edit the panel name.
	Background color	To define a background color for the panel, select a color from the color palette. 
Grid	**Spacing**	To define the spacing of the snap grid.  The snap grid supports the alignment of visualization objects on the panel.
	Show	To show / hide the snap grid.
	Snap	To activate / deactivate the snap grid.

⇨ Start / Stop button, 40
Freeze button, 40
Visualization panel, 177

7.4 Project channels panel in Visualization

To display, click the **Visualization** tab.



A Live channels, 176
B Review channels, 176
C Search box, 47
D Burger menu, 176
E Folder icons, 113
F Signal status icons, 109

The **Project channels** panel displays the live and review channels available for visualization.

Live channels

A live channel delivers measured values from a sensor or calculated values.

Review channels

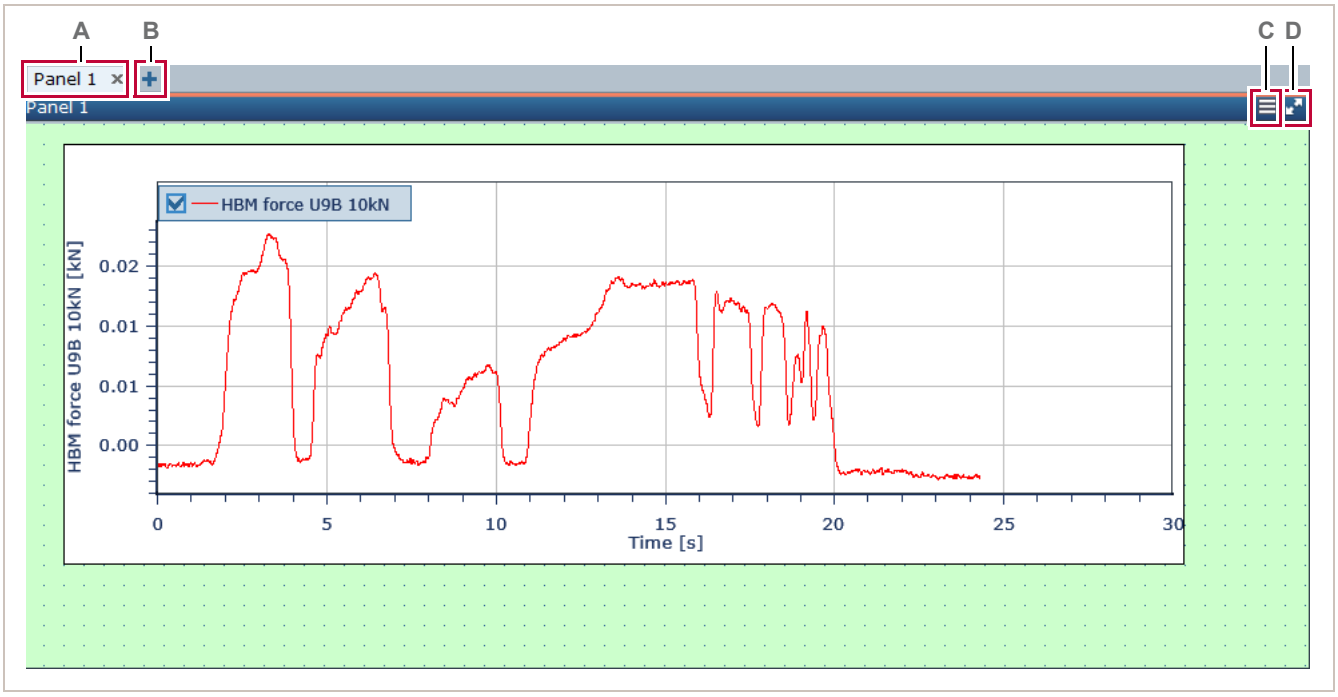
A review channel delivers measured values from a file.

Burger menu

Command	Description
Help on project channels	To access context-sensitive help.

7.5 Visualization panel

To display, click the **Visualization** tab.



A Panel tab, 177

B Add panel button, 177

C Burger menu, 177

D Full screen button, 178

The visualization panel displays the visualization objects you defined, e.g., a y(t) chart.

Panel tab

You can define more than one visualization panel, e.g., to display visualization objects on a second screen.

To display a panel, click the respective **Panel** tab.

To move a panel to another screen, click the **Panel** tab, hold down the pointer, and then drag it to the other screen.

Add panel button

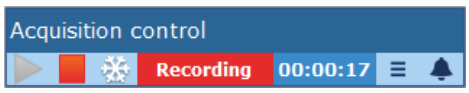
To add a visualization panel, click .

Burger menu

Command	Description
Help on visualization	To access context-sensitive help.

Full screen button

To maximize the visualization panel, click . The status panel turns into a floating panel (Status panel, 41).



To exit full screen mode, click or press Esc.

7.6 Working with visualization objects

The following procedures demonstrate how to work with visualization objects.

Prerequisites

- Data acquisition device and sensors are connected.
- Project channels are selected (To select project channels, 59).
- Project channels are configured (Configuring project channels, 60).

7.6.1 Visualizing signals in a y(t) chart

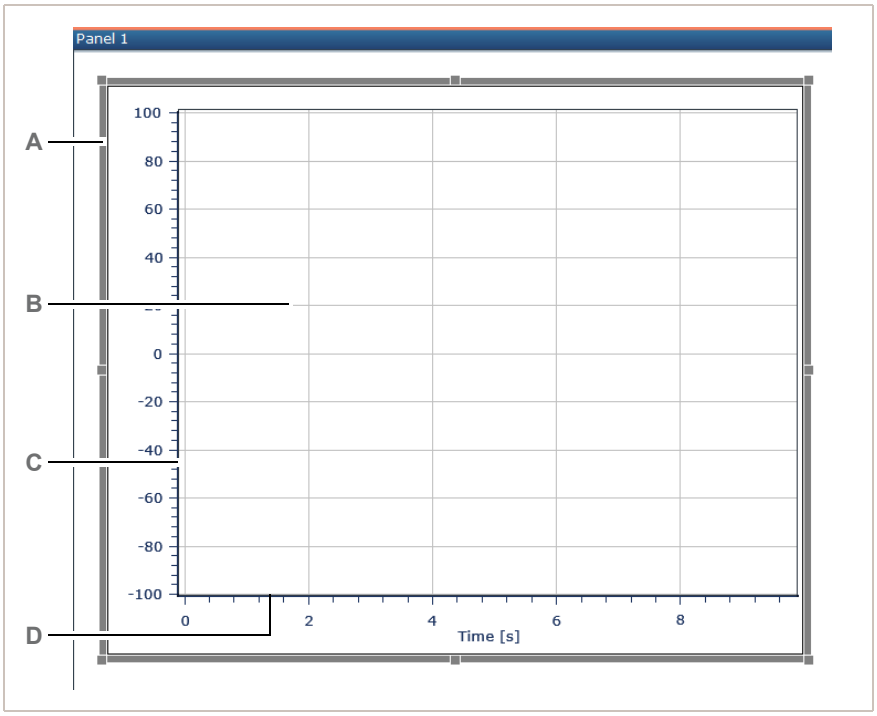
A y(t) chart visualizes measured values over time.

To visualize signals in a y(t) chart

1 Click the **Visualization** tab.

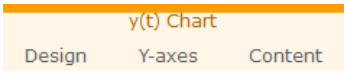
2 Click **y(t)**.

✓ An empty y(t) chart is displayed.



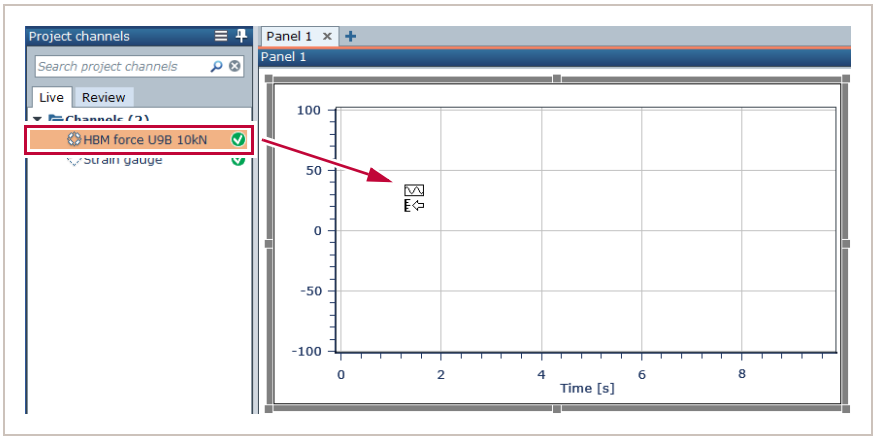
A	Frame	C	y-axis (with scale for measured values)
B	Trace area	D	x-axis (with time scale in [s])

ⓘ On the ribbon, the contextual tab set for y(t) charts is displayed (Contextual tabs for y(t) charts, 151).



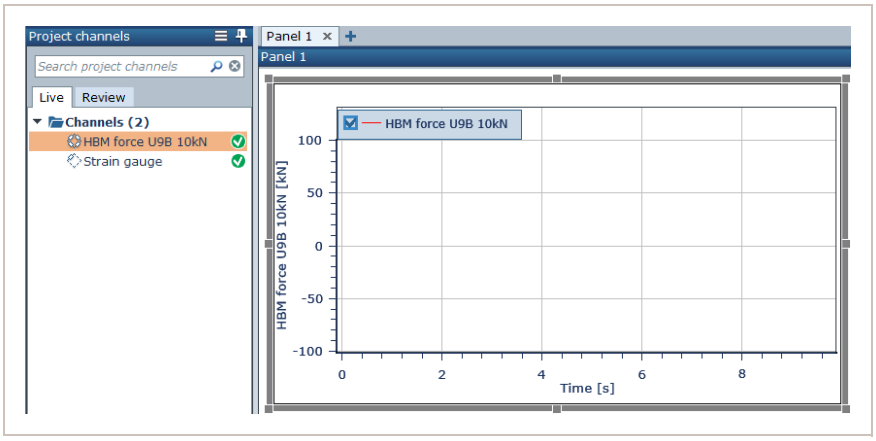
ⓘ Alternatively, drag a project channel to an empty space on the visualization panel and, on the context menu, click **Insert y(t) chart**. A y(t) chart is displayed and the project channel is assigned to the y-axis. However, for the rest of this procedure, it is assumed that you start with an empty y(t) chart.

3 Drag a live project channel or review channel to the chart, and when the pointer turns into a , release the pointer to assign the project channel to the y-axis.

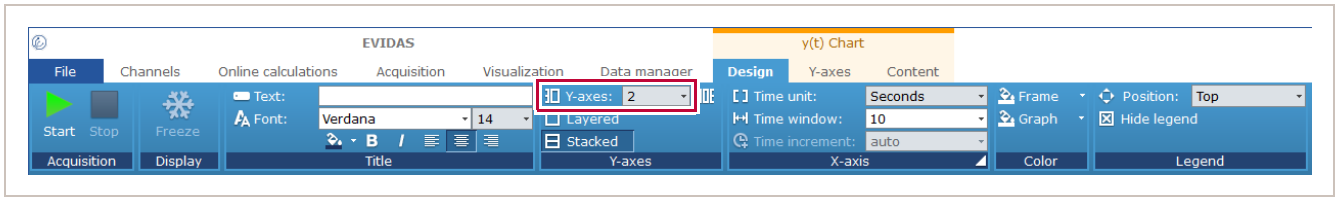


ⓘ In dual coordinate systems (Design tab for y(t) charts, 152), if you move the pointer towards the right y-axis, the pointer turns into a . Release the pointer to assign the project channel to the right y-axis.

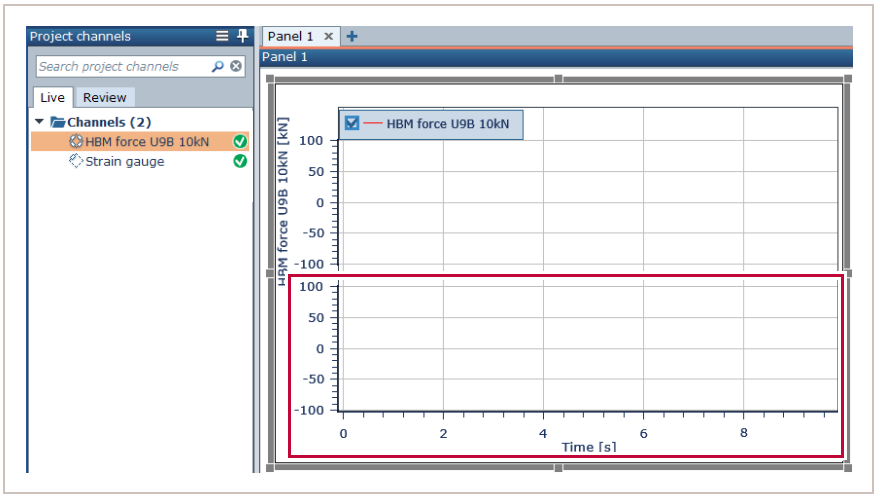
✓ The y-axis is labeled and a legend is displayed.



4 Click the **Design** tab and select **Y-axes** = **2**.



✓ A second coordinate system is displayed.



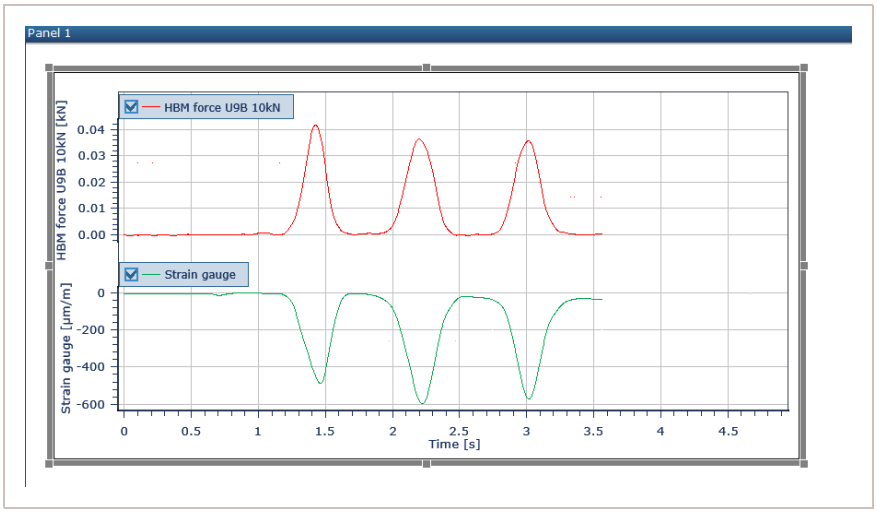
ⓘ You can display the coordinate systems stacked (see above) or layered. For layered systems, you can display the scales of the y-axes either all to the left, or to the left and to the right (dual) (Design tab for y(t) charts, 152↗).

5 Drag a project channel to the empty coordinate system.

✓ Both y-axes are labeled and a legend for each coordinate system is displayed.

6 Click ▶ **Start**.

✓ Both signals are displayed.



⇨ Contextual tabs for y(t) charts, 151↗

7.6.2 Visualizing signals in an X-Y chart

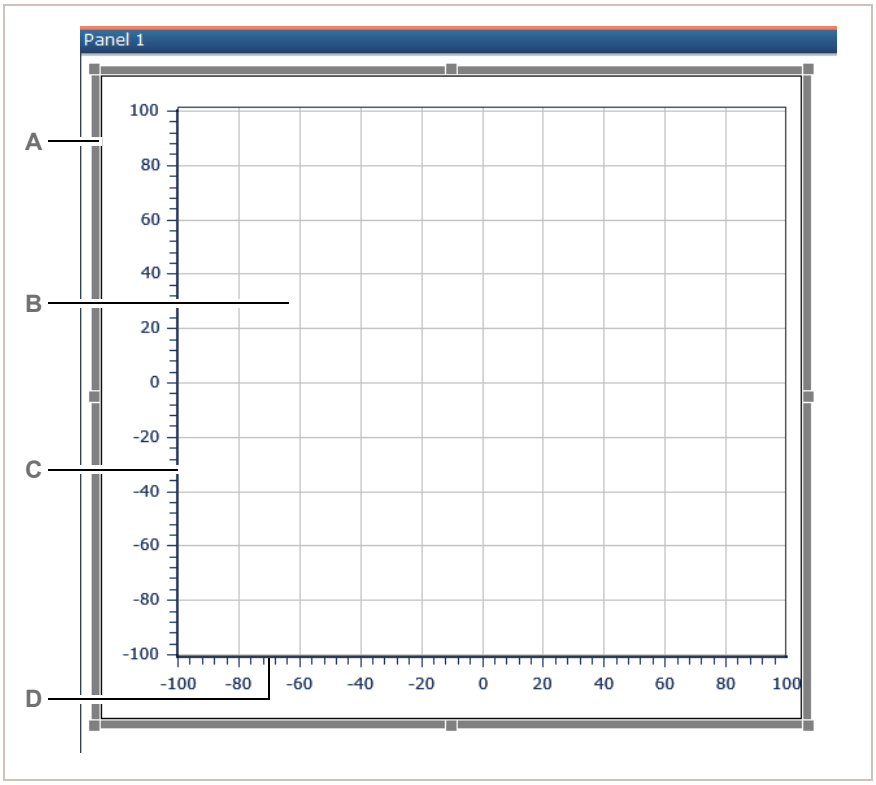
An X-Y chart visualizes the relationship between two signals. One signal is assigned to the x-axis, the other to the y-axis. The signal on the x-axis is called the X-channel, the signal on the y-axis is the Y-channel.

To visualize signals in an X-Y chart

1 Click the **Visualization** tab.

2 Click **X-Y**.

✓ An empty X-Y chart is displayed.

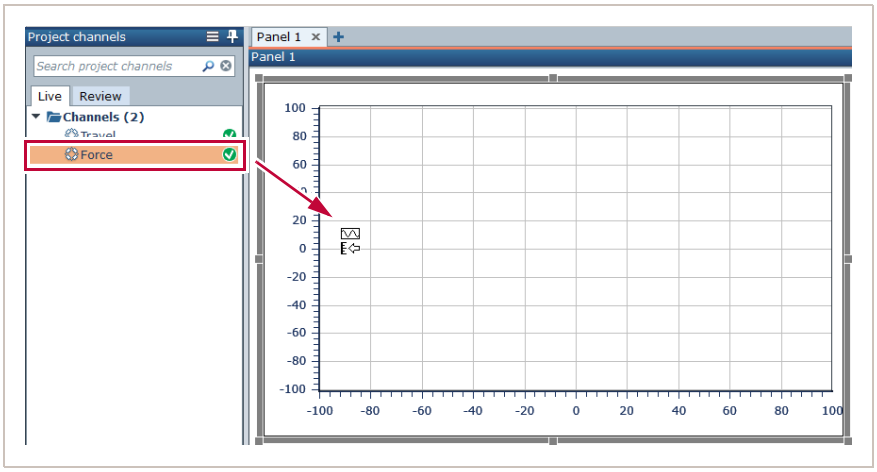


A	Frame	C	y-axis (with scale for Y-channel)
B	Trace area	D	x-axis (with scale for X-channel)

ⓘ On the ribbon, the contextual tab set for X-Y charts is displayed (Contextual tabs for X-Y charts, 156).

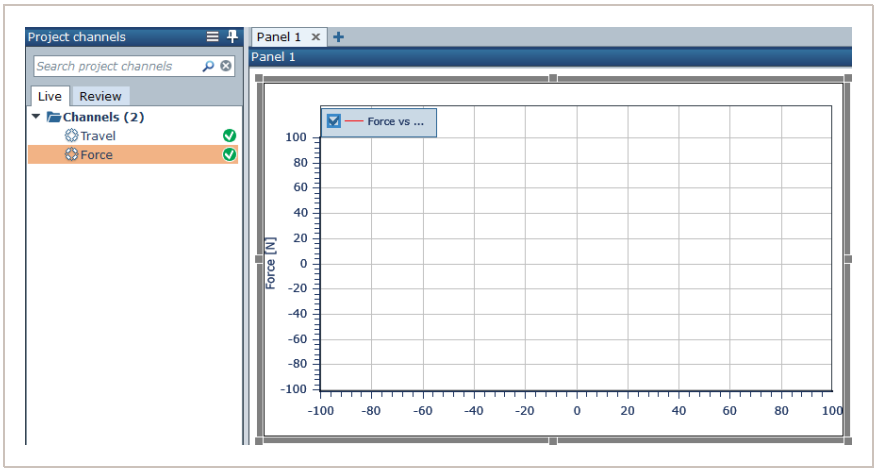
ⓘ Alternatively, drag a project channel to an empty space on the visualization panel and, on the context menu, click **Insert X-Y chart**. An X-Y chart is displayed and the project channel is assigned to the y-axis. However, for the rest of this procedure, it is assumed that you start with an empty X-Y chart.

3 Drag a project channel to the y-axis of the chart.

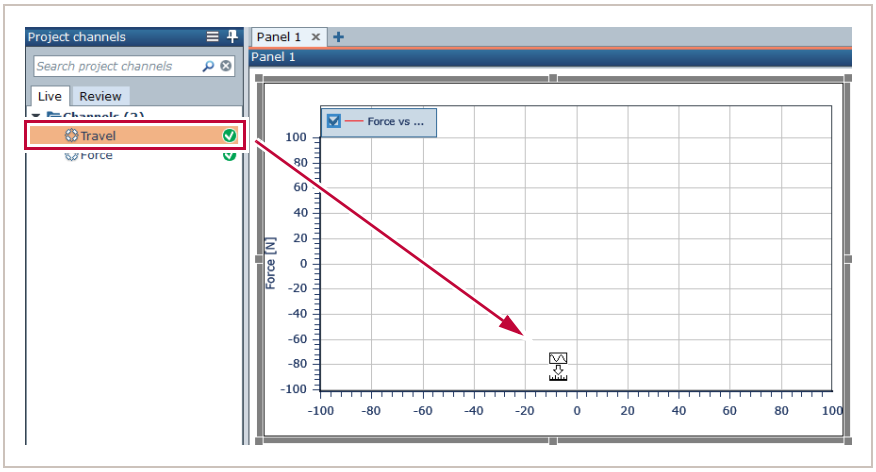


ⓘ When the pointer turns into a , release the pointer to assign the project channel to the y-axis.

✓ The y-axis is labeled and the first half of the legend is displayed.

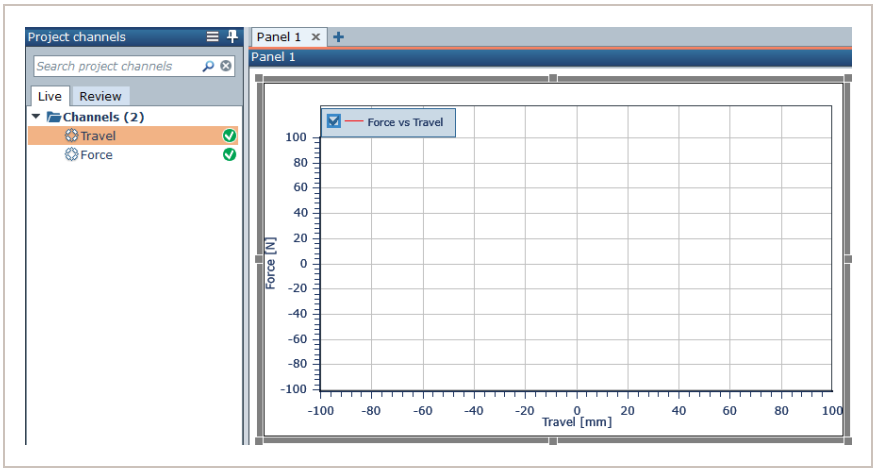


4 Drag a project channel to the x-axis of the chart.



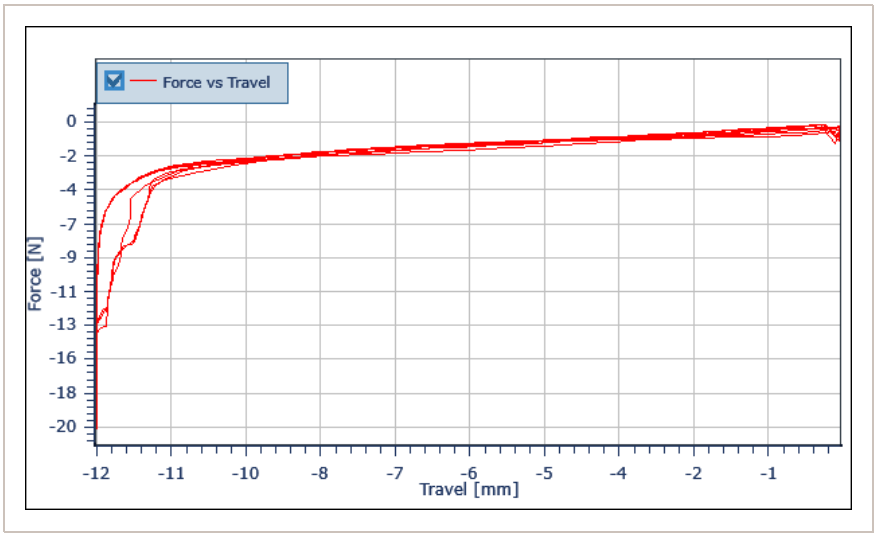
ⓘ When the pointer turns into a , release the pointer to assign the project channel to the x-axis.

✓ The x-axis is labeled and the full legend is displayed.



5 Click ▶ **Start**.

✓ The X-Y trace is displayed.



⇨ Contextual tabs for X-Y charts, 156

7.6.3 Visualizing the frequency spectrum of a signal in an FFT chart

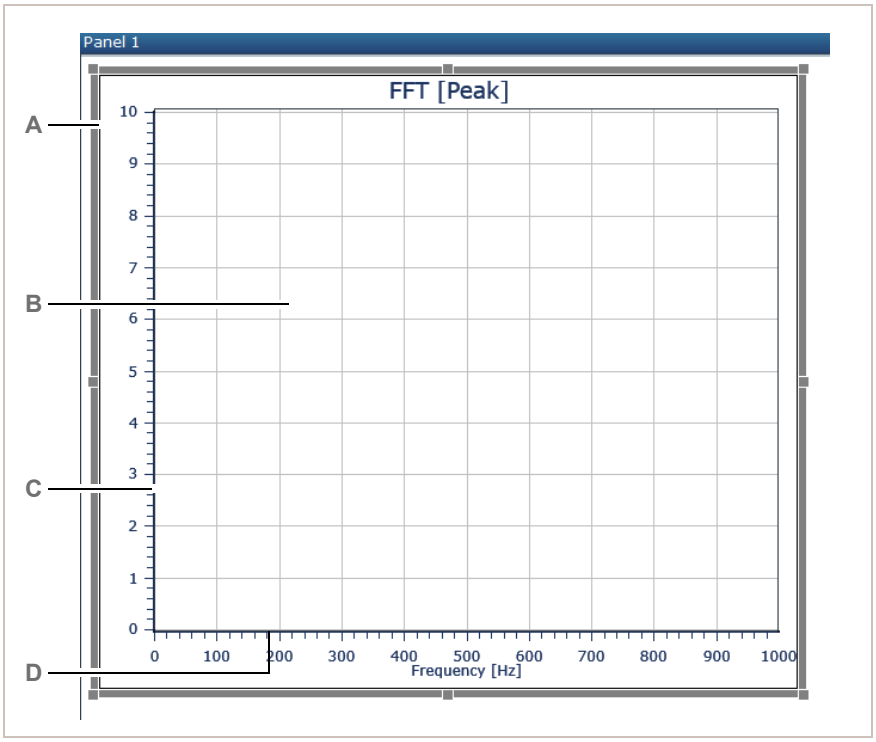
An FFT chart visualizes the frequency components of a signal. The amplitudes of the frequency components represent their respective contribution to the signal (Appendix D: FFT Introduction, 257).

To visualize the frequency spectrum of a signal in an FFT chart

1 Click the **Visualization** tab.

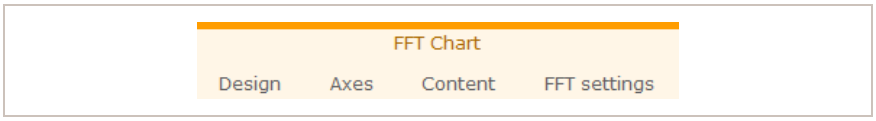
2 Click **FFT**.

✓ An empty FFT chart is displayed.



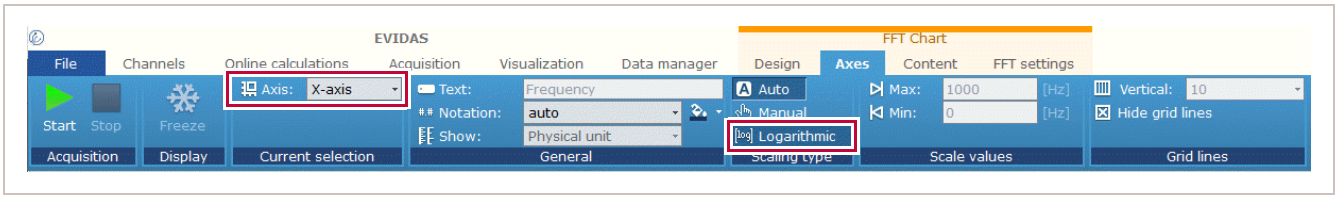
A	Frame	**C**	y-axis (amplitude)
B	Trace area	**D**	x-axis (frequency in [Hz])

ⓘ On the ribbon, the contextual tab set for FFT charts is displayed (Contextual tabs for FFT charts, 161).



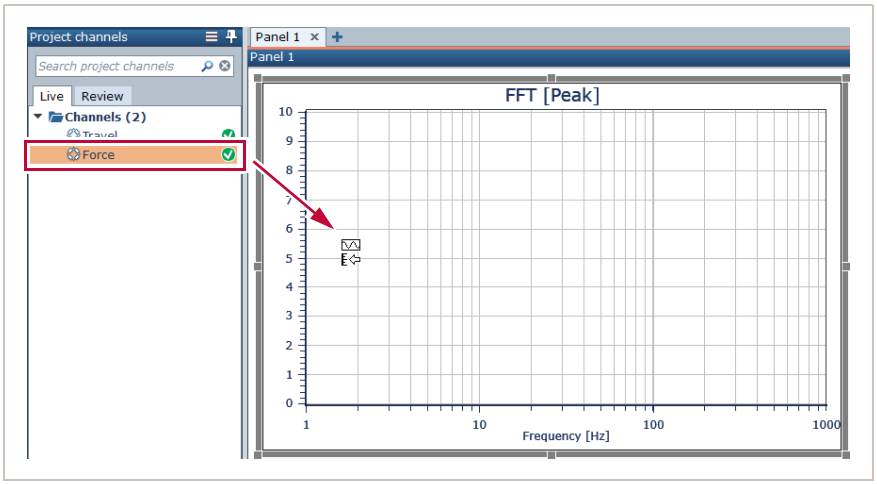
ⓘ Alternatively, drag a project channel to an empty space on the visualization panel and, on the context menu, click **Insert FFT chart**. An FFT chart is displayed and the project channel is assigned to the chart. However, for the rest of this procedure, it is assumed that you start with an empty FFT chart.

3 Click the **Axes** tab, select **Axis** = **X-axis**, and then click **Logarithmic**.



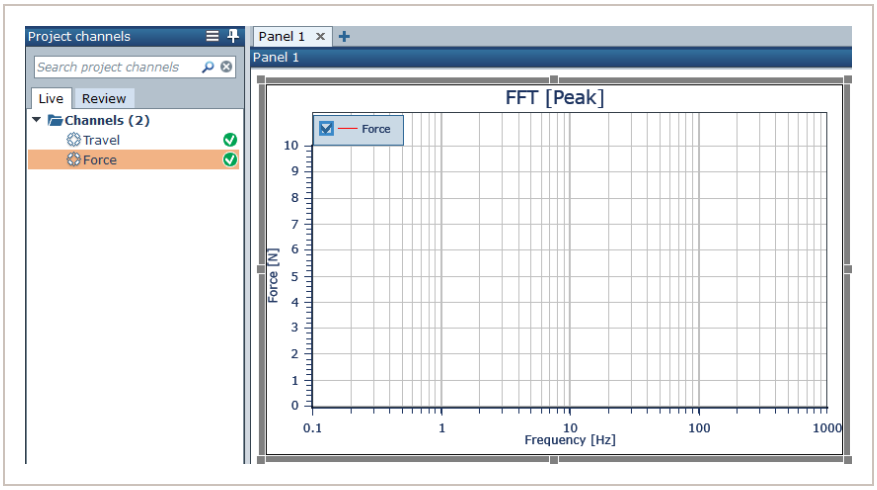
ⓘ The scaling of the x-axis is changed to logarithmic.

4 Drag a project channel to the FFT chart.



ⓘ When the pointer turns into a , release the pointer to assign the project channel to the FFT chart.

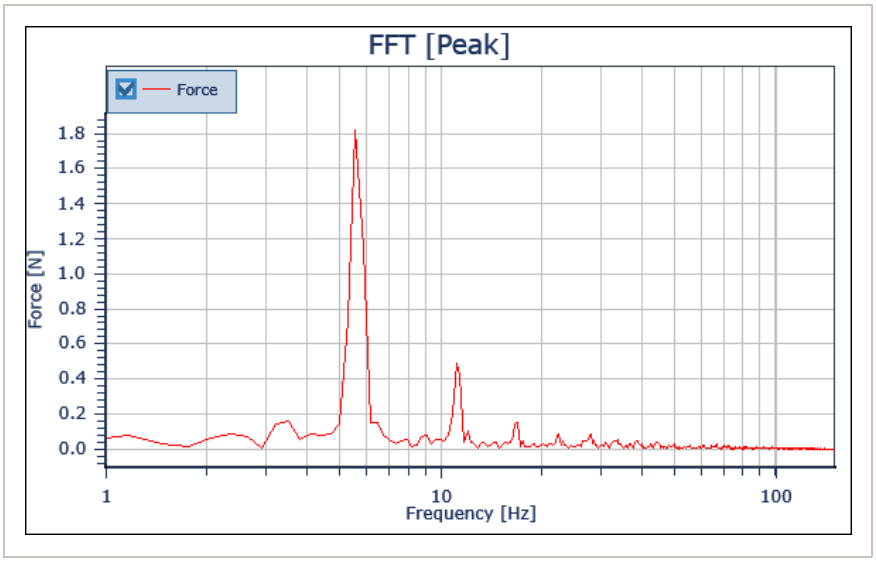
✓ The y-axis is labeled and a legend is displayed.



ⓘ On the y-axis, you can display the amplitude of the signal either in its physical unit, e.g., in Newton [N], or in decibel [dB] (Axes tab for FFT charts, 164⇗).

5 Click ▶ **Start**.

✓ The frequency spectrum of the signal is displayed.



⇨ Contextual tabs for FFT charts, 161⇗
Appendix D: FFT Introduction, 257⇗

7.6.4 Moving or copying a visualization object and changing its size

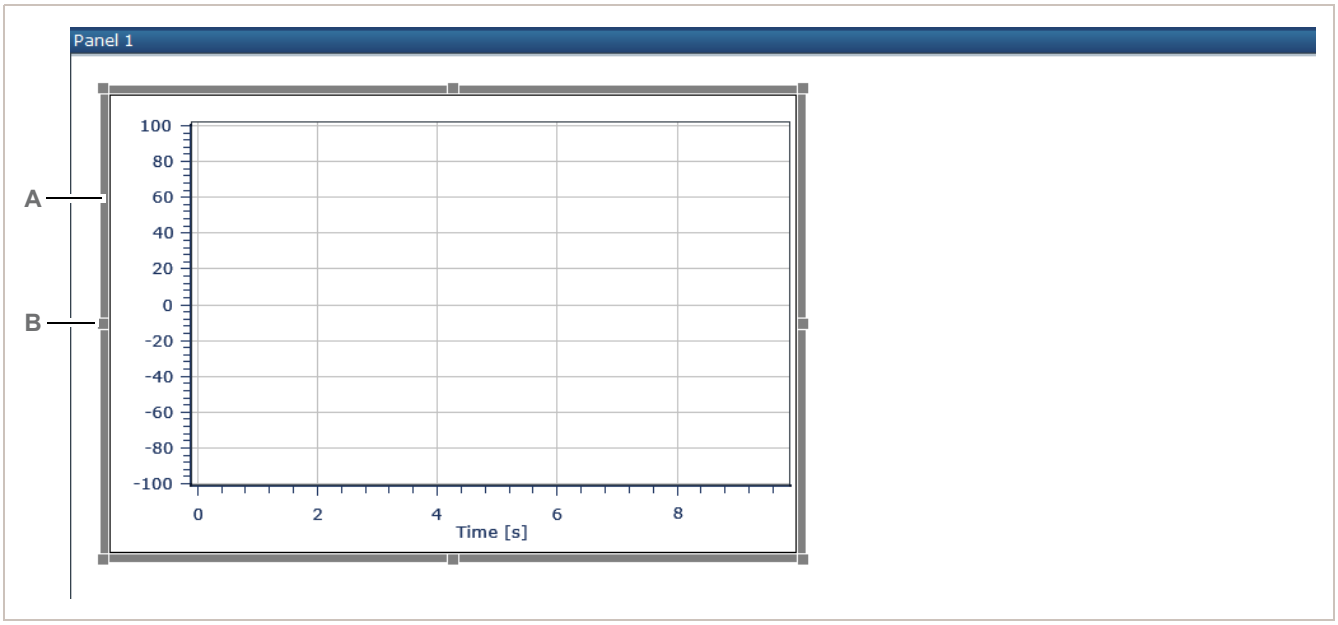
You can move a visualization object to any position on the visualization panel, you can copy the object, e.g., to another panel, and you can change the size of the object.

To move or copy a visualization object and to change its size

1 Click the **Visualization** tab.

2 Insert a visualization object, e.g., a y(t) chart (Visualizing signals in a y(t) chart, 179).

✓ The visualization object is displayed.

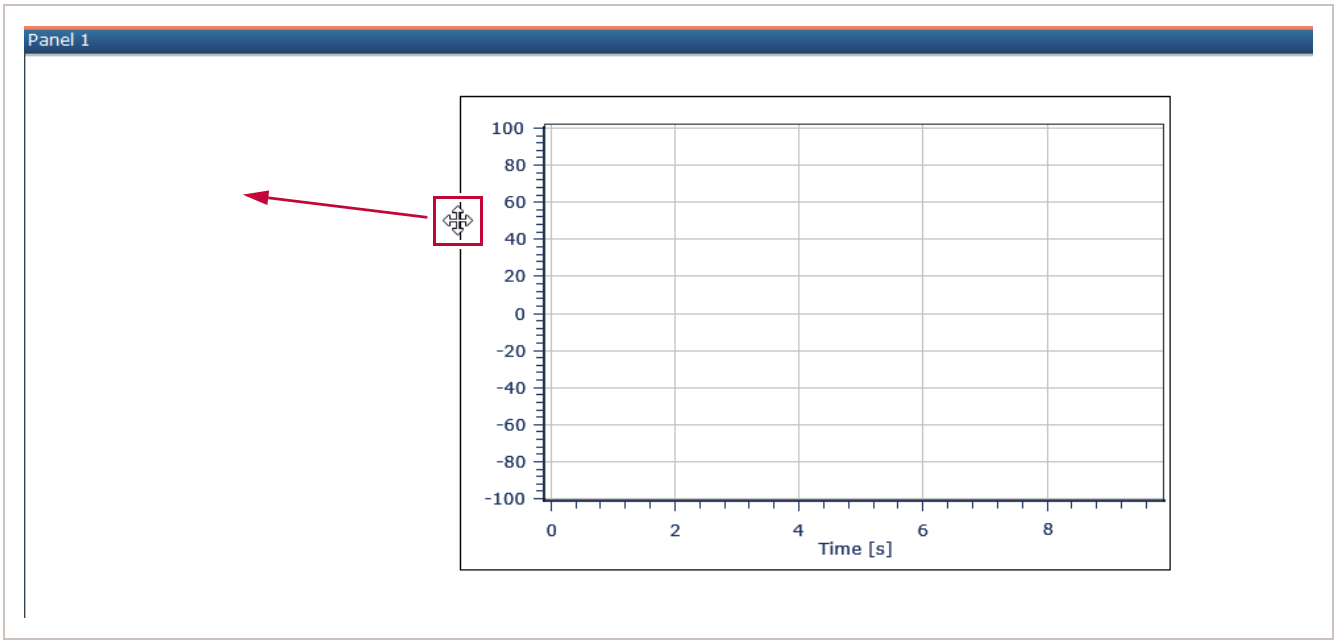


A Grey frame

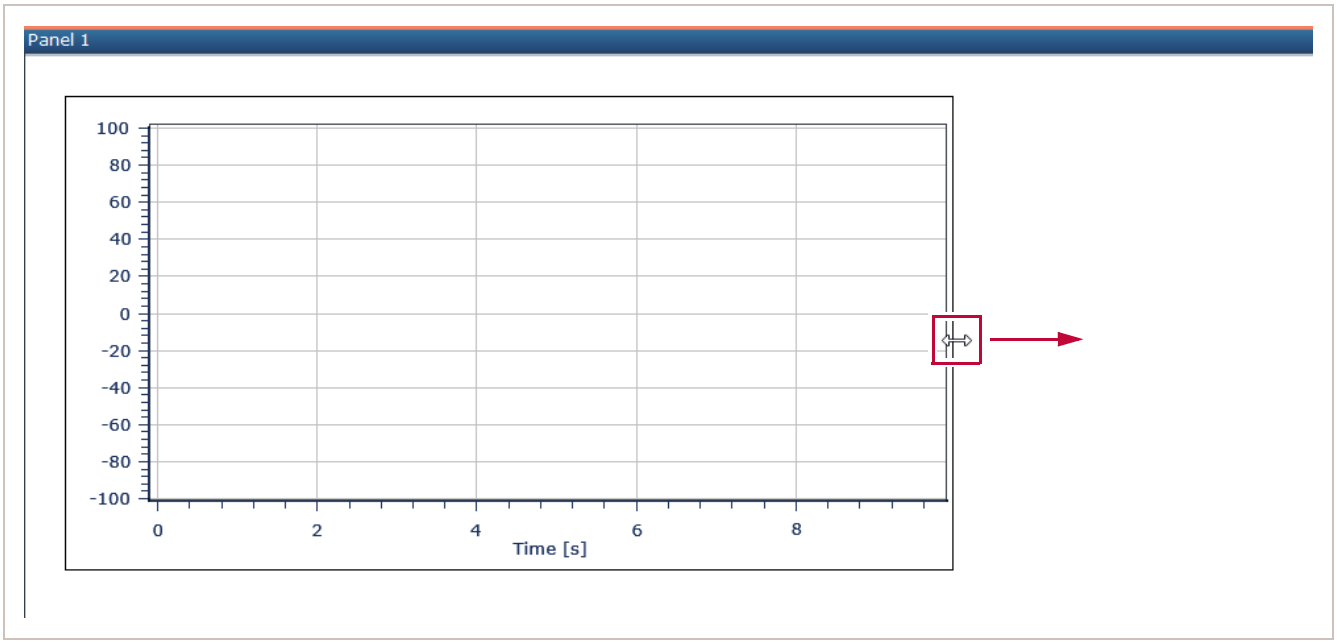
B Handle

ⓘ The grey frame indicates that the visualization object is active. The contextual tabs of the visualization object are displayed on the ribbon (Contextual tabs for visualization objects, 150).

3 Point to the grey frame. When the pointer turns into a ✥, drag the pointer to move the visualization object.



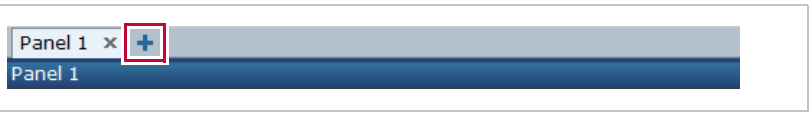
4 Point to one of the handles. When the pointer turns into a ⇔, drag the pointer to change the size of the visualization object.



5 Right-click the visualization object and, on the context menu, click **Copy**.

ⓘ The visualization object is copied to a buffer.

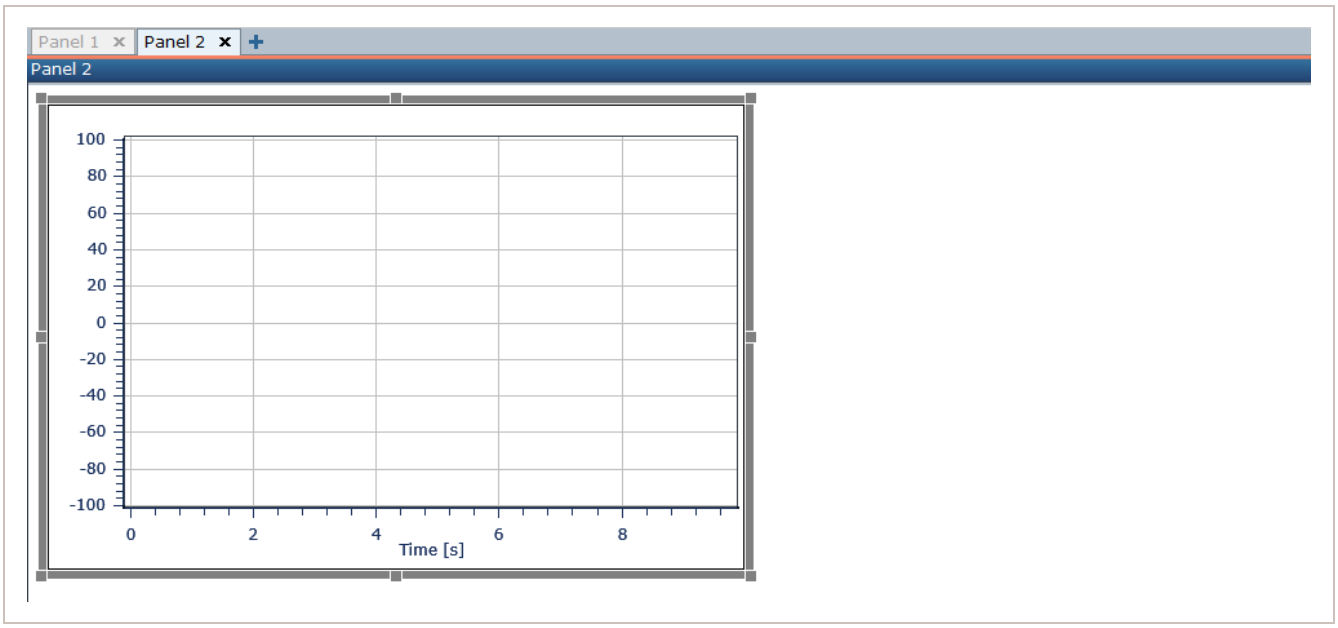
6 On the visualization panel header, click + (Add panel button, 177).



✓ A new visualization panel and its tab are displayed.

7 Right-click the new visualization panel and, on the context menu, click **Paste**.

✓ The visualization object is copied to the new panel.



⇨ Visualization commands, 146

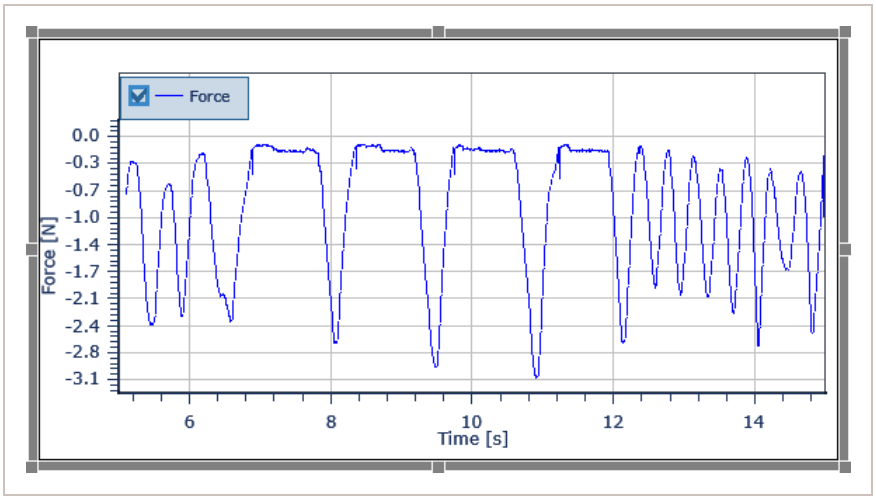
7.6.5 Deleting a trace

You can delete traces from any chart.

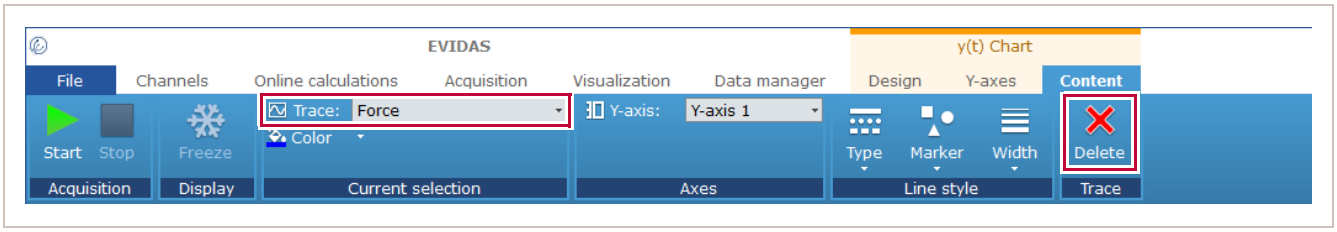
To delete a trace

1 Click a chart, e.g., a y(t) chart.

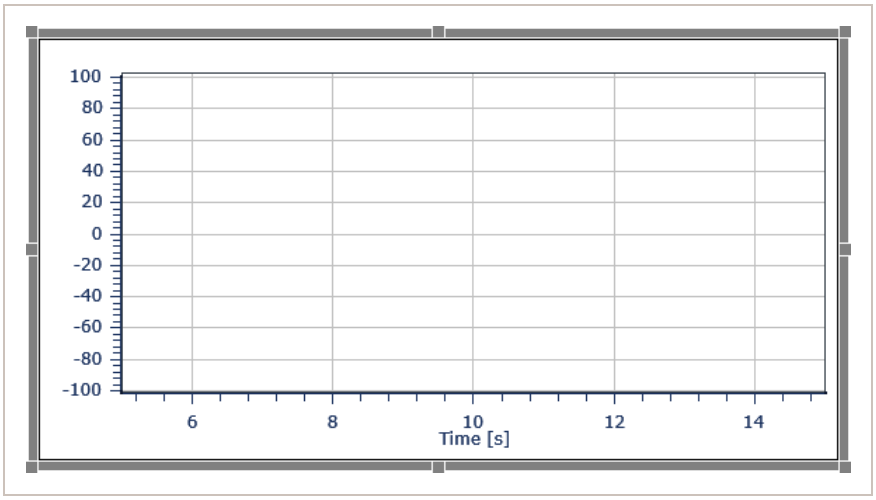
✓ A grey frame is displayed.



2 On the **Content** tab, select a trace, and then click **Delete**.



✓ The trace is removed from the chart.



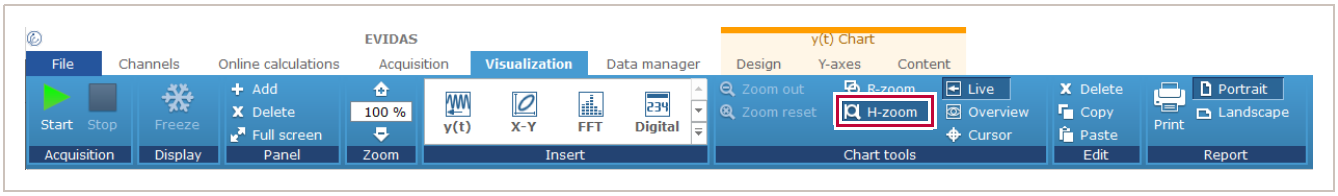
ⓘ Alternatively, right-click the legend of the trace, and then click **Delete trace from chart**.

7.6.6 Zooming a trace

You can zoom a section of a trace.

To zoom a trace

1 On the **Visualization** tab, click **H-zoom**.

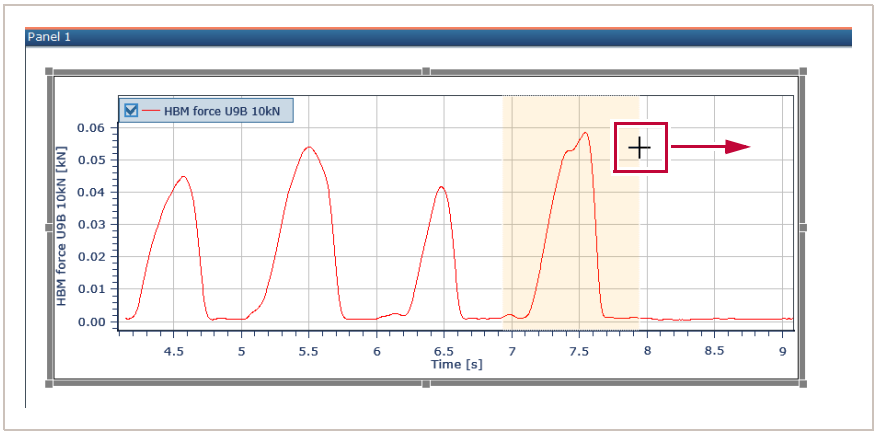


✓ The horizontal zoom tool is activated (Visualization commands, 146).

2 Click a chart, e.g., a y(t) chart.

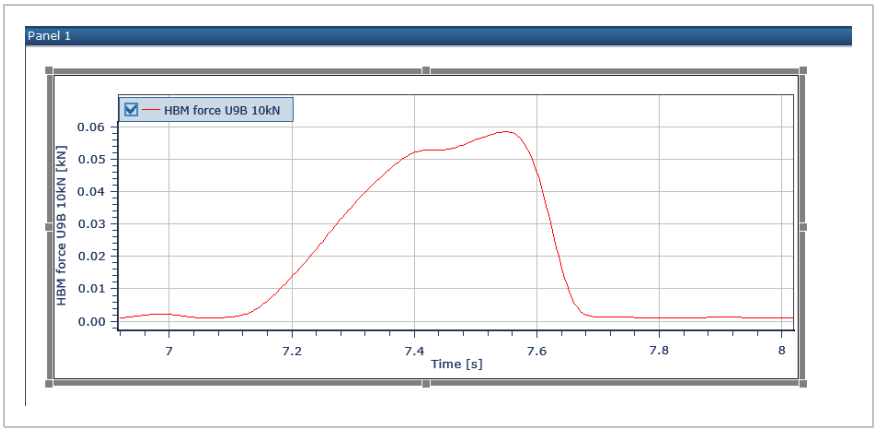
✓ A grey frame is displayed.

3 On the chart, click into the trace area and hold down the pointer. When the pointer turns into a ┼, drag the pointer to select a zoom window.



4 Release the pointer.

✓ The section is zoomed.



5 Click **Zoom out** to restore the previous view.

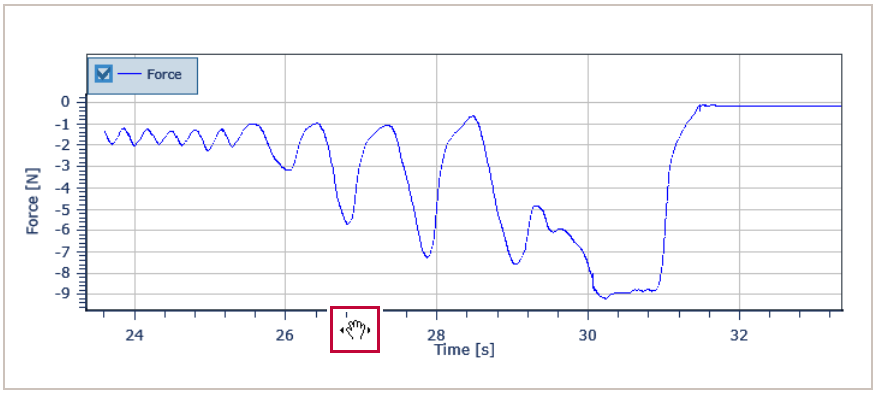
7.6.7 Scrolling a trace

You can scroll a trace horizontally or vertically.

To scroll a trace

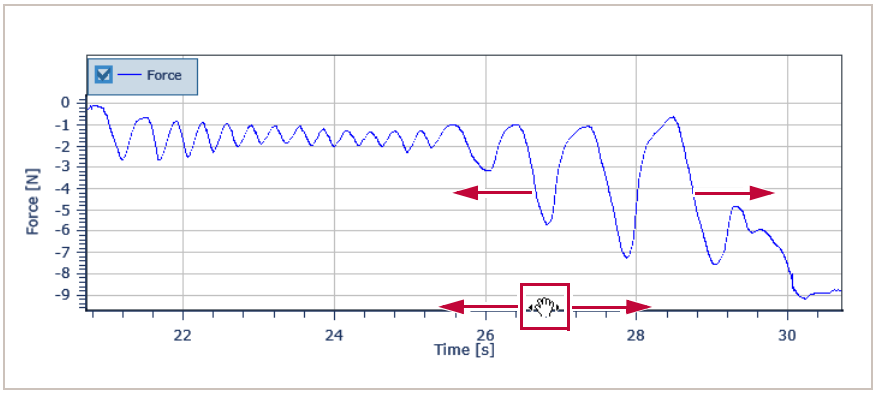
1 Point to the x-axis, e.g., of a y(t) chart.

✓ The pointer turns into a ‹›.



2 Drag the pointer to the left or right.

✓ The trace is scrolled horizontally.

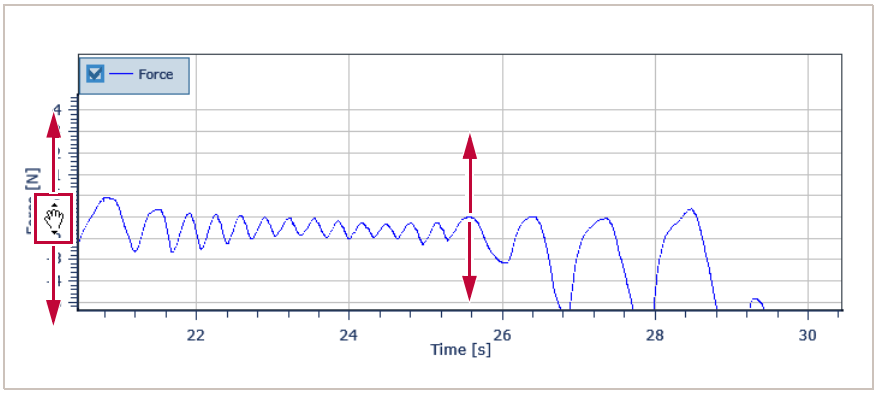


3 Point to the y-axis

✓ The pointer turns into a .

4 Drag the pointer up or down.

✓ The trace is scrolled vertically.



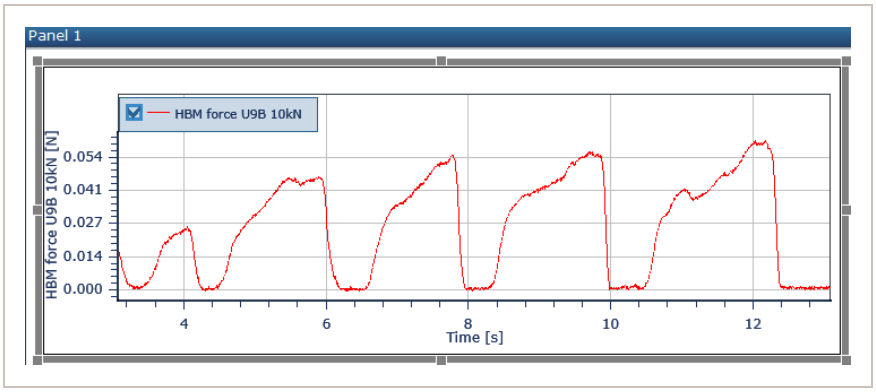
7.6.8 Changing the line style of a trace

The following procedure shows how you can change the line style of a trace.

To change the line style of a trace

1 Click a chart, e.g., a y(t) chart.

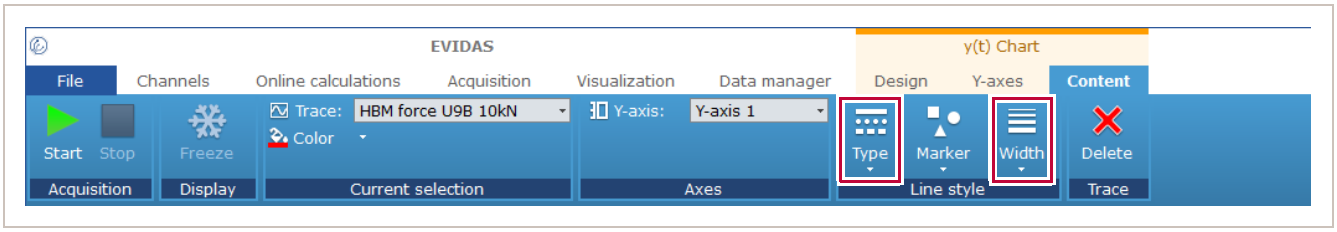
✓ A grey frame is displayed.



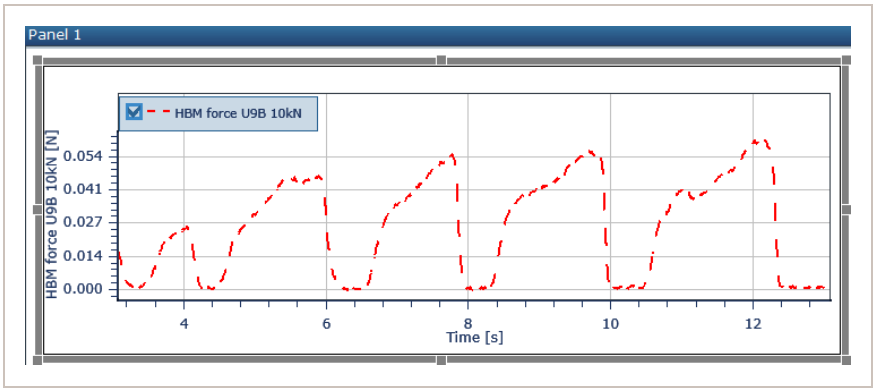
2 On the **Content** tab, select the following options:

a) **Type** = **Dash**.

b) **Width** = **2**.



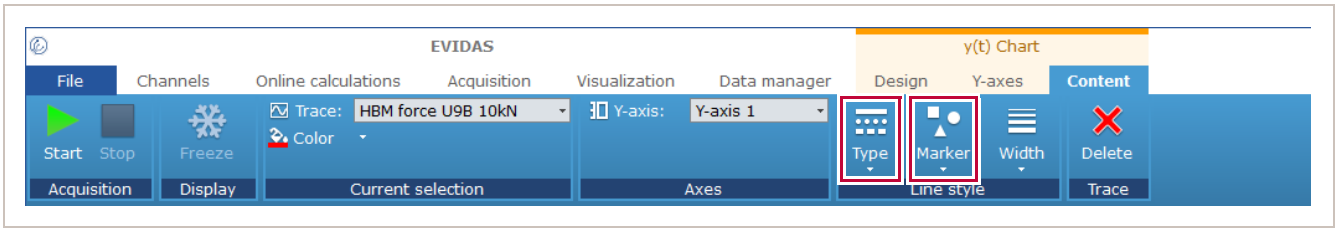
✓ The trace is displayed dashed and bold.



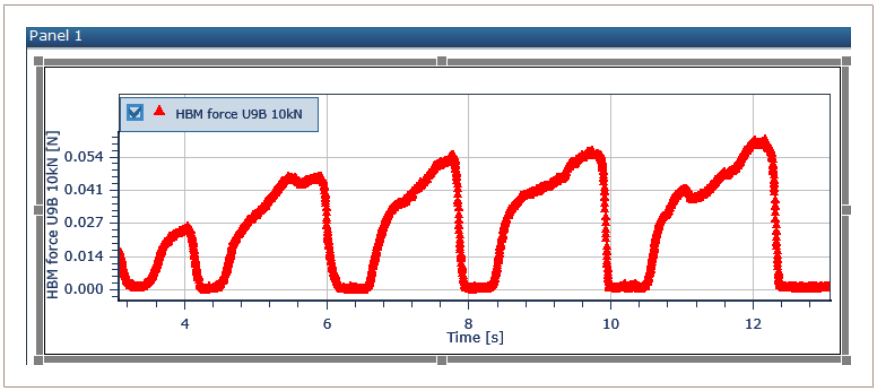
3 Or select the following options:

a) **Type** = **None**.

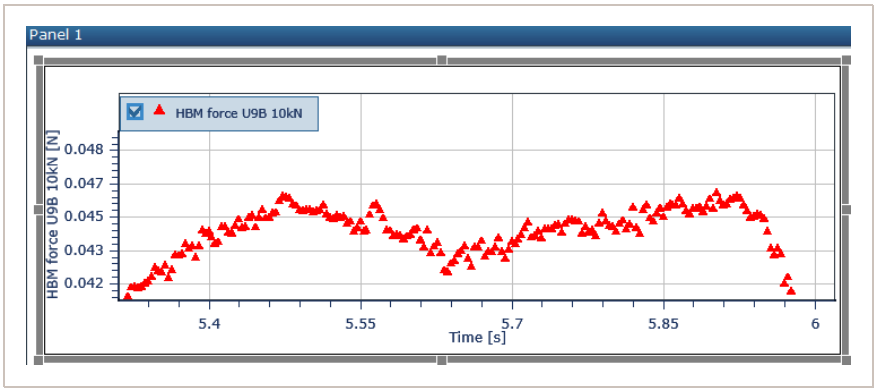
b) **Marker** = **Triangle**.



✓ The markers representing the measured values are displayed.



ⓘ If you cannot see the markers, zoom the trace (Zooming a trace, 193).



7.6.9 Working on several visualization objects at the same time

You can select several visualization objects in order to work on them at the same time.

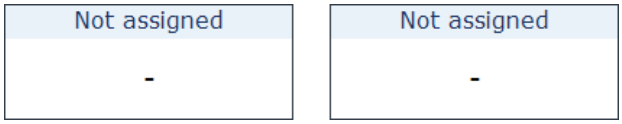
If you select visualization objects of different types, e.g., a y(t) chart and a digitial meter, you can move, copy, or delete them together.

If the selected visualization objects are of the same type, e.g., two digital meters, you can change many of their design features together, e.g., the design of the title, borders, and values displayed. However, you cannot change content features of several visualization objects together.

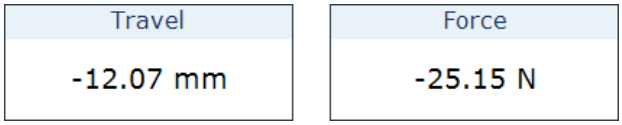
To work on several visualization objects at the same time

1 Click the **Visualization** tab.

2 Click **Digital** twice and drag the digital meters next to each other.

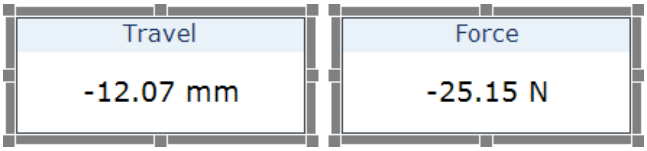


3 Assign a project channel to each digital meter and click **Start**.

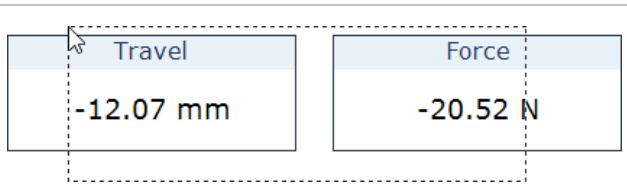


4 Hold down Ctrl while clicking each of the two digital meters.

✓ A grey frame is displayed for each digital meter.

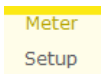


ⓘ Alternatively, point away from the digital meters, drag a frame across them, and then release the pointer.

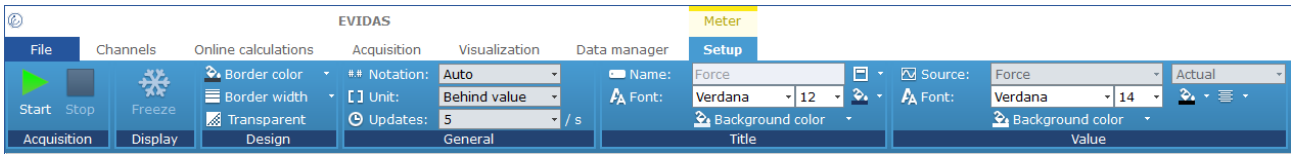


ⓘ The visualization objects are selected even if the frame overlaps only partially.

5 On the **Meter** contextual tab, click **Setup**.



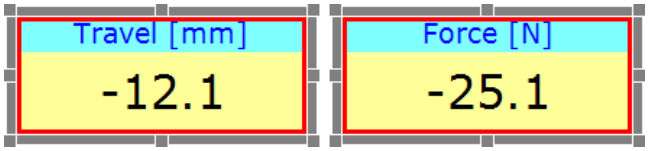
✓ The **Setup** tab for digital meters is displayed.



ⓘ Settings that are displayed active (not grey) can be edited for both digital meters at the same time.

6 Experiment with the settings.

✓ The respective features are changed for both digital meters at once.



7 To move several visualization objects together, select them, point to one of the grey frames, and then drag the objects to the desired location.

✓ The selected objects are moved to the new location.

8 To copy and paste several visualization objects together, select them, right-click them, select **Copy** from the context menu, right-click somewhere else on the same or on another visualization panel, and then select **Paste** on the context menu.

✓ A copy of the selected objects is inserted.

9 To delete several visualization objects together, select them, right-click them, and then select **Delete …** and confirm the delete command.

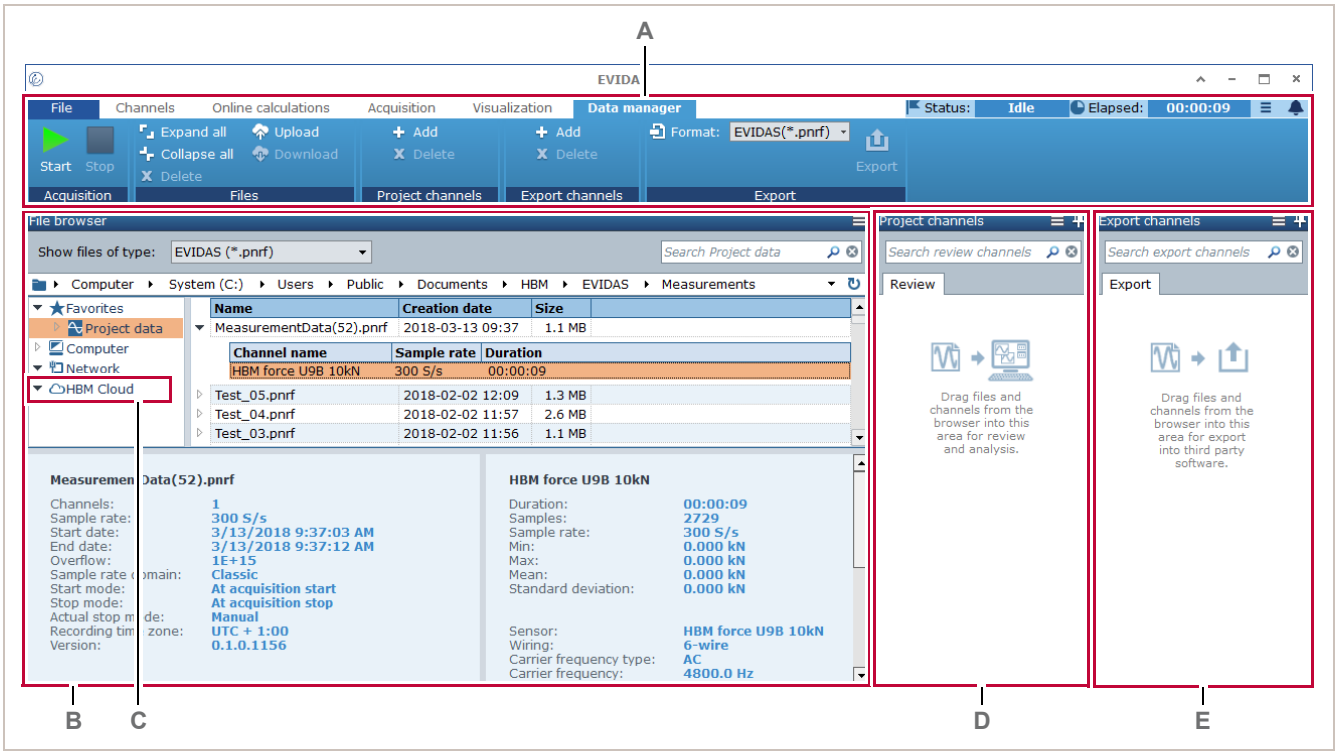
✓ The selected objects are deleted.

⇨ Contextual tab for digital meters, 169

8 Data manager

On the **Data manager** tab and its panels, you can do the following:

- Browse folders (File browser, 205↗).
- Browse data files (Traceability data, 207↗).
- Define favorite folders (Folder list, 206↗).
- Open folder in Explorer (Opening a folder in Explorer, 208↗).
- Upload data files to HBM cloud (HBM cloud, 208↗).
- Select channels for review (Project channels panel in Data manager, 210↗).
- Export channels to various file formats (Export channels panel, 211↗).



A Data manager tab, 203↗

B File browser, 205↗

C HBM cloud, 208↗

D Project channels panel in Data manager, 210↗

E Export channels panel, 211↗

Review channels

To select a channel for review, drag it from the **File browser** to the **Project channels** panel. The channel is now available as a review channel in **Visualization** (Visualization, 145↗).

You can display a review channel and a live signal in the same y(t) chart (Comparing a live signal with a previous test, 212↗).

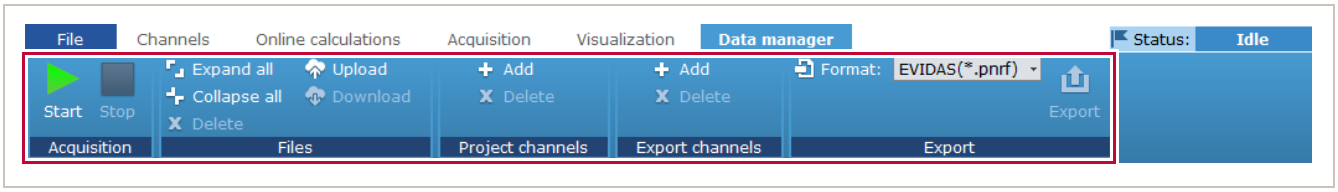
Export channels

To export a channel, drag it from the **File browser** to the **Export channels** panel, select an export format, e.g., **Excel**, and then click **Export**.

During export, all channels on the **Export channels** panel are merged into one export file.

Therefore, you can use the export function to merge channels from different files into one file (Merging channels from different files, 216).

8.1 Data manager tab



Data manager commands

Group	Command	Description
Files	**Expand all**	To display the channels of each file in the **File browser**.
	Collapse all	To hide all channels displayed in the **File browser**.
	Delete	To delete a file in the **File browser**. Click the file you want to delete, and then click **Delete**.
	Upload	To upload a file to the HBM cloud. In the **File browser**, click the file you want to upload, and then click **Upload**. The file is uploaded to the HBM cloud (HBM cloud, 208).
	Download	To download a file from the HBM cloud. In the **File browser**, click **HBM cloud**, click the file you want to download, and then click **Download**. The file is downloaded to the **Project data** folder (Data default folder and file name, 136).
Project channels	**Add**	To copy a file from the **File browser** to the **Project channels** panel (Project channels panel in Data manager, 210). Click the file or channel you want to review, and then click **Add**. Alternatively, you can drag the file or channel to the **Project channels** panel.
	Delete	To delete a channel or file from the **Project channels** panel. On the **Project channels** panel, click the channel or file you want to delete, and then click **Delete**.
Export channels	**Add**	To copy a file from the **File browser** to the **Export channels** panel (Export channels panel, 211). Click the file or channel you want to export, and then click **Add**. Alternatively, you can drag the file or channel to the **Export channels** panel. You can only export data files that are in the EVIDAS data file format (*.pnrf).
	Delete	To delete a channel or file from the **Export channels** panel. On the **Export channels** panel, click the channel or file you want to delete, and then click **Delete**.

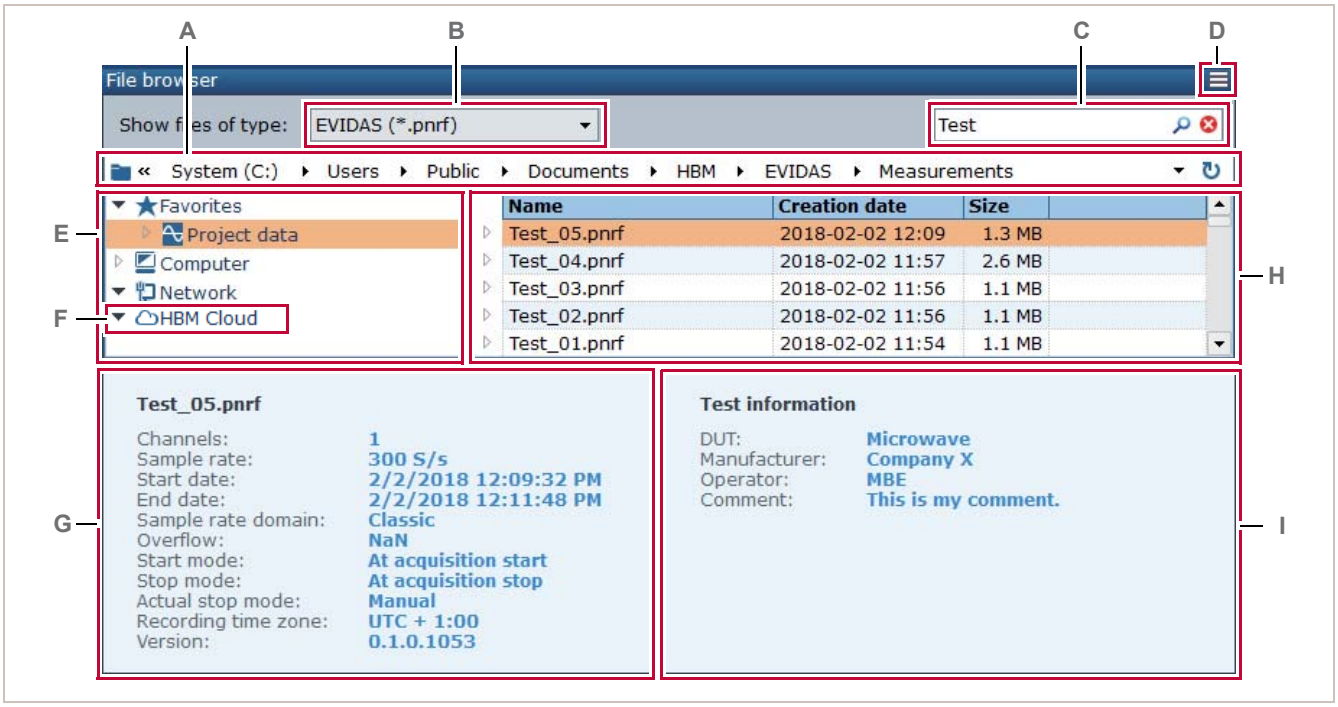
Data manager commands

Group	Command	Description
Export	**Format**	To select an export format, e.g., **EVIDAS (*.pnrf)** or **MATLAB (*.mat)**.
	Export	To export all channels on the **Export channels** panel to a file in the selected export format (Export channels panel, 211).

⇨ Start / Stop button, 40

8.2 File browser

To display, click the **Data manager** tab.



- **A** Path, 205
- **B** Format filter, 206
- **C** Filename filter, 206
- **D** Burger menu, 206
- **E** Folder list, 206
- **F** HBM cloud, 208
- **G** Recording information, 206
- **H** File list, 206
- **I** Traceability data, 207

In the **File browser**, you can select a folder in the folder list (E) to display its files in the file list (H). In the file list you can filter by name (C) and extension, i.e., by file format (B). Click a file to display metadata on the recording (G).

Double-click a file to display its channels. You can drag channels to the **Project channels** panel to select them as review channels (Project channels panel in Data manager, 210).

You can also drag channels to the **Export channels** panel for export (Export channels panel, 211).

Click a channel to display its traceability data (I), i.e., the sensor settings, some basic statistical data on the recording, and an optional comment from the **Project channels** panel (Project channels panel, 107) regarding the channel.

Path

This box displays the full pathname of the folder selected in the folder list (E).

The files of the selected folder are displayed in the file list (H).

Format filter

You can filter the file list by extension, e.g., to display catman data files (*.bin) in the selected folder.

Filename filter

You can filter the file list for a string in the filename, e.g., to display all files recorded on a specific date if you used placeholders to insert the date into the filenames.

Burger menu

Command	Description
Open folder in Explorer	To open the selected folder in the Explorer (Opening a folder in Explorer, 208).
Help on file browser	To access context-sensitive help.

Folder list

In the folder list, you select a folder to view its files.

The default folder for data files is the **Project data** folder in **Favorites**. The default folder is defined in **Acquisition** (Data default folder and file name, 136).

You can define your own favorite folders:

- To add a folder to **Favorites**, right-click the folder in the folder list, and then click **Add to favorites**.
- To remove a folder from **Favorites**, right-click the folder in **Favorites**, and then click **Remove from favorites**.

File list

The file list displays the files in the selected folder that meet the filter criteria (Filename filter, 206 , Format filter, 206).

Filenames in grey indicate that those files cannot be selected for review or export.

Use the following arrow keys to display traceability data on files and channels:

Arrow key	Function
Up / Down	To display information on either the files or the channels.
Left / Right	To display information on both the files and their channels.

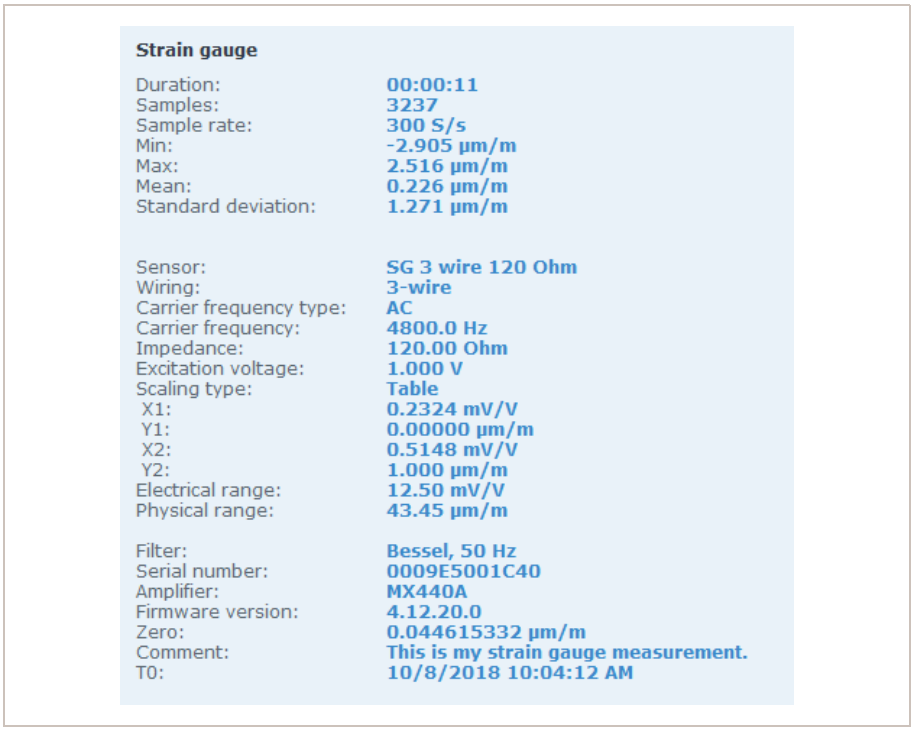
Recording information

In this area, the filename and metadata on the recording are displayed, e.g., the number of channels and the sample rate.

Traceability data

The information displayed in this area depends on whether you select a file or channel.

Selection in file list	Content
File	The user-defined test information and comment are displayed. This information is defined on the **Data file** tab (Metadata, 140↗).
Channel	The sensor settings and some basic statistical data on the measured values are displayed. If you added a **Comment** column on the **Project channels** panel (Burger menu, 108↗), any comment typed into this column is displayed with the traceability data of the respective channel.



☐ Sensor settings and basic statistical data on a strain gauge recording (example)

8.3 Opening a folder in Explorer

To supplement the functionality of the **Data manager** (Data manager, 201), you can open a folder in the **Explorer**, e.g., to rename, move, or copy data files.

To open a folder in Explorer

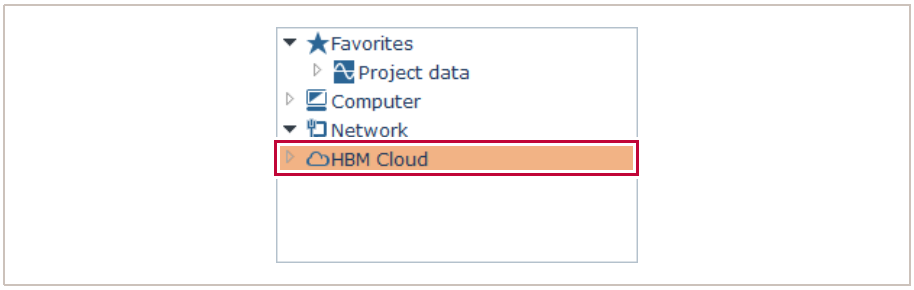
1 Click the **Data manager** tab.

2 In the **File browser**, right-click a folder or file, and then click **Open folder in Explorer**.

✓ All files of the current folder are displayed in the **Explorer**.

8.4 HBM cloud

To display, click the **Data manager** tab.



The HBM cloud provides external storage capacity for EVIDAS data files (*.pnrf). You can also upload catman data files (*.bin).

HBM cloud activation

By default, the HBM cloud is deactivated. To activate the HBM cloud, see: Activating the HBM cloud, 23.

HBM cloud features

Each EVIDAS license includes 5 GB of cloud storage capacity. Please contact HBM if you need more (Manufacturer, 4).

You can use the HBM cloud to backup EVIDAS data files (*.pnrf) and to share them with colleagues or customers (HBM cloud storage website, 209).

Your data is protected from loss through copies at a redundant cloud storage location of the cloud hosting provider.

The test information (Metadata, 140) and traceability data (Traceability data, 207) of the EVIDAS data files in the cloud are displayed in the **File browser** in the same way as for EVIDAS data files on your computer or network.

HBM cloud upload options

Option	Description
Manual upload	Click a file in the **File browser**, and then click **Upload** (Data manager tab, 203).
Automatic upload	Select the **Cloud storage** option (Cloud storage, 138). A copy of each new EVIDAS data file is uploaded to the HBM cloud.

HBM cloud storage website

Once your data files are uploaded to the HBM cloud, you can access them via any web browser on **https://storage.hbm-solutions.com (Internet Explorer not supported)**.

To access data files on the HBM cloud storage website, you, or any person you want to share data with, will need your email address and cloud password.

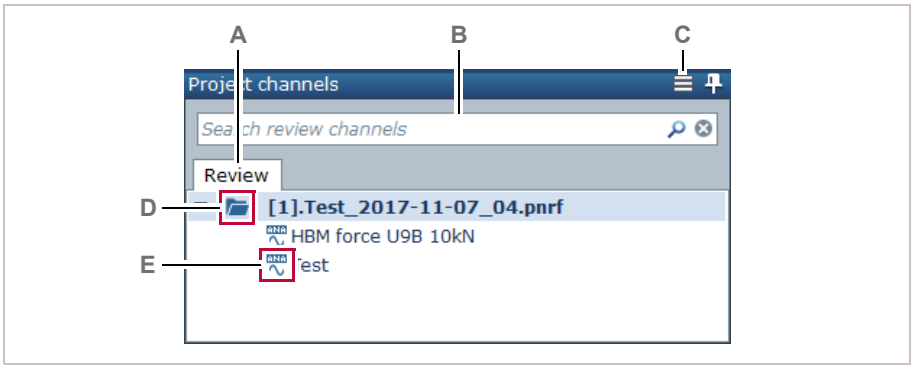
A temporary password is mailed to your email address when activating the HBM cloud (Activating the HBM cloud, 23).

The HBM cloud storage website provides the following services:

- Display of total cloud storage capacity.
- Display of remaining cloud storage capacity.
- Display of cloud storage location, e.g., West Europe.
- List of data files.
- List of project channels for each data file.
- Data file download.

8.5 Project channels panel in Data manager

To display, click the **Data manager** tab.



A Review channels, 210
B Search box, 47
C Burger menu, 210
D Folder icons, 210
E Channel icons, 210

Review channels

To select a channel for review, drag it from the **File browser** (File browser, 205) to the **Project channels** panel.

If required, the data file is converted to EVIDAS format (*.pnrf) (File formats, 139). For example, if you drag a catman data file (*.bin) to the **Project channels** panel, the file is converted to *.bin.pnrf.

The channels on the **Project channels** panel are now available as review channels in **Visualization** (Visualization, 145)

You can display a review channel and a live signal in the same y(t) chart (Comparing a live signal with a previous test, 212).

Burger menu

Command	Description
Help on review channels	To access context-sensitive help.

Folder icons

Channels folder (collapsed).

Channels folder (expanded).

Channel icons

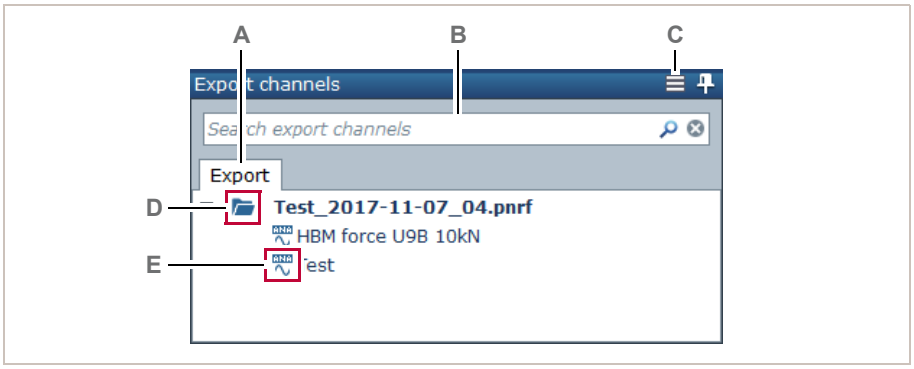
Analog channel.

CAN channel.

Digital channel.

8.6 Export channels panel

To display, click the **Data manager** tab.



A Export channels, 211
B Search box, 47
C Burger menu, 211
D Folder icons, 210
E Channel icons, 210

Export channels

To export a channel, drag it from the **File browser** (File browser, 205) to the **Export channels** panel, select an export format, e.g., **Excel**, and then click **Export**.

You can also drag a review channel from the **Project channels** panels to the **Export channels** panel, select an export format, and then click **Export**.

During export, all channels on the **Export channels** panel are merged into one export file.

You can only export data files that are in the EVIDAS data file format (*.pnrf) (File formats, 139). If you want to export a file of a different format, e.g., a catman data file (*.bin), drag it from the **File browser** to the **Project channels** panel. The data file is converted to EVIDAS data file format (*.pnrf).

On the **Project channels** panel, right-click the file, and then click **Add to export channels**.

You can use the export function to merge channels from different files into one file in EVIDAS data file format (Merging channels from different files, 216).

Burger menu

Command	Description
Help on export channels	To access context-sensitive help.

8.7 Comparing a live signal with a previous test

In a y(t) chart, you can use a channel from a previous test as reference channel for a live signal.

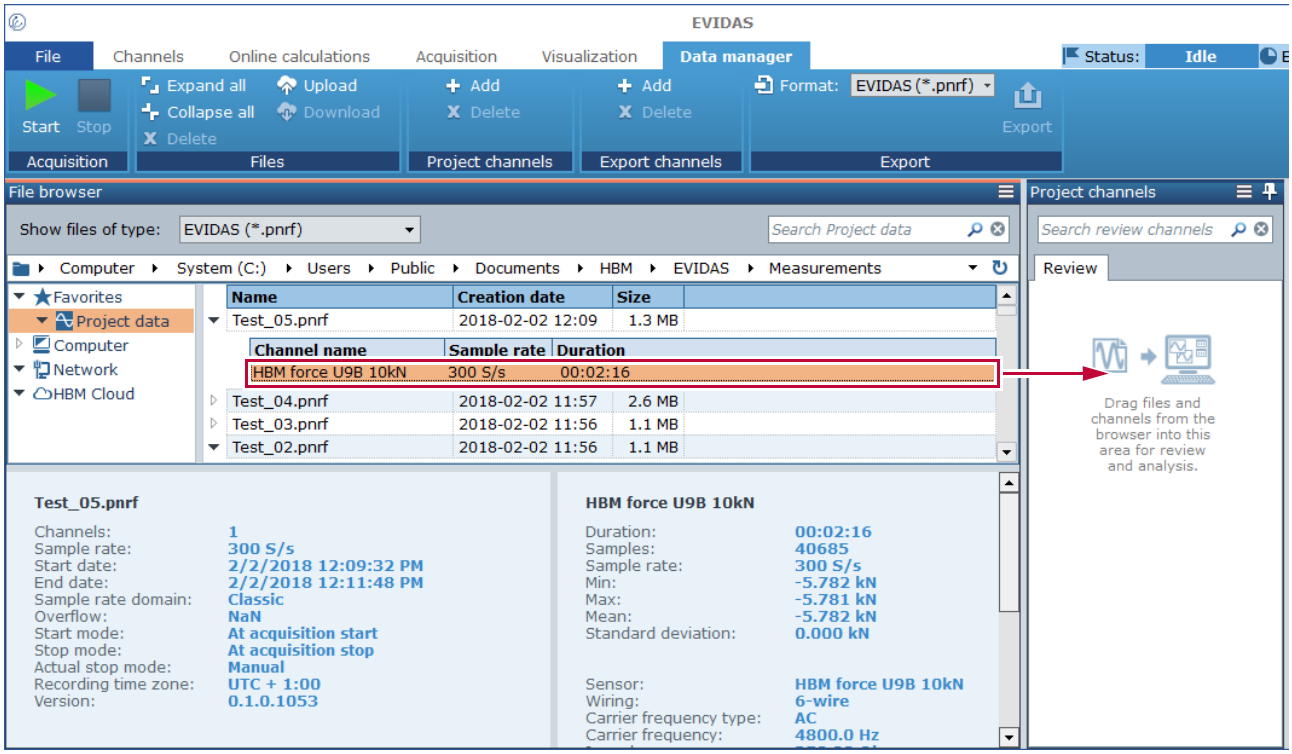
Prerequisites

- Data acquisition device and sensors are connected.
- EVIDAS is started.

To compare a live signal with a previous test

1 Click the **Data manager** tab.

2 In the **File browser**, drag the channel from the previous test to the **Project channels** panel.



✓ The channel is available as review channel.

3 Click the **Visualization** tab.

4 On the **Project channels** panel, click the **Review** tab.

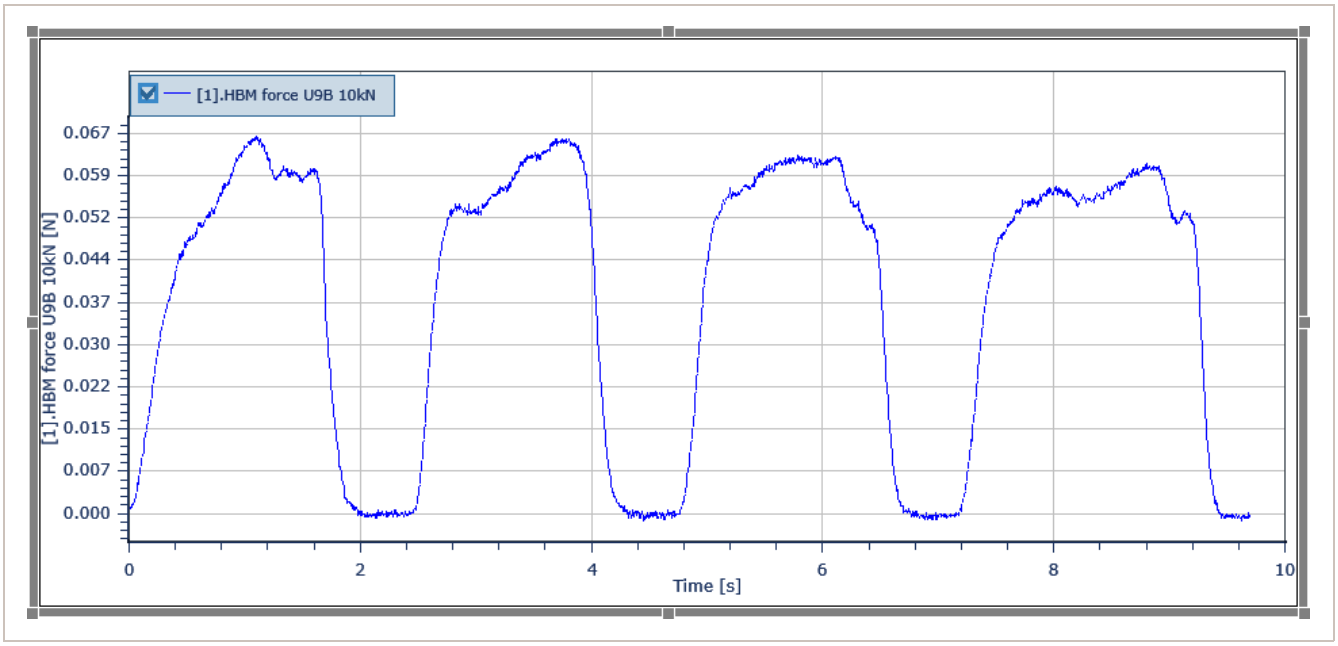
✓ The review channel you selected is displayed.

5 Click **y(t)**.

✓ An empty chart is displayed.

6 Drag the review channel to the y(t) chart.

✓ The review channel is displayed in overview mode.

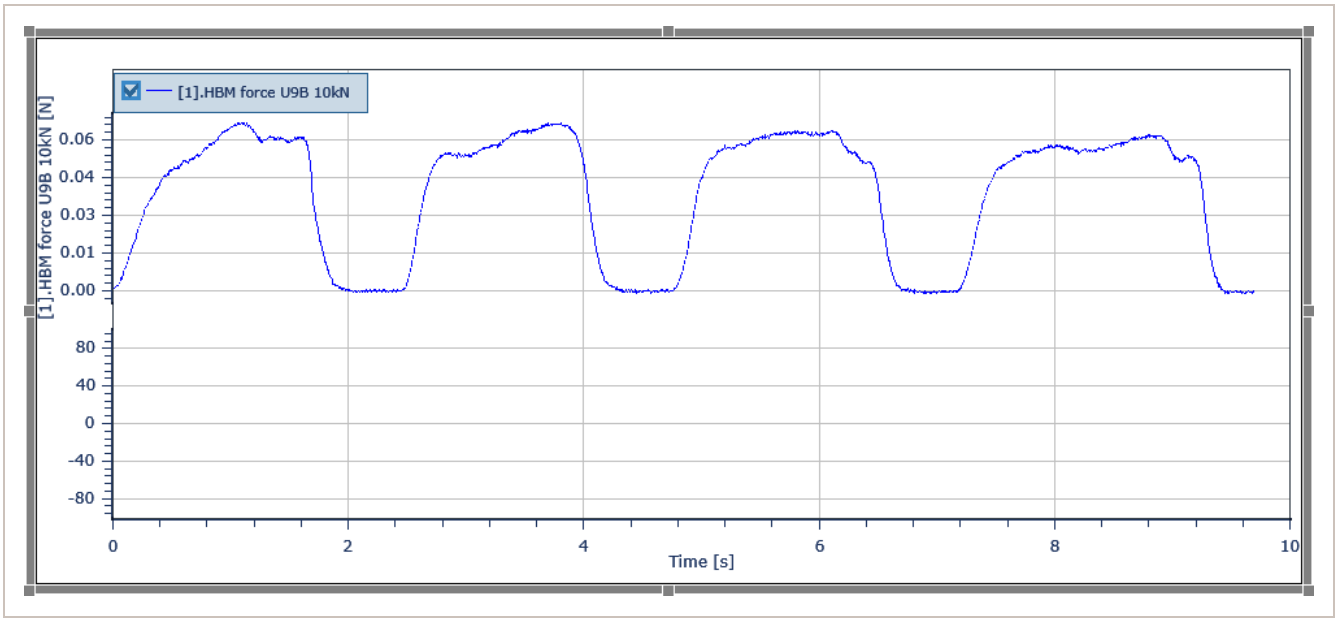


7 Click the y(t) chart.

✓ The contextual tab set is displayed.

8 Click the **Design** tab and select **Y-axes** = 2.

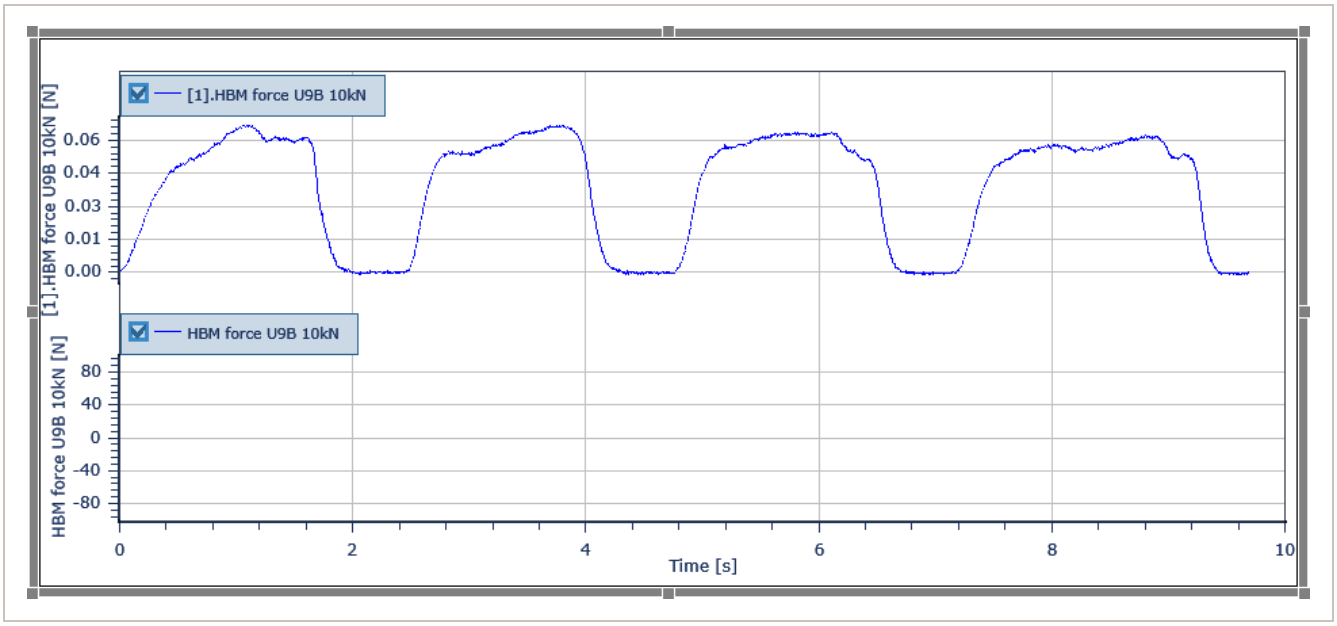
✓ A second coordinate system is displayed.



9 On the **Project channels** panel, click the **Live** tab and drag the live project channel to the second coordinate system.

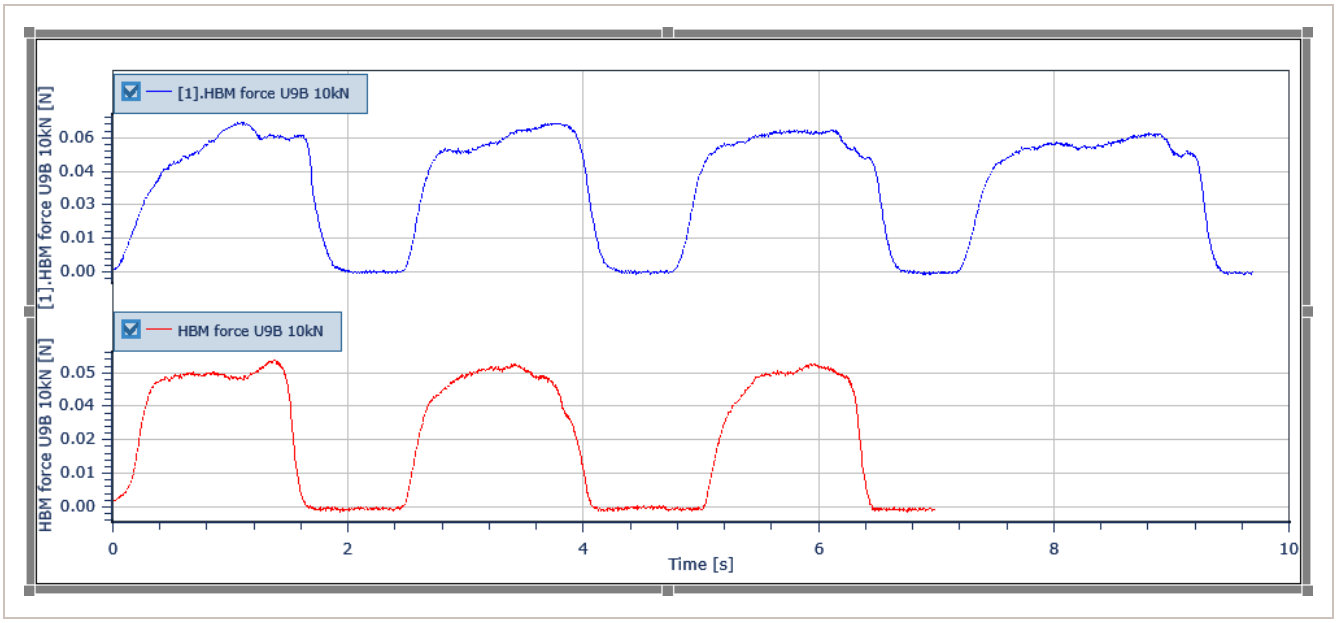
✓ The legend of the channel is displayed.

ⓘ Alternatively to steps 7 to 9, drag the live project channel to the very left side of the y-axis until the pointer becomes a , and then release the pointer. A second coordinate system is displayed and the live project channel is assigned to it.

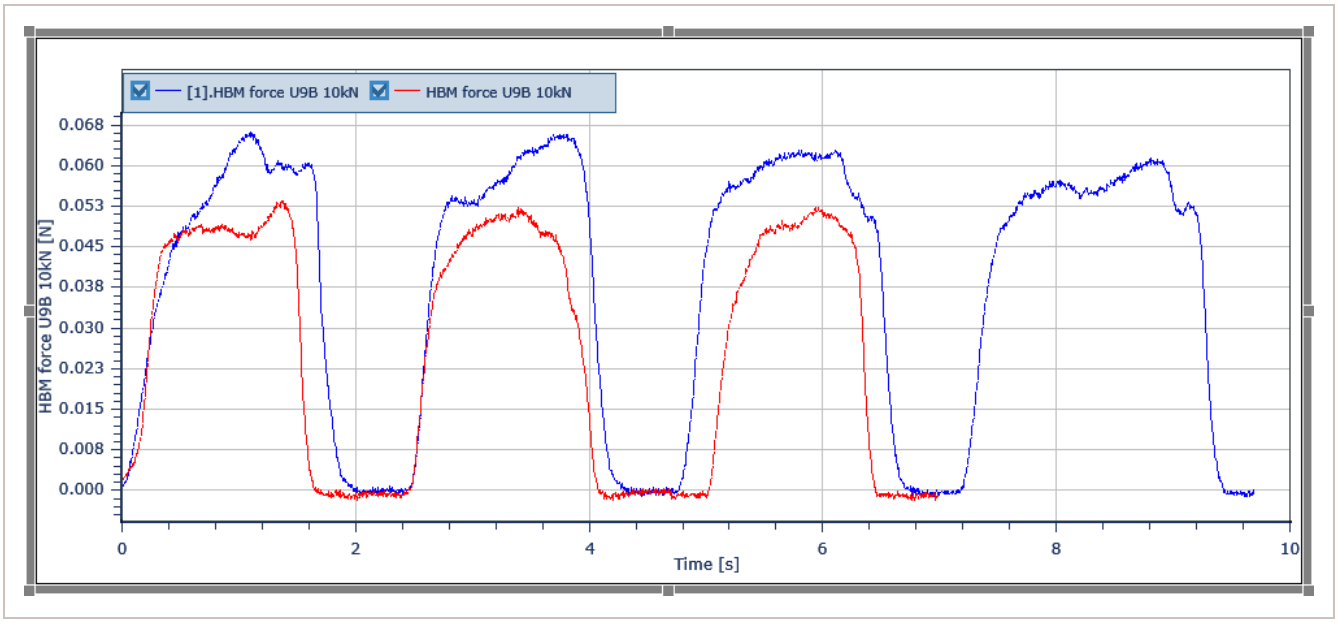


10 Click **Start**.

✓ The live project channel is displayed below the reference channel.



ⓘ To display the traces in the same coordinate system, delete the trace in the second coordinate system (Deleting a trace, 192), drag the live project channel to the first coordinate system, and on the the **Design** tab select **Y-axes** = 1.



8.8 Merging channels from different files

On the **Data manager** tab, you can use the **Export** command to merge channels from different files into one file.

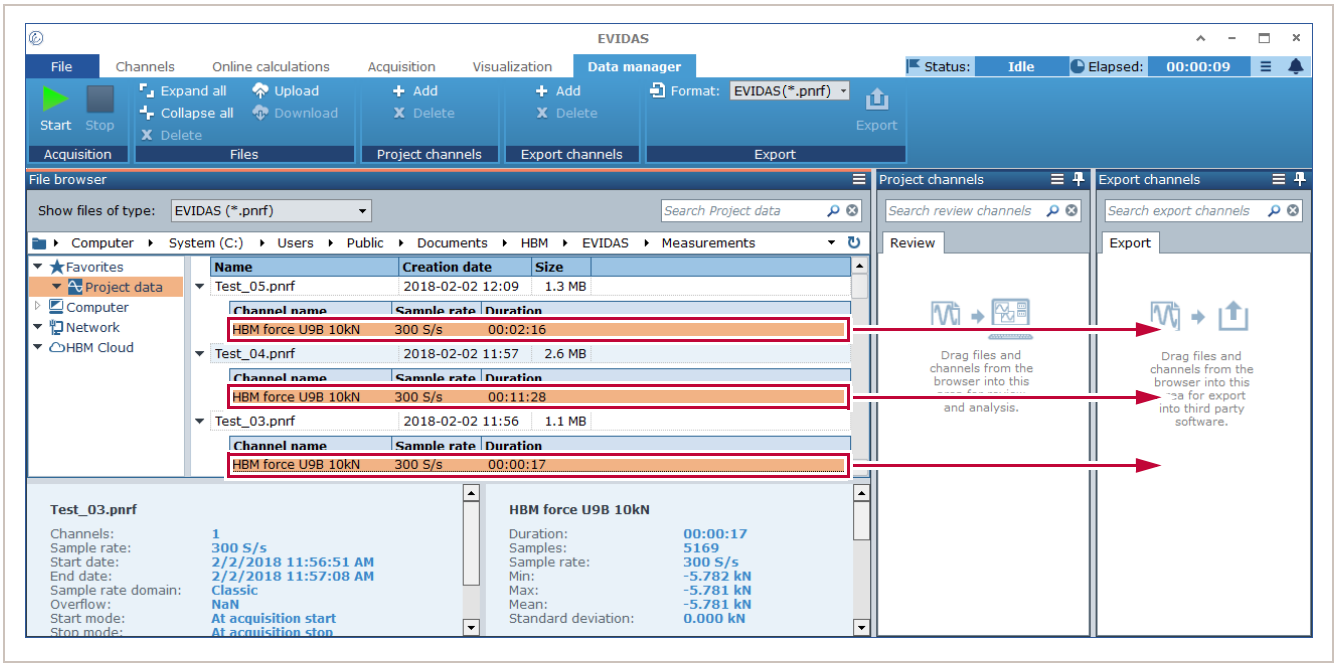
Prerequisites

- Data acquisition device and sensors are connected.
- EVIDAS is started.
- Several tests have been recorded.

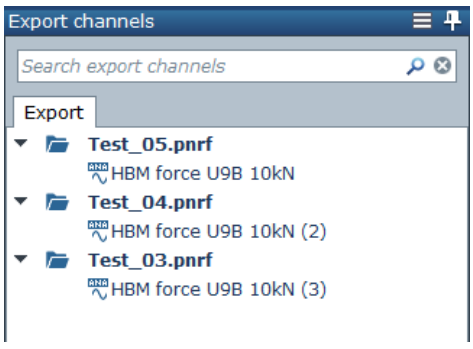
To merge channels from different files

1 Click the **Data manager** tab.

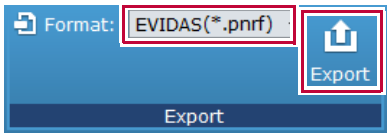
2 In the **File browser**, drag the channels you want to merge to the **Export channels** panel.



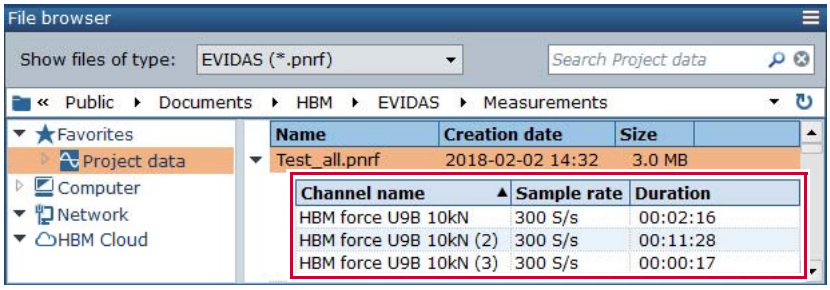
✓ The channels are selected for export.



3 In the **Export** group, select **EVIDAS (*.pnrf)** as export format, and then click **Export** and enter a filename.



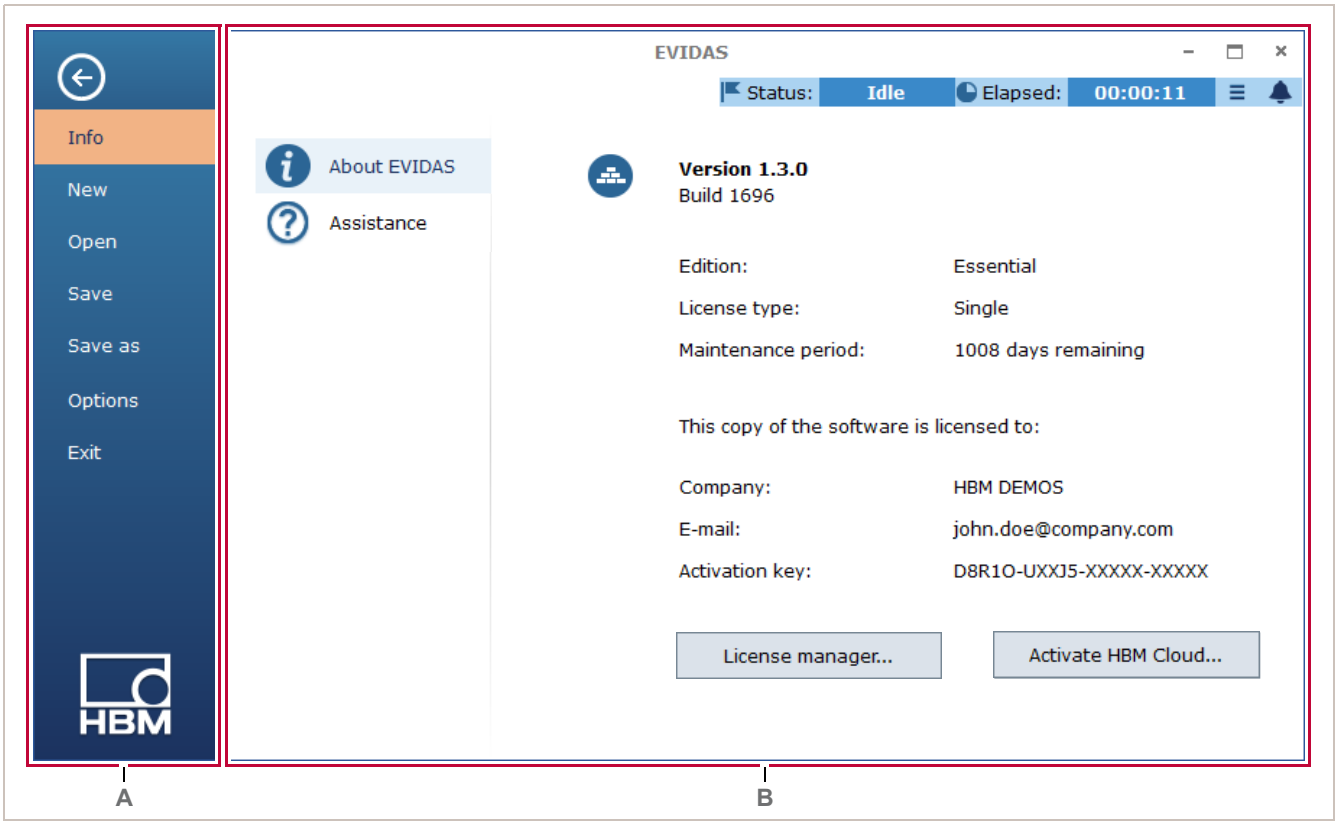
✓ The export file is written. It contains the channels from the **Export channels** panel.



9 File

On the **File** menu and its panels, you can do the following:

- Activate your EVIDAS license (Info panel, 222).
- Activate the HBM cloud (Info panel, 222).
- Renew your EVIDAS license (Info panel, 222).
- Borrow a network license if you are registered for such (Info panel, 222).
- Get help on using EVIDAS (Info panel, 222).
- Administrate EVIDAS projects (File menu, 220).
- Select user options, e.g., the language of the EVIDAS user interface (Options panel, 224).

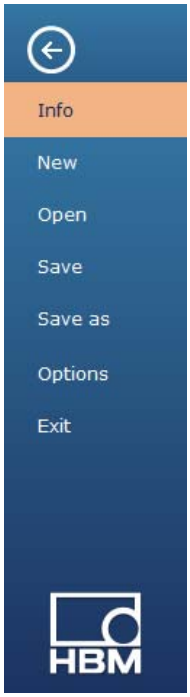


A File menu, 220

B Info panel, 222

9.1 File menu

To display, click the **File** tab.

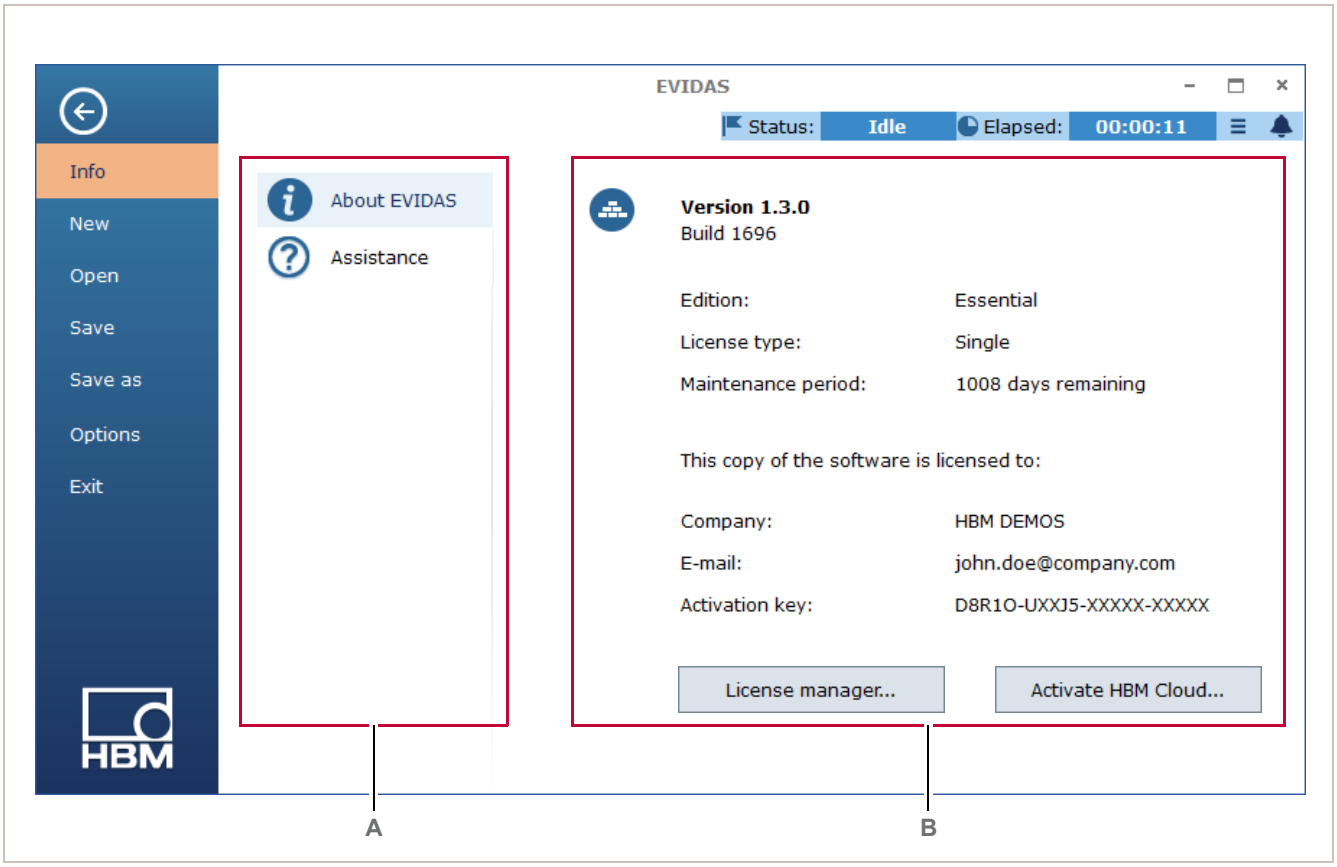


Options on the File menu

Option	Description
	To return to the ribbon (Ribbon, 39).
Info	To activate your EVIDAS license, to activate the HBM cloud, to borrow a network license if you are registered for such, to renew your EVIDAS license, and to get assistance on using EVIDAS (Info panel, 222). The release number and license information are displayed.
New	To start a new project (*.evidas) (Starting a new project, 58). You must save the current project before starting a new one. Otherwise, the settings of the current project are lost, e.g., the project channels, recording options, and visualization objects you defined.
Open	To open an existing project (*.evidas) (Opening a project, 71). You can select a recent project file or navigate to a project file. The project channels and settings are loaded from the project file. If EVIDAS detects that a data acquisition device required for this project is not connected to the system, a hardware mapper is displayed (Hardware mapper, 72). The default folder for project files is C:\Users\Public\Documents\HBM\EVIDAS\Projects.
Save	To save the current project (Saving a project, 70).
Save as	To save the current project under a different name.
Options	To select user options, e.g., the language of the EVIDAS user interface (Options panel, 224).
Exit	To exit EVIDAS.

9.2 Info panel

To display, click the **File** tab.



A Info groups

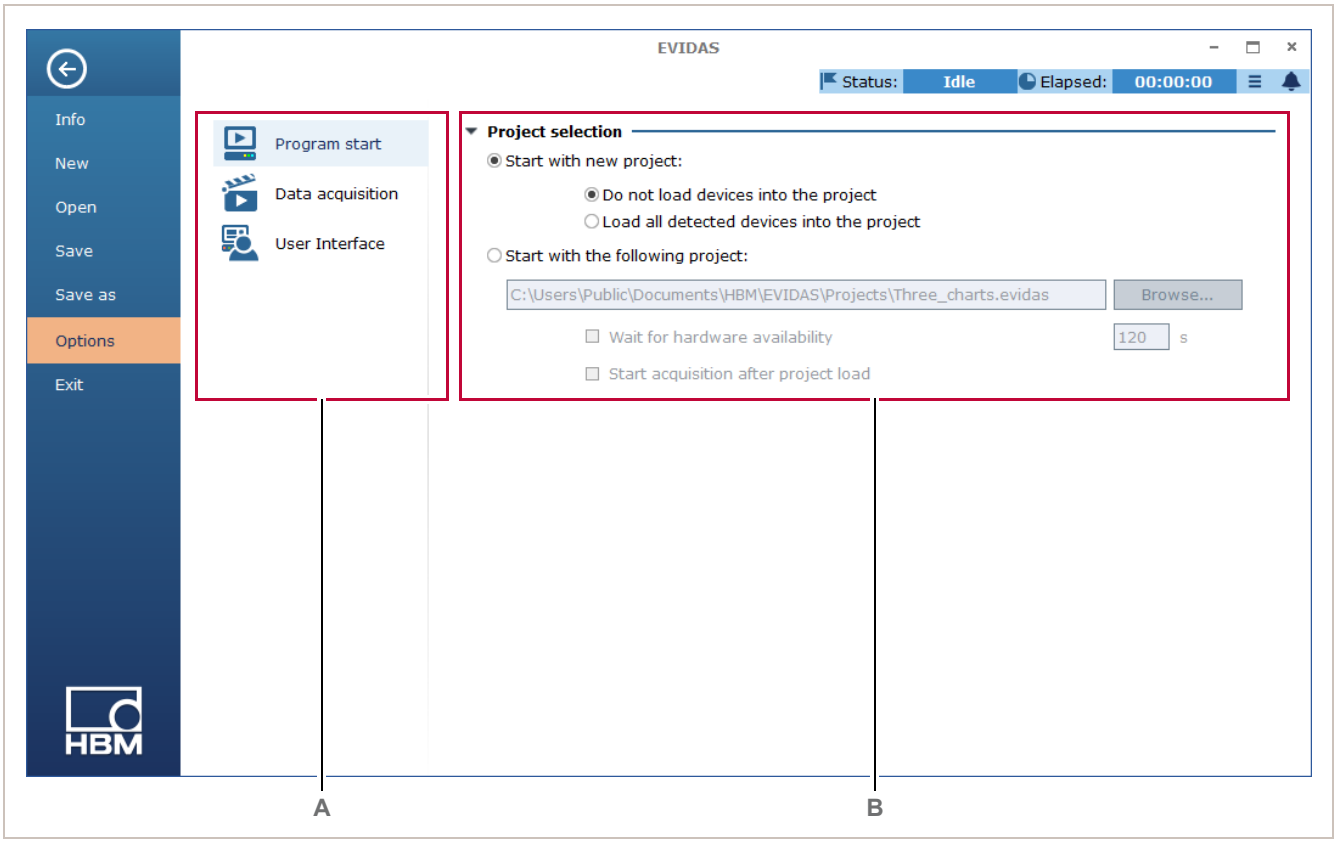
B Content of info groups

Infos and buttons on the Info panel

Info group	Info section / Button	Description
About EVIDAS	**Release and licence information**	The release number and license information are displayed. The license type defines which product features are enabled: **Viewer** - You can use EVIDAS as a free viewer for data files. **Trial** - You can test a full version of EVIDAS for 30 days. **Single** - You have installed and activated a full EVIDAS single license. **Network** - You are registered for using an EVIDAS network license.
	Activate software ...	This button is displayed if your license type is **Viewer** or **Trial**. Click this button to display the **EVIDAS activation** dialog box in order to activate an EVIDAS single or network license: Activating EVIDAS, 12
	License manager...	This button is displayed of your license type is **Single** or **Network**. Click this button to display the **License manager** dialog box in order to perform one of the following tasks: Renewing an EVIDAS license, 34 Borrowing an EVIDAS network license, 37
	Activate HBM Cloud...	This button is enabled if you have activated your license. Click this button to activate the HBM cloud: Activating the HBM cloud, 23
Assistance	Open online help	Click this button to display the EVIDAS online help. The online help contains a full description of EVIDAS: Online help, 28
	Open printable help (PDF)	Click this button to open a PDF help file. The PDF help file contains a full description of EVIDAS.
	Go to support web site	Click this button to contact the HBM support team.

9.3 Options panel

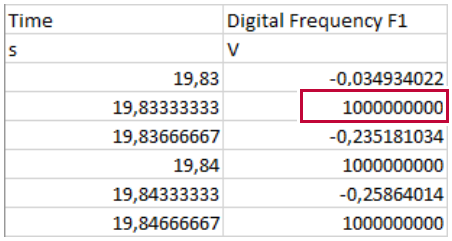
To display, click the **File** tab, and then click **Options**.



A Groups of options

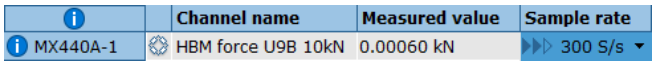
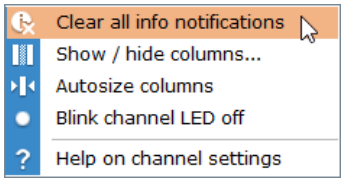
B Options

Options for configuring EVIDAS

Group of options	Option	Description
Program start	**Project selection**	To define what EVIDAS is supposed to do when being started. You have the following options: 1. EVIDAS starts a new project from scratch. 2. EVIDAS starts a new project and loads all channels of all detected data acquisition devices as project channels. 3. EVIDAS loads a project file, (optionally) waits a specified time for hardware availability, and (optionally) starts data acquisition.
Data acquisition	**Data acquisition stopped**	To add the project channels to the list of review channels after completing data acquisition.
	Overflow replacement	An overflow value is a value that cannot be measured because it is outside the measuring range of the data acquisition device. A signal in overflow is handled as follows: 1. On the data acquisition device, the channel LED turns red. 2. On the **Project channels** panel, the overflow icon is displayed (Signal status icons, 109) and in the **Measured value** column, the message text **Overflow** is displayed. 3. In the EVIDAS data file, instead of physical values, the user-defined **Overflow replacement** value is recorded.  Make sure the **Overflow replacement** value is large enough to not interfere with the measuring range of the data acquisition device. The **Overflow replacement** value helps you to identify when a signal was in overflow, e.g., after exporting data into a different format. 4. On the **Visualization** panel, the message text **Overflow** is displayed on the digital meter and in the **Actual** column of the data table.
	Channel configuration	To define whether the sample rate and filter are configured automatically when adding a new project channel (Configuring the sample rate and filter of project channels automatically, 100).

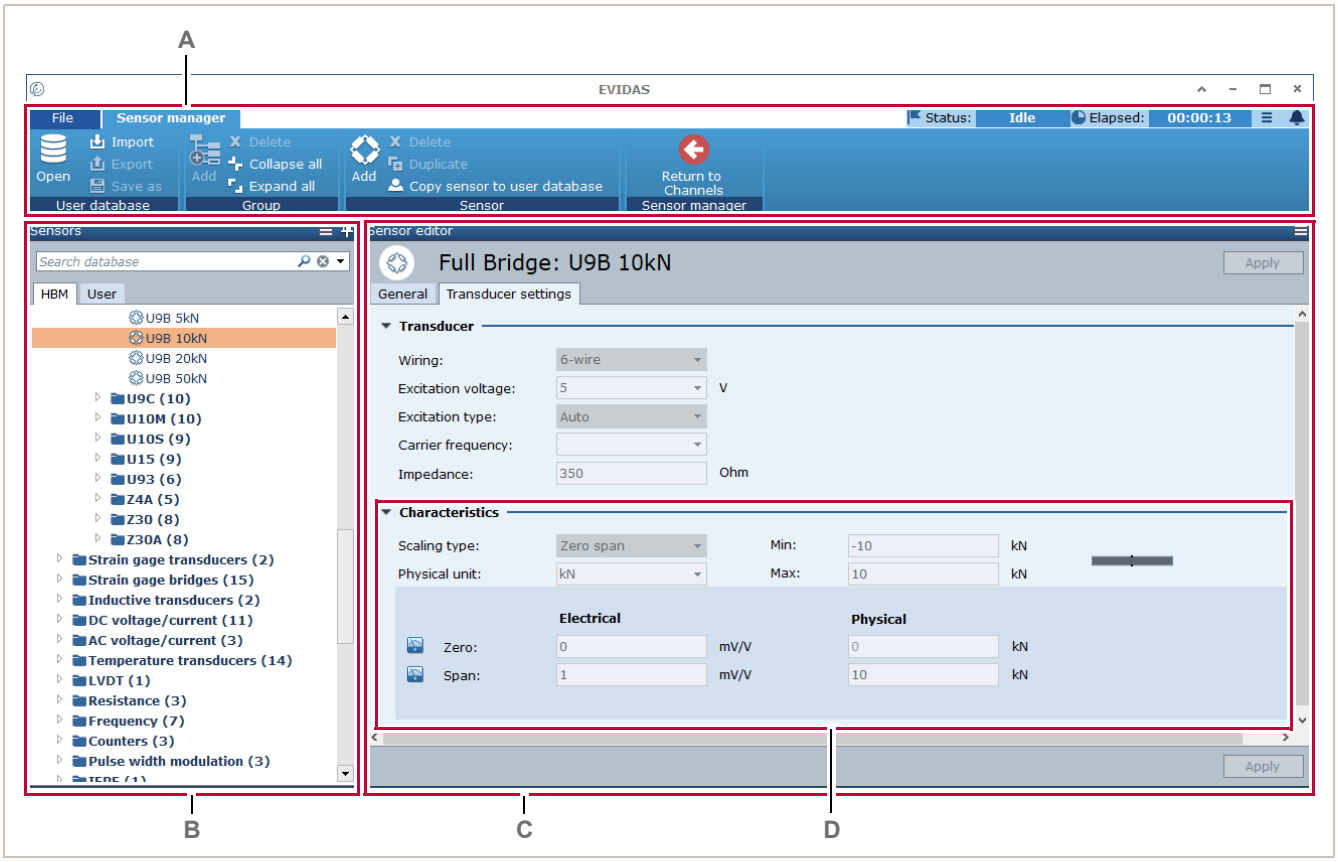
Time	Digital Frequency F1
s	V
19,83	-0,034934022
19,83333333	1000000000
19,83666667	-0,235181034
19,84	1000000000
19,84333333	-0,25864014
19,84666667	1000000000

Options for configuring EVIDAS

Group of options	Option	Description
User interface	**Application language**	To define the language of the user interface. You must restart the application before this change takes effect.
	Reset options	A. To redisplay hidden dialog boxes. B. To restore the factory settings of the software.
	Error handling	To alert you that a project channel setting has been changed either by a device or by EVIDAS. On the **Project channels** panel, the info icon is displayed and the setting is highlighted.  Point to for a report. To acknowledge a changed setting and to delete , on the burger menu of the **Project channels** panel, click **Clear all info notifications** (Burger menu, 108). 

10 Appendix A: Sensor manager

To display, click the **Channels** tab, and then click **Manage**.



A Sensor manager tab, 228↗

B Sensors panel, 230↗

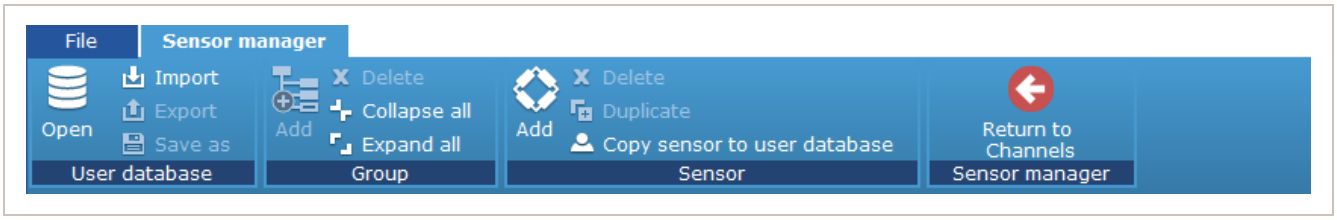
C Sensor editor, 234↗

D Scaling type, 238↗

On the **Sensor manager** tab and its panels, you can copy HBM sensors to the user-defined sensor database and edit their settings there.

10.1 Sensor manager tab

To display, click the **Channels** tab, and then click **Manage**.



Most commands on the **Sensor manager** tab apply to the user-defined sensor database.

Sensor manager commands

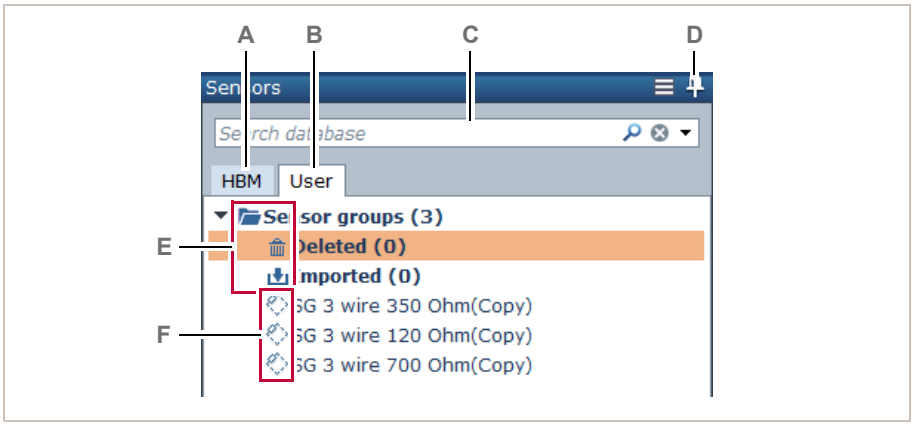
Group	Command	Description
User database	**Open**	To open a user-defined sensor database.
	Import	To import sensors or a CAN database (Importing a CAN database, 245↗). The settings are imported into the **Imported** sensor group of the user-defined sensor database.
	Export	To export sensors from the user-defined sensor database. On the **Sensors** panel, hold down Ctrl while you click the sensors you want to export, and then click **Export**.
	Save As	To save a user-defined sensor database.
Group	**Add**	To add a new sensor group to the user-defined sensor database (Tree view, 49↗). On the **Sensors** panel, click the sensor group into which you want to insert the new sensor group, and then click **Add**.
	Delete	To delete a sensor group. On the **Sensors** panel, click the sensor group you want to delete, and then click **Delete**. The sensor group and its sensors are moved to the **Deleted** sensor group. To recover a sensor group, click it in the **Deleted** sensor group and drag it to **Sensor groups**.
	Collapse all	To collapse the tree view on the **Sensors** panel (Tree view, 49↗).
	Expand all	To expand all sensor groups on the **Sensors** panel (Tree view, 49↗).

Sensor manager commands

Group	Command	Description
Sensor	**Add**	To add a new sensor to the user-defined database. On the **Sensors** panel, click the user-defined sensor group in which you want to insert the new sensor, click **Add**, and then click a sensor type and sensor. The new sensor contains only basic settings and must be configured.
	Delete	To delete a user-defined sensor. On the **Sensors** panel, hold down Ctrl while you click the sensors you want to delete, and then click **Delete**. The sensors are moved to the **Deleted** sensor group. To recover a sensor, click it in the **Deleted** sensor group and drag it to a sensor group.
	Duplicate	To copy a sensor. On the **Sensors** panel, click the user-defined sensor that you want to copy, and then click **Duplicate**. The sensor settings are copied to the new sensor.
	Copy sensor to user database	To copy sensors from the HBM sensor database to the user-defined sensor database. On the **Sensors** panel, right-click an HBM sensor, and then click **Copy sensor to user database**. The sensor is copied to the **Imported** sensor group of the user-defined sensor database.
Sensor manager	**Return to Channels**	To return to the **Channels** tab.

10.2 Sensors panel

To display, click the **Channels** tab, and then click **Manage**.



A HBM sensor database, 230↗
B User-defined sensor database, 230↗
C Search box, 230↗
D Burger menu, 231↗
E Sensor group icons, 231↗
F Sensor icons, 231↗

HBM sensor database

The HBM sensor database contains all available HBM sensors with their default settings. The database is updated with each release of the software.

You cannot edit any settings in the HBM sensor database.

To edit the settings of an HBM sensor, e.g., to match its settings to the specifications on the sensor data sheet, you must copy the HBM sensor to the user-defined sensor database and edit the settings there (Copying an HBM sensor to the user-defined sensor database, 232↗).

If you configure a project channel with an HBM sensor (Configure with HBM sensor database, 62↗) without matching the sensor settings to the sensor data sheet, the measured values may not be accurate.

User-defined sensor database

The user-defined sensor database can contain copies of HBM sensors and imported CAN databases (Importing a CAN database, 245↗).

You can edit the user-defined sensors in the **Sensor editor** (Sensor editor, 234↗).

Search box

The search box enables you to search the content of the sensor database (Search box, 47↗).

Burger menu

Command	Description
Help on sensors	To access context-sensitive help.

Sensor group icons

Sensor group (collapsed).

Sensor group (expanded).

Group of deleted sensors.

Group of imported sensors.

Sensor icons

The sensor icons represent sensor types, e.g., for a full bridge sensor.

Point to the icon to display information on the sensor type.

10.3 Copying an HBM sensor to the user-defined sensor database

To use an HBM sensor, you must copy it to the user-defined sensor database and adjust its settings there.

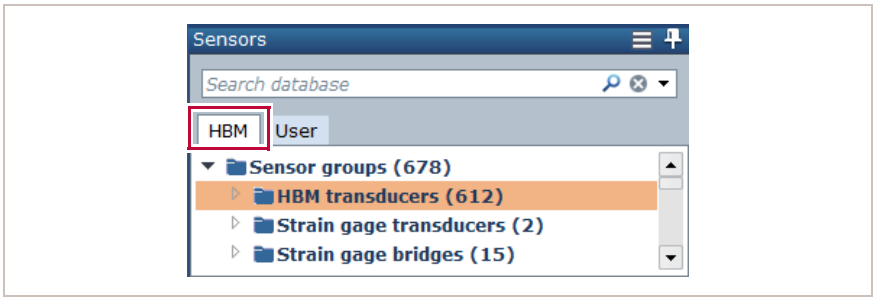
To copy an HBM sensor to the user-defined sensor database

1 Click the **Channels** tab.

2 Click **Manage**.

✓ The **Sensor manager** tab is displayed.

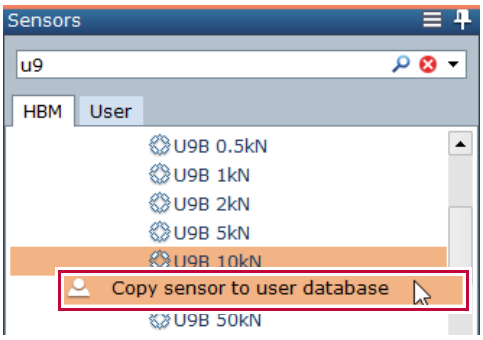
3 On the **Sensors** panel, click **HBM**.



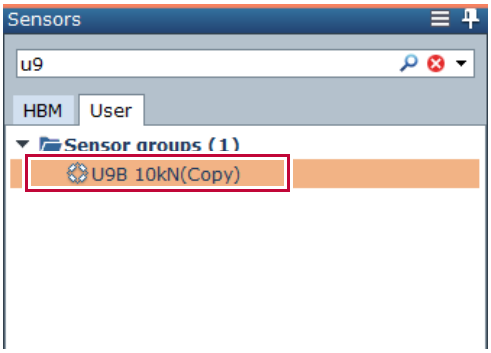
4 In the **Sensors** search box, enter the required sensor type.

✓ The matching sensors are displayed.

5 Right-click the HBM sensor you need, and then click **Copy sensor to user database**.



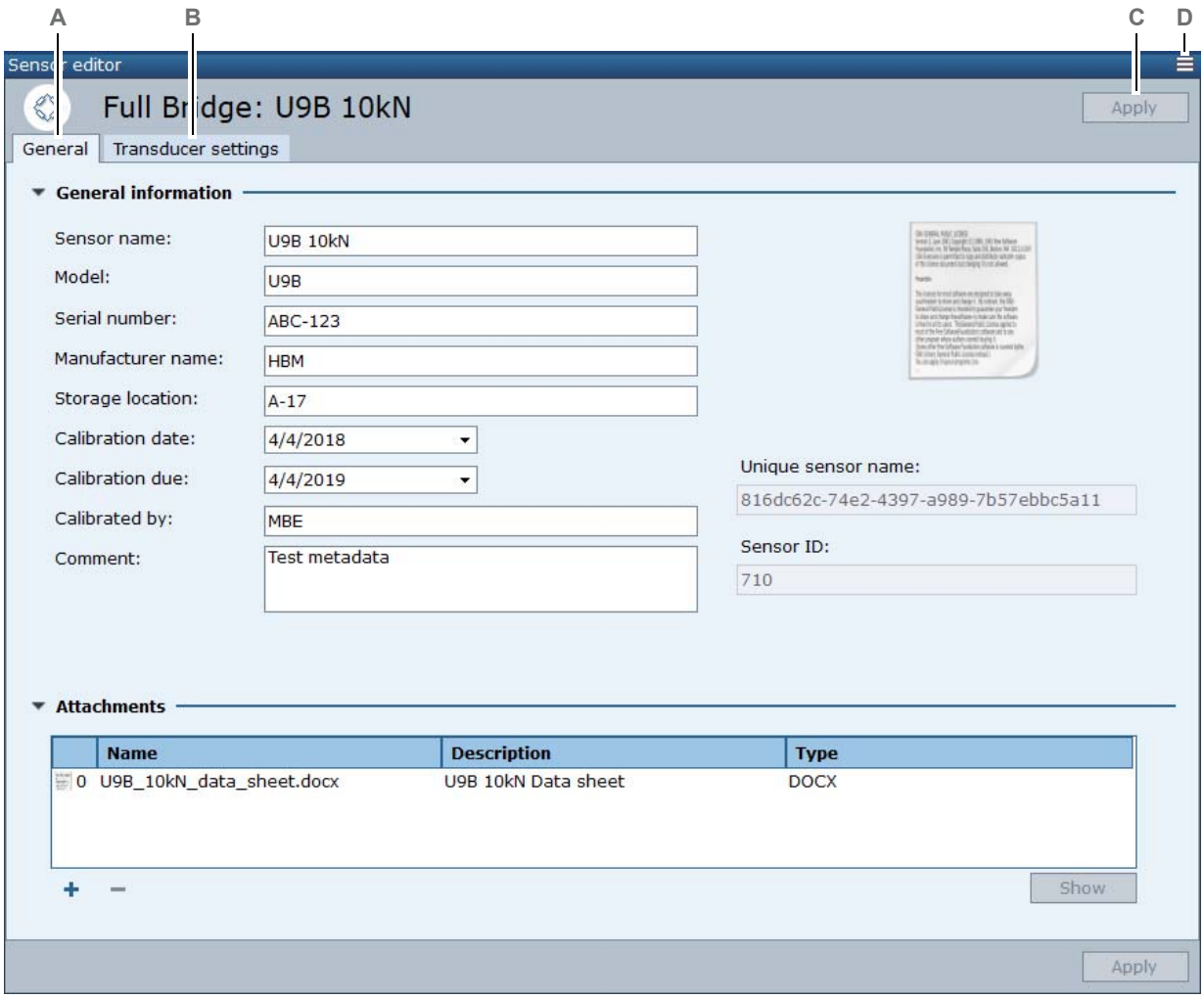
✓ The sensor is copied to the user-defined sensor database.



ⓘ You can edit the sensor settings in the **Sensor editor** (Sensor editor, 234↗).

10.4 Sensor editor

To display, click the **Channels** tab, and then click **Manage**.



A General tab, 236↗

B Transducer settings, 237↗

C Apply button, 234↗

D Burger menu, 235↗

To edit the settings of an HBM sensor, you must copy it to the user-defined sensor database and edit the settings there (Copying an HBM sensor to the user-defined sensor database, 232↗).

Apply button

Click the **Apply** button to save any changes.

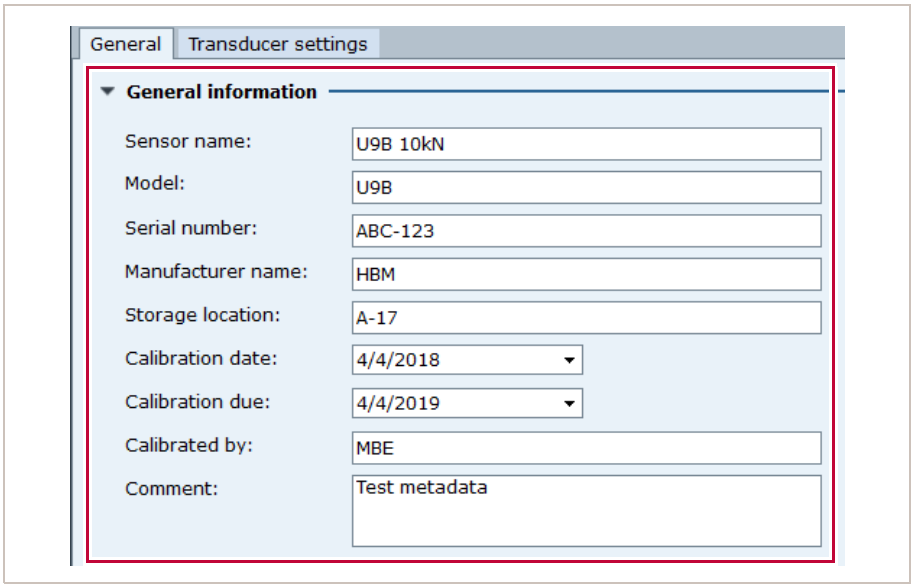
Burger menu

Command	Description
Help on sensor editor	To access context-sensitive help.

10.5 General tab

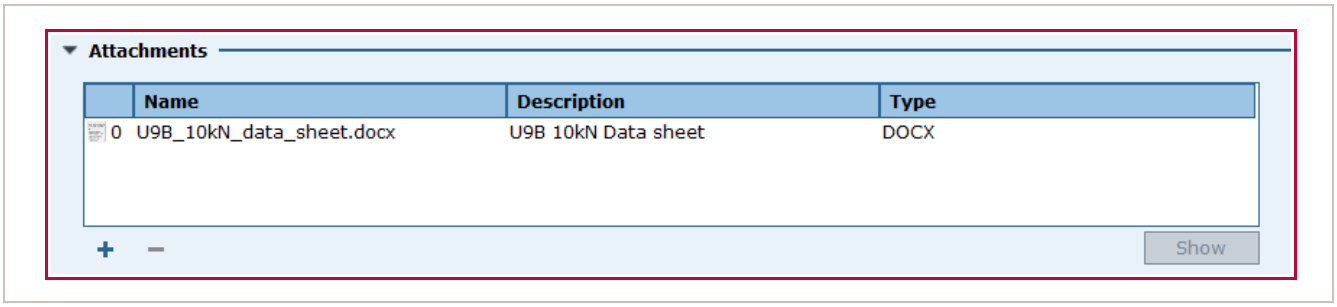
To display, click the **Channels** tab, click **Manage**, and then click **General**.

General information



Type the metadata of the sensor into the respective text boxes.

Attachments



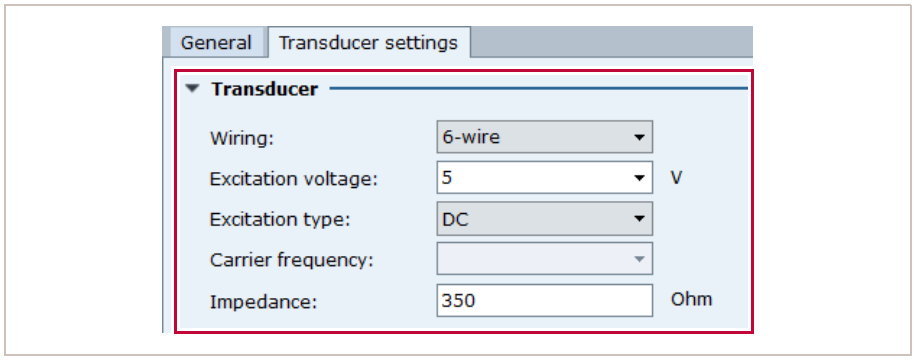
You can attach a file to the sensor information, e.g., a scan of its data sheet.

- To add an attachment, click +, and then double-click a file in the **Explorer**.
- To delete an attachment, click the attachment, and then click −.
- To display an attachment, click the attachment, and then click the **Show** button.

10.6 Transducer settings

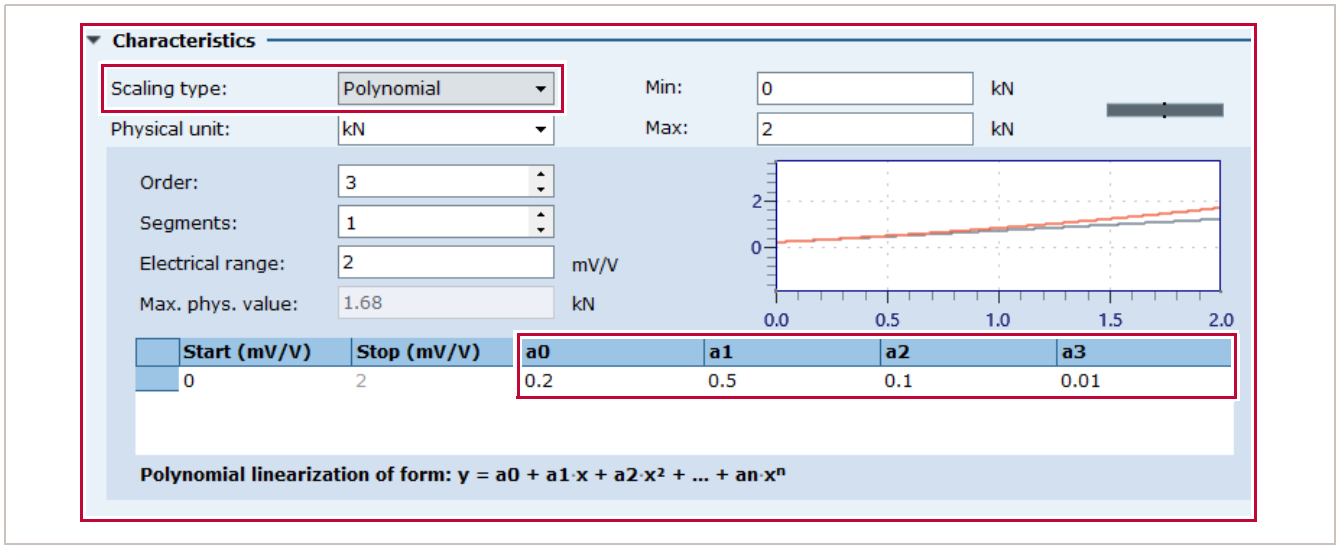
To display, click the **Channels** tab, click **Manage**, and then click **Transducer settings**.

Transducer



Enter the sensor settings into the respective text boxes. The sensor settings vary with each type of sensor.

Characteristics



A characteristic is a function that calibrates the measured values. It defines how a valid range of electrical values, e.g., from 0 to 10 mV/V, is mapped to the corresponding physical values, e.g., from 0 to 50 Newton [N].

Select a scaling type (Scaling type, 238), e.g., **Polynomial**, and enter the parameters that define the characteristic, e.g., the polynomial coefficients.

10.7 Scaling type

The scaling type determines the calibration characteristic and defines how a valid range of electrical values, e.g., from 0 to 10 mV/V, is mapped to the corresponding physical values, e.g., from 0 to 50 Newton [N].

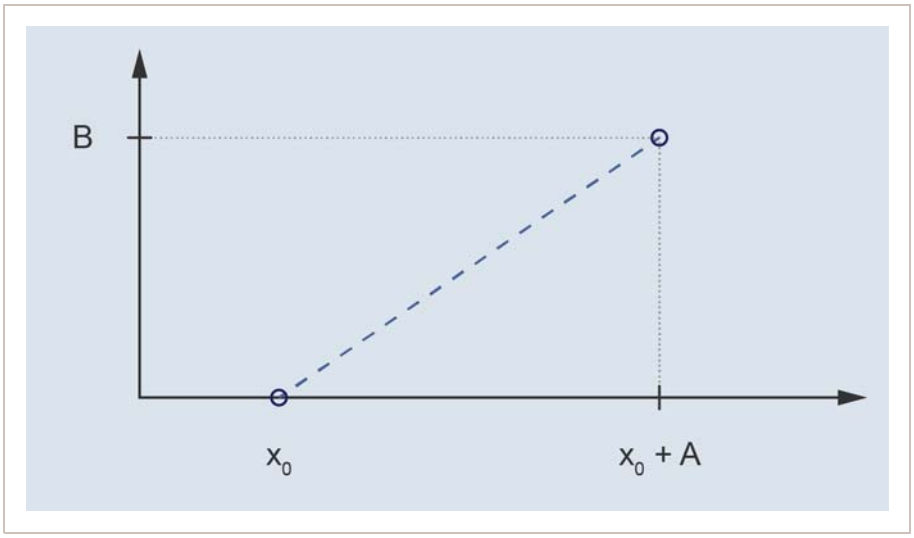
The following scaling types are available:

- Zero span, 238
- Two-point, 239
- Table, 239
- Polynomial, 240
- Gauge, 240
- Internal, 240
- Off, 240

In the following illustrations, the x-values represent electrical values and the y-values represent the corresponding physical values.

Zero span

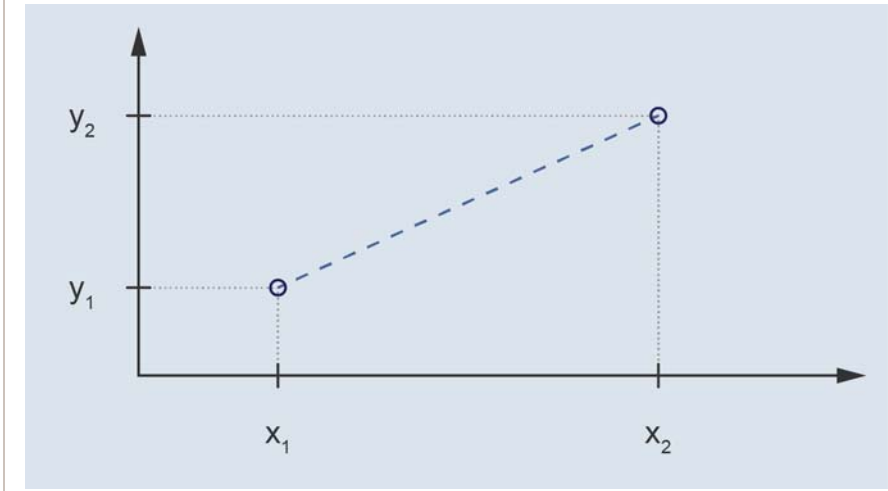
This scaling type defines a linear characteristic represented by a straight line through a calibration point (x_0|0) and a second point defined by a span (A|B) added to (x_0|0).



Zero span is a special case of two-point scaling (Two-point, 239).

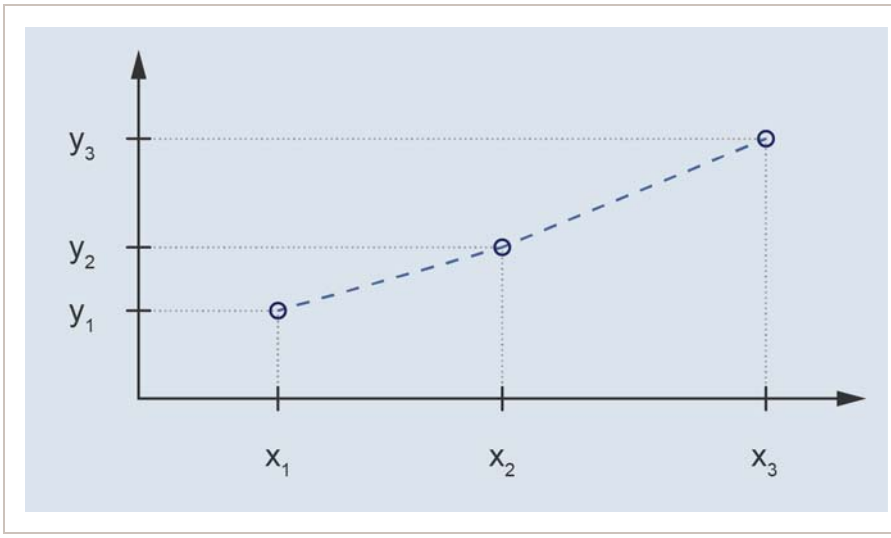
Two-point

This scaling type defines a linear characteristic represented by a straight line through two calibration points ($x_1|y_1$) and ($x_2|y_2$).



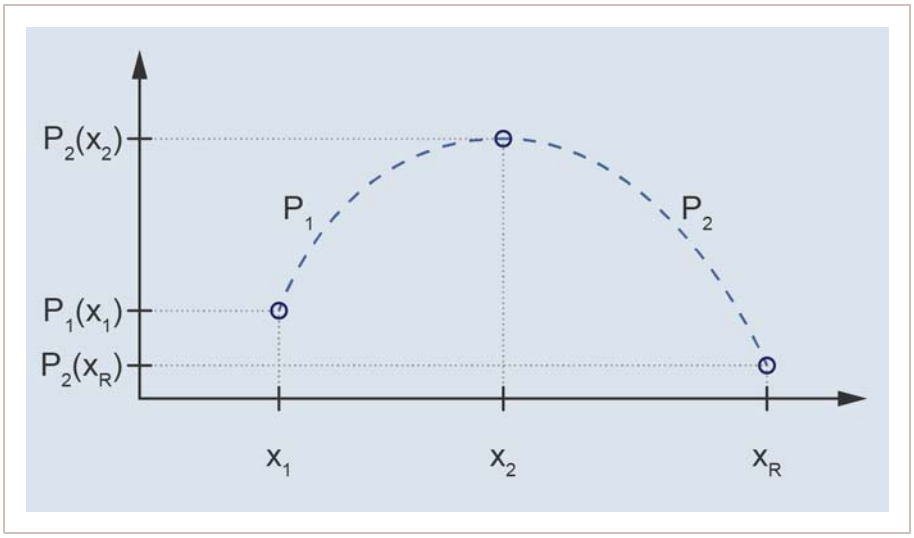
Table

This scaling type defines a linear characteristic represented by adjacent segments of straight lines between several calibration points ($x_1|y_1$), ($x_2|y_2$), ($x_3|y_3$), etc.



Polynomial

This scaling type defines a non-linear characteristic represented by adjacent polynomial segments P_1, P_2, etc. that start in $(x_1|P_1(x_1))$, $(x_2|P_2(x_2))$, etc. and go as far as the electrical range x_R.



Gauge

This scaling type is for strain gauges only and defines a linear characteristic by using the gauge factor, e.g., 2.1, and the bridge factor (1, 2, or 4) of the sensor. For the gauge factor, refer to the sensor data sheet. The bridge factor is determined by the number of resistors in the bridge circuit.

Internal

The calibration characteristic is defined in the data acquisition device.

Off

No characteristic is applied, i.e., the measured electrical values are taken 1:1 as the physical values.

11 Appendix B: CAN channels

CAN messages (**C**ontroller **A**rea **N**etwork messages) are transmitted and received via a serial CAN bus system, e.g., in a car to communicate the status of various components, such as the engine, the brakes, and the airbags.

A CAN message consists of several bytes organized in a data frame with a message ID, control bits, and a data field. The data field contains the CAN signal. The encoding of a CAN signal may use one or two bits or several bytes.

The classic CAN protocol allows data fields with up to 8 bytes. However, the increasing number of sensors and control systems, e.g., in cars, demanded a higher bandwidth, i.e., the option to send more data in the same frame at a higher speed.

The CAN FD (Flexible Data Rate) protocol was developed in response to this need and allows to increase the bit rate during the transmission of the data field by a factor of up to 8. CAN FD data fields can therefore contain up to 64 bytes and are still transmitted in the same time as the 8 bytes in classic CAN. The rest of the CAN message frame, however, uses the same bit rate as classic CAN.

If provided with a CAN database, EVIDAS can identify CAN messages, decode their signals, and feed their values into project channels. Each CAN project channel then represents one CAN signal.

To work with CAN channels, you must do the following:

- Connecting EVIDAS to a CAN bus, 242
- Defining the CAN bus settings of a connector, 243
- Importing a CAN database, 245
- Configuring CAN project channels, 247

Once you have defined the CAN bus settings and configured the CAN channels, you can record, visualize, and review CAN signals in the same way as any other signal.

However, not every data acquisition device is CAN-capable (CAN-capable data acquisition devices, 250).

For a list of CAN bus settings see: CAN bus settings, 251.

11.1 Connecting EVIDAS to a CAN bus

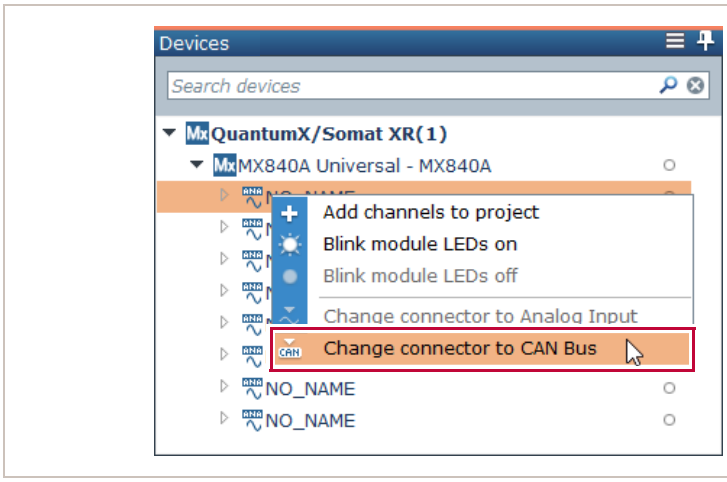
Use this procedure to switch the first connector of a QuantumX MX840 to CAN bus mode.

The connectors of the other CAN-capable data acquisition devices (CAN-capable data acquisition devices, 250) can be connected straight to a CAN bus.

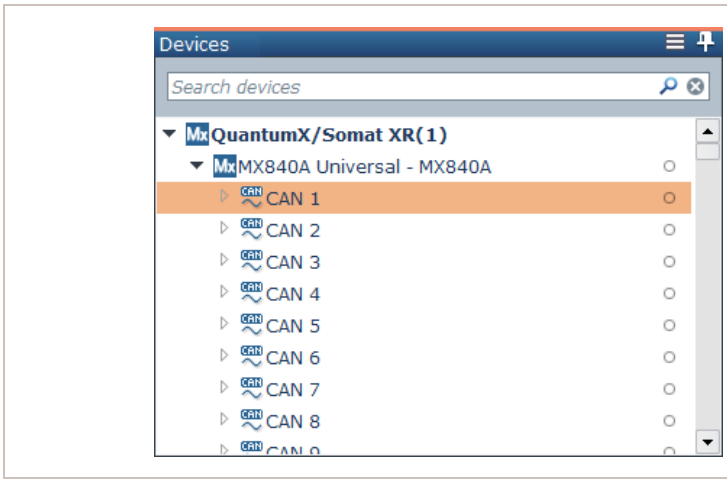
To switch the first connector of an MX840 to CAN bus mode

1 Click the **Channels** tab.

2 In **Devices**, right-click the first channel of the MX840 module, and then click **Change connector to CAN Bus**.



✓ The connector is switched to CAN bus mode.



ⓘ You have 128 CAN channels available.

11.2 Defining the CAN bus settings of a connector

You must define a separate set of CAN bus settings for each connector (CAN bus settings, 251).

Prerequisites

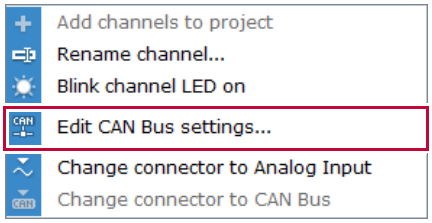
- CAN-capable data acquisition device (CAN-capable data acquisition devices, 250).

To define the CAN bus settings of a connector

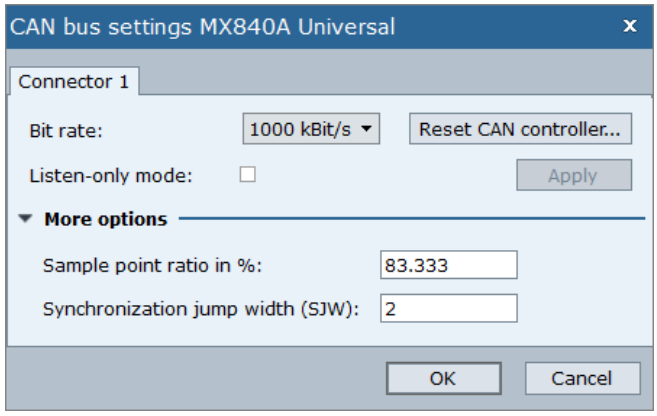
1 In **Devices**, right-click a CAN channel.

✓ The context menu is displayed.

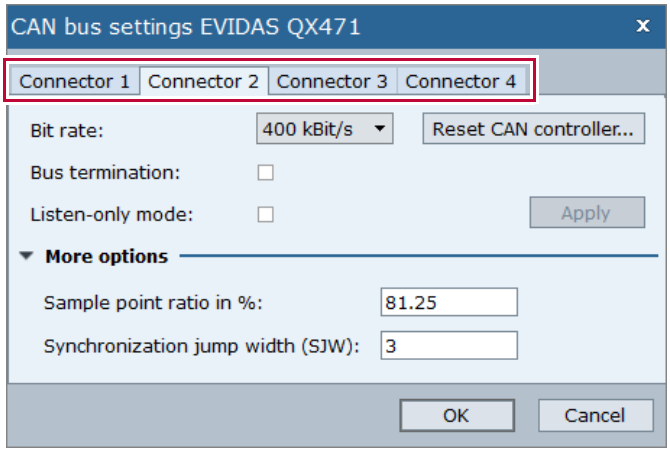
2 Click **Edit CAN bus settings**.



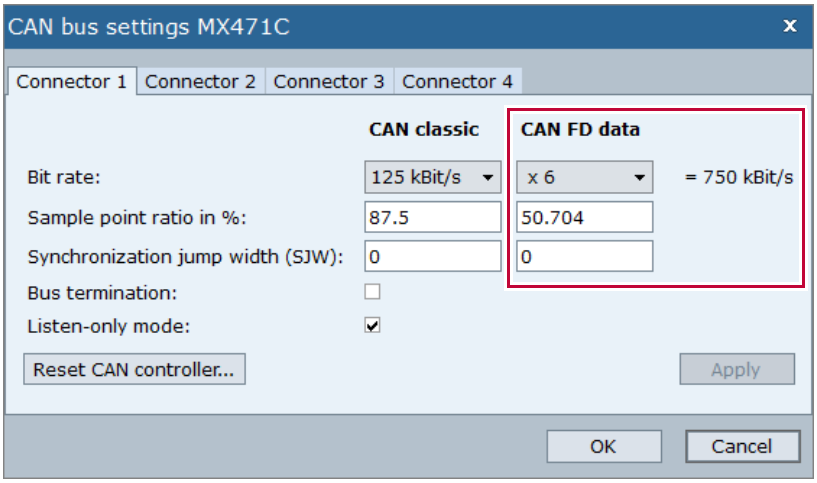
✓ The **CAN bus settings** dialog box is displayed.



ⓘ If your data acquisition device, e.g., a QuantumX MX471B, has several CAN-capable connectors a tab for each connector is displayed.



ⓘ If your data acquisition device, e.g., a QuantumX MX471C, uses CAN FD, some settings must be defined for both the classic CAN as well as for CAN FD.



3 For each connector that is connected to a CAN bus, enter the settings used by the respective CAN bus (CAN bus settings, 251).

ⓘ You cannot overrule the settings of the CAN bus controller. However, with the **Reset CAN controller** button, you can reset the CAN bus controller.

11.3 Importing a CAN database

A CAN database is a user-defined *.dbc file that describes how the CAN signals are encoded and packed into CAN messages.
You must import a CAN database into EVIDAS so that EVIDAS can decode the CAN signals and feed their values into project channels.

Prerequisites

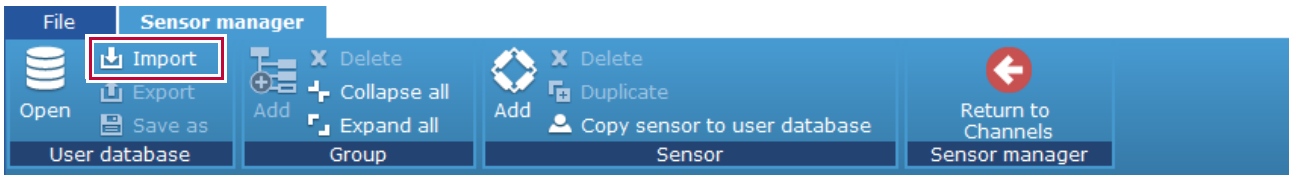
- CAN-capable module is connected (CAN-capable data acquisition devices, 250).
- EVIDAS is started.

To import a CAN database

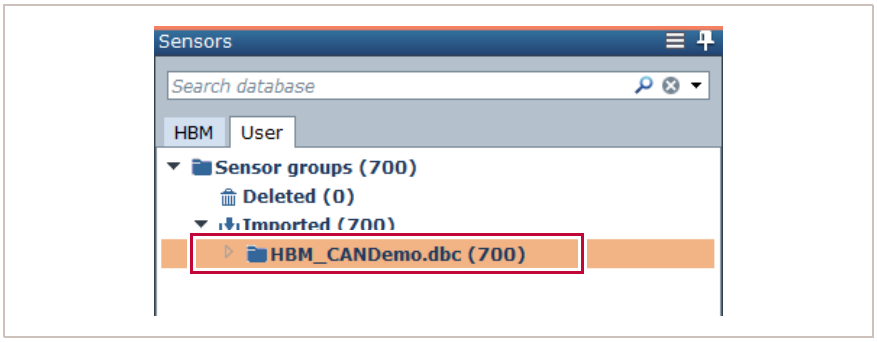
1 Click the **Channels** tab.

2 Click **Manage**.
✓ The **Sensor manager** tab is displayed.

3 Click **Import**.



4 Navigate to the location of the *.dbc file, click the file, and then click **Open**.
✓ The CAN database is imported as a user-defined sensor database.



5 Click **Expand all**.

✓ The CAN messages and their signals are displayed.

6 Click **Return to Channels**.

11.4 Configuring CAN project channels

To decode the CAN signals of a CAN message, you must configure the CAN project channels with the signal settings from the CAN database.

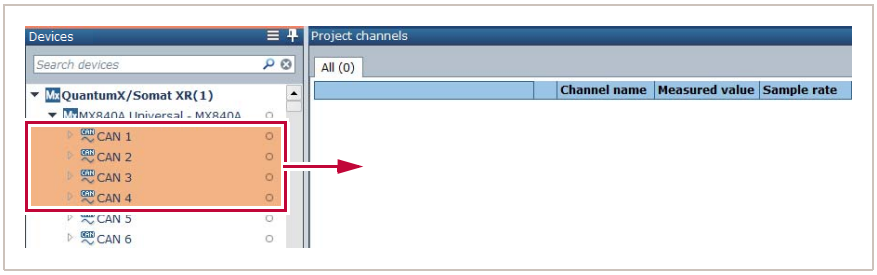
Prerequisites

- CAN-capable module is connected (CAN-capable data acquisition devices, 250).
- EVIDAS is started.
- CAN database is imported (Importing a CAN database, 245).

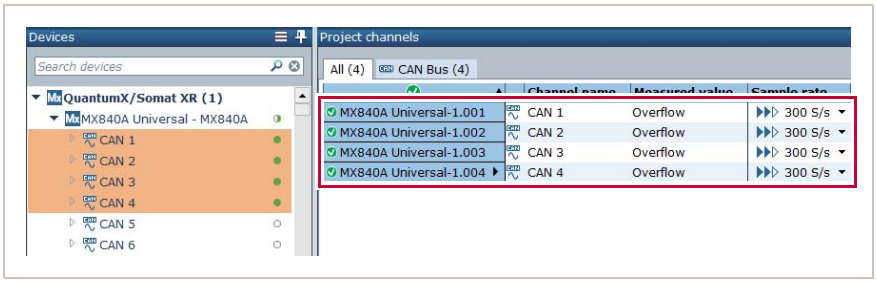
To configure CAN project channels

1 Click the **Channels** tab.

2 In **Devices**, click the CAN channels you need and drag them to **Project channels**.

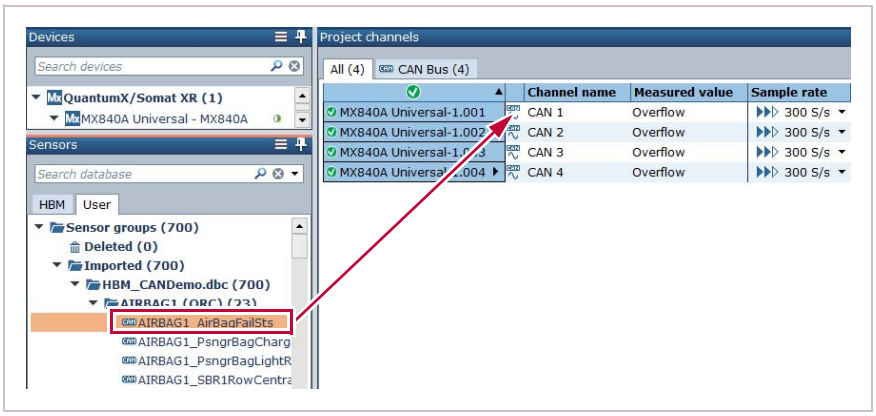


✓ The selected CAN channels are now project channels.



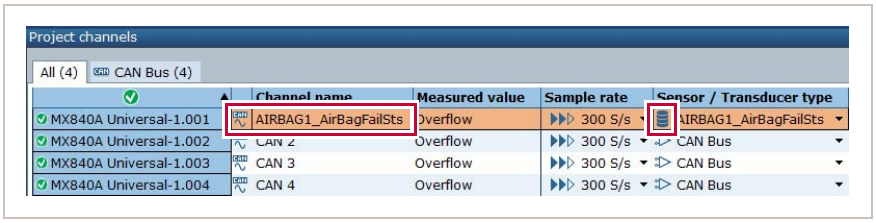
ⓘ The CAN **CAN Bus** tab in **Project channels** displays the default settings of the CAN project channels.
However, you must configure the CAN project channels with the signal settings from the CAN database to correctly decode the CAN signals.

3 In **Sensors**, click the CAN signal you need and drag it to the respective CAN project channel.

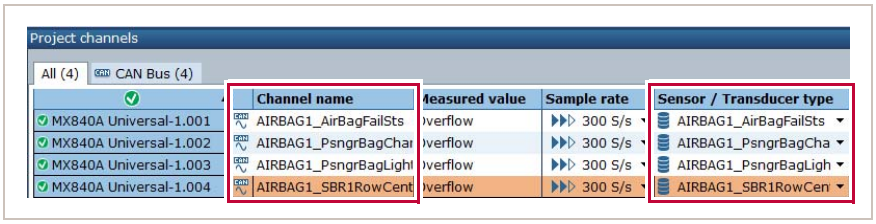


✓ The CAN project channel is configured with the signal settings from the CAN database.

ⓘ In the **Channel name** column, the CAN signal name is displayed, and in the **Sensor / Transducer type** column, indicates that the settings are taken from the CAN database.

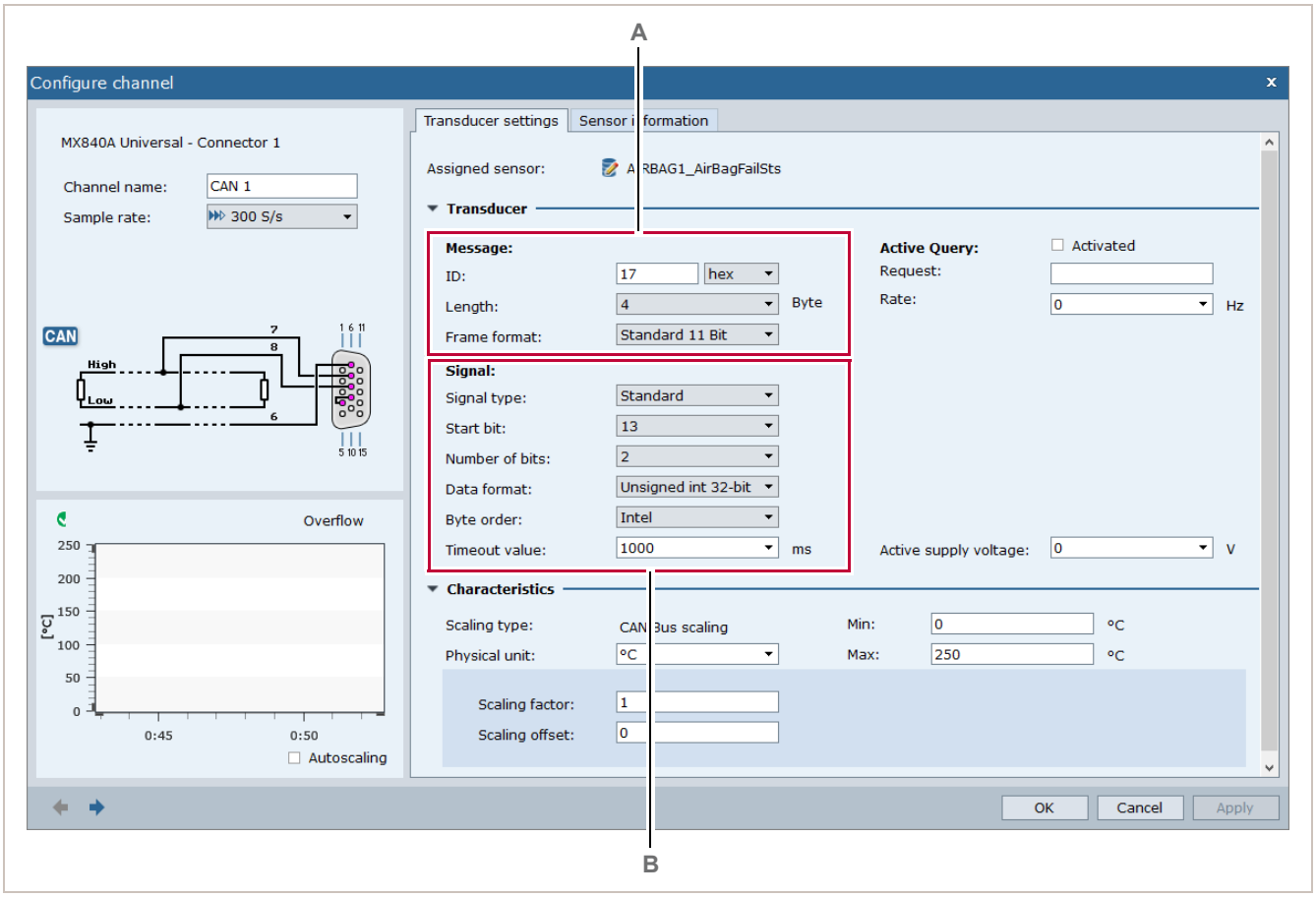


4 Repeat step 3 until all CAN project channels are configured.



5 To display the settings of a CAN signal, click the respective CAN project channel, and then click **Configure**.

✓ The **Configure channel** dialog box is displayed.



A CAN message settings

B CAN signal settings

11.5 CAN-capable data acquisition devices

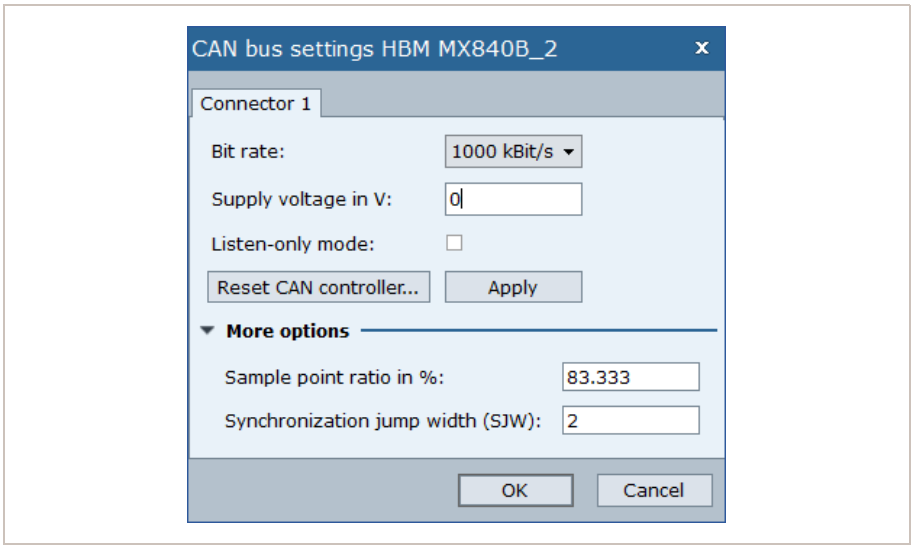
To connect EVIDAS to a CAN bus, you need a data acquisition device that can communicate with a CAN bus.

Module	Description
QuantumX MX840A **QuantumX MX840B**	These modules have 8 connectors each. On each of the two modules you can switch the first connector to classic CAN bus mode (To switch the first connector of an MX840 to CAN bus mode, 242). After switching to CAN bus mode, the first connector has 128 CAN channels.
QuantumX MX471B	This module supports classic CAN and has 4 connectors with 128 CAN channels each. You can connect one CAN bus to each connector.
QuantumX MX471C	This module supports CAN FD and has 4 connectors with 128 CAN channels each. You can connect one CAN bus to each connector.
MGCplus ML70B/AP71 **MGCplus ML71B/AP71**	These modules use AP71 connection boards to communicate with CAN buses. Each AP71 has 2 connectors with 128 CAN channels each. Depending on the MGCplus housing, one or more AP71 connection boards can be mounted. You can connect the CAN buses straight to the AP71 boards, one CAN bus to each connector on the board.

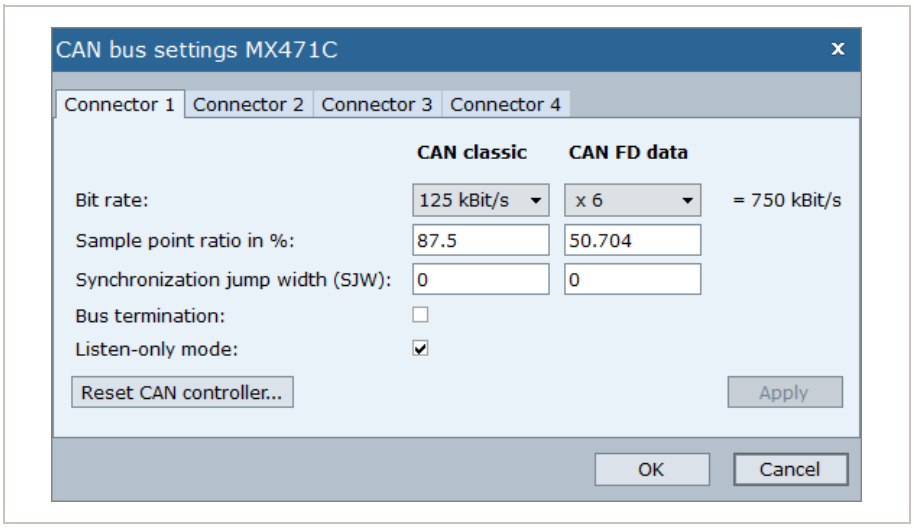
11.6 CAN bus settings

To display, click the **Channels** tab, right-click a CAN channel, and, on the context menu, click **Edit CAN bus settings**.

The settings displayed depend on the data acquisiton device.



CAN bus settings for Quantum MX840B module



CAN bus settings for Quantum MX471C module

Most CAN bus settings are used by both, the classic CAN and CAN FD protocols.

The **CAN FD data factor**, however, exclusively applies to CAN FD. Depending on this factor, you also need to adjust the **Sample point ratio in %** and the **Synchronization jump width**.

CAN bus settings

CAN bus setting	Description
Bit rate	The speed at which the CAN bus operates. Each participant on the CAN bus must use the same bit rate. The maximum speed is 1 Mbit/s with classic CAN and 8 Mbits/s with CAN FD (only for the data field of the CAN message). A slower speed allows for longer signal propagation time and therefore for a longer bus.
Supply voltage in V	This setting is exclusively available for the Quantum MX840 module. Some sensors require a supply voltage to operate. In general, this voltage is specified in the **Configure channel** dialog box (Configure channel dialog box, 76). However, for CAN channels on the Quantum MX840, the setting is available in the **CAN bus settings** dialog box.
Bus termination	Depending on the configuration of the CAN bus, select this option if the data acquisition device must terminate the bus.
Listen-only mode	The data acquisition device reads the CAN messages on the CAN bus, but does not send any messages.
CAN FD data factor	Defines the increase of speed at which the data field of the CAN message is transferred by the CAN FD protocol. The maximum **CAN FD data factor** is 8.
Sample point ratio in %	The sample point ratio is the point inside the time frame assigned to each bit at which the signal is read and interpreted as a logic zero or logic one. CAN controllers allow this point to be configured and it is specified as a percentage of the total bit time in relation to its start. A late sample point allows for longer signal propagation time and therefore for a longer bus. The sample point ratio must fit to the bit rate, the most common recommendation being 87.5%. For CAN FD, you must adjust the sample point ratio due to the reduced bit time during the transmission of the data field of the CAN message.
Synchronization jump width (SJW)	Noise, phase shifts, and oscillator drift create situations where the nominal bit rate does not equal the actual bit rate in the system. The synchronization jump width adjusts the bit clock as necessary by 1 to 4 time quanta (TQ) to maintain synchronization with the transmitted message. The length of a TQ is based on the oscillator period. The base TQ equals twice the oscillator period.

⇨ Defining the CAN bus settings of a connector, 243

12 Appendix C: Digital channels

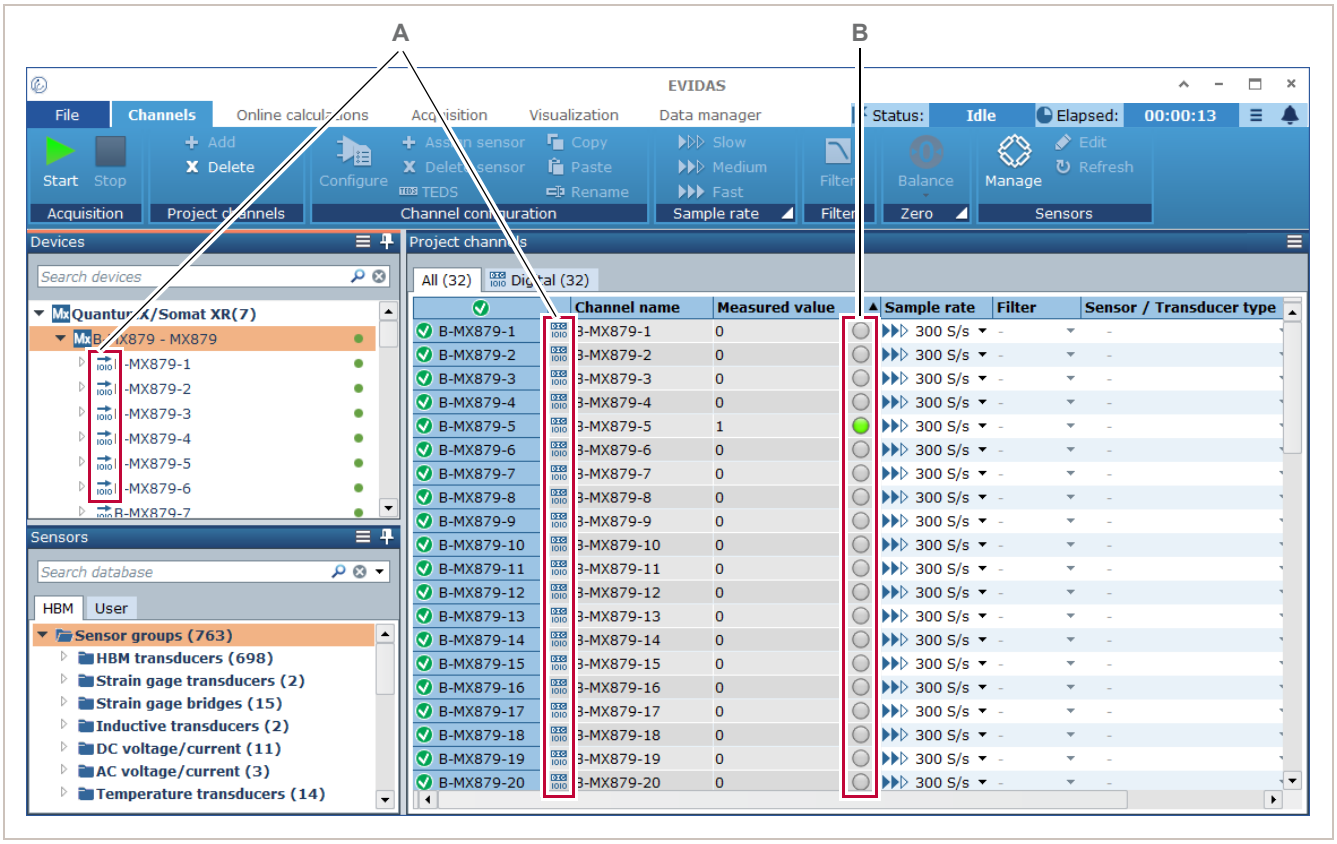
Digital channels are binary channels, i.e., their measured values are mapped to either 0 or 1. To use digital channels, you need a specialized data acquisition device, e.g., a QuantumX MX 879B.

You can use a digital channel, e.g., to monitor the status of an electrical switch in order to determine when it was turned on. To know the exact time of an event can help you to evaluate other project channels, e.g., the voltage and electrical current recorded immediately after the switch is turned on.

12.1 Digital project channels

The digital channels of a data acquisition device are displayed on the **Devices** panel. A digital channel can be in output mode or in input mode .

You can drag a digital channel that is in input mode to the **Project channels** panel. EVIDAS does not support digital channels that are in output mode as project channels. If you want to use a digital channel that is in output mode as project channel, you must first switch it to input mode (Switching digital channel to input mode, 256).



A Digital channel icons, 254

B Digital channel LEDs, 255

Digital channel icons

Digital channel in input mode.

Digital channel in output mode.

Digital project channel.

Digital channel LEDs

Measured value = 0.

Measured value = 1.

You can do the following with digital project channels:

- Define an online calculations channel using a digital project channel as input channel in a formula (Online calculations, 111).
- Visualize a digital project channel in a y(t) chart, on a digital meter, or in a data table (Visualization, 145).
- Record the measured values in a data file (Acquisition, 125).
- Review the measured values (Data manager, 201).

Digital project channels do not need sensor settings or a filter. The electrical value ranges that are mapped to either 0 or 1 are defined in the specialized data acquisition device.

However, you can select a sample rate for a digital project channel.

If you have a digital channel on the **Project channels** panel and, on the **Devices** panel, you switch the digital channel from input mode  to output mode  , the channel is removed from the **Project channels** panel.

12.2 Switching digital channel to input mode

If you want to use a digital channel as project channel and the connector is in output mode, you must first switch the connector to input mode.

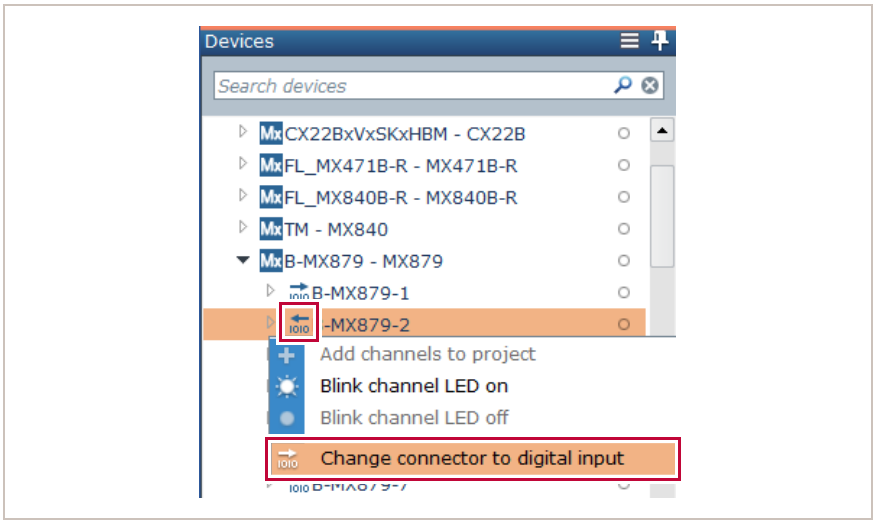
Prerequisites

- Data acquisition device with digital channels is connected.
- EVIDAS is started.
- The digital channel you want to use is in output mode.

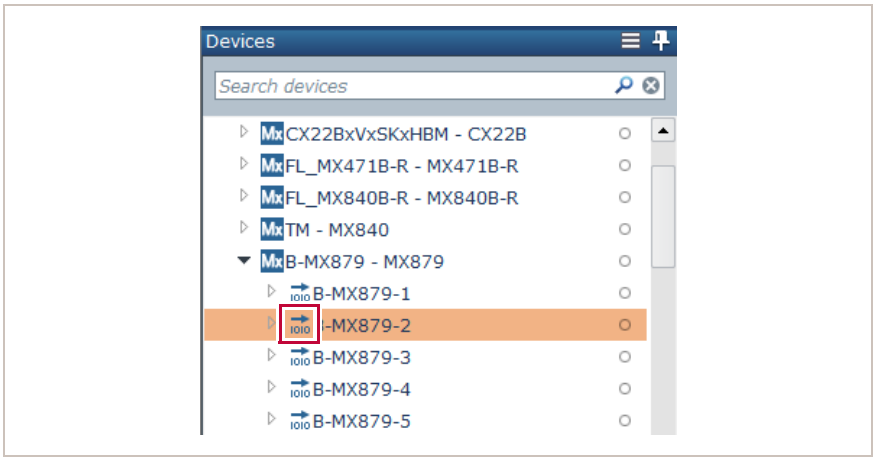
To switch a digital channel to input mode

1 Click the **Channels** tab.

2 In **Devices**, right-click the digital channel that is in output mode, and then click **Change connector to digital input**.



✓ The digital channel is switched to input mode.



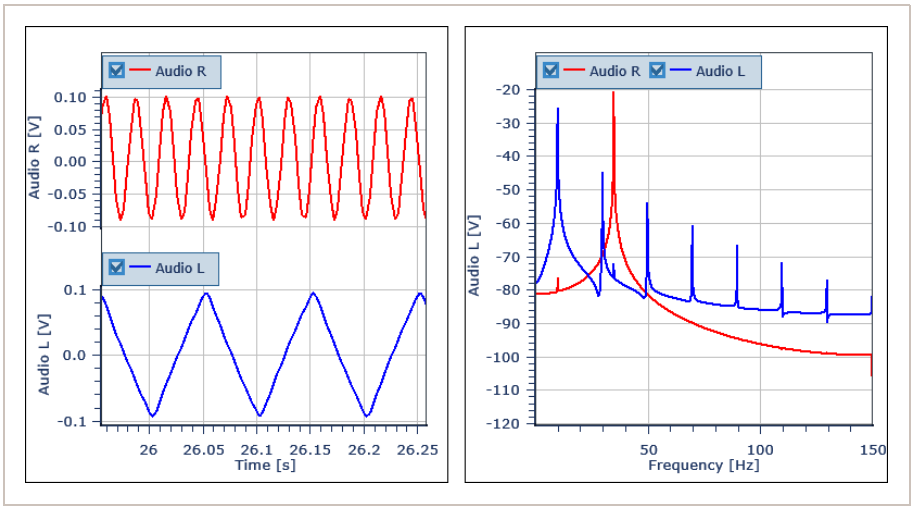
ⓘ A digital channel must be in input mode before you can use it as project channel.

13 Appendix D: FFT Introduction

The fast Fourier transform (FFT) is a specific form of the discrete Fourier transform (DFT) using 2^n measured values as a data block to compute the frequency components of a signal in real time. Mathematically, this is possible because any periodic function y(t) can be decomposed into a weighted sum of sines and cosines, i.e., y(t) can be represented as a spectrum of frequencies f(ω).

13.1 FFT charts

An FFT chart visualizes the frequencies that together form a signal by displaying amplitude over frequency.



A 35 Hz sine and a 10 Hz triangular waveform in a y(t) chart to the left, and an FFT chart with the same signals to the right. In the FFT chart, the frequency components that contribute to the respective waveforms become apparent.

Frequency analysis is used to analyze periodic as well as transient signals. It is applied in acoustics, communications, geology, modal testing, and many more application areas. It allows you to analyze the content of mixed signals such as vibrations, shocks, or noisy signals beyond what is visible in the time domain display (Time domain versus frequency domain, 258).

Example: If a speaker hisses or buzzes, an analysis of the frequency spectrum can show which frequency is causing the unwanted noise, e.g., a 50 Hz / 60 Hz signal from the power supply.

⇨ Visualizing the frequency spectrum of a signal in an FFT chart, 186

13.2 Time domain versus frequency domain

An FFT calculation transfers blocks of data from the time domain to the frequency domain.

Time domain

In the time domain you observe and analyze how a signal changes over time.

You define measurement settings such as the sample rate and filters to eliminate noise (Channels tab, 74). You are interested in individual measured values, e.g., to determine the resistance of a component to stress and strain.

Frequency domain

Looking at the same signal in the frequency domain reveals how the signal's energy is distributed over a range of frequencies.

In the frequency domain, you are not concerned about individual measured values. You must decide about a whole different set of settings that influence the accuracy and performance of the FFT calculation, such as the block size, frequency resolution, or number of lines (Block settings, 259).

Some of those settings depend on each other (Dependencies between block settings, 262).

In order to minimize the effects of spectral leakage (Spectral leakage, 264), you must select a window function to reduce the discontinuities at the edges of the data block (Windowing, 264).

In order to control the noise in the signal and to find its stationary frequencies, you can use an averaging option (Averaging, 267).

The frequency spectrum displayed on the FFT chart is not stored. However, it can be recalculated from the measured values in the review file.

13.3 Block settings

The FFT is applied to 2^n measured values, i.e., a limited block of data. Therefore, the block size and related settings (Definitions of block settings, 259) determine the accuracy and performance of the FFT calculation.

The block settings are defined on the FFT settings tab (FFT settings tab, 167).

13.3.1 Definitions of block settings

Sample rate f_s

The sample rate f_s is the number of measured values acquired per second [S/s]. Its unit is Hertz [Hz] = [1/s].

The sample rate used in the frequency domain is the same as in the time domain.

Sample period Δt

The sample period Δt is the increment in time during which one measured value is acquired. Its unit is seconds [s]. It is related to the sample rate f_s.

$$\Delta t = \frac{1}{f_s}$$

Example: If the measured values are acquired at a sample rate of f_s = 9600 Hz, the sample period Δt is

$$\Delta t = \frac{1}{9600}\ \text{s}$$

Block size N

The block size N is the number of measured values needed for an FFT calculation and must be 2^n with n ≥ 8, i.e., 256, 512, 1024, 2048, 4096, 8192, 16384, etc.

The FFT calculation takes one block of measured values at a time to calculate the frequency spectrum of the signal, e.g., a set of 2048 measured values. Once the frequency spectrum is calculated and displayed, the FFT waits until the next block is acquired, i.e., the next 2048 measured values. As soon as those values are available, the next frequency spectrum is calculated and displayed.

Sample blocks can be overlapped and frequency spectrums can be averaged to reduce noise (Averaging, 267).

The larger the block size, the longer the FFT calculation will take and the more accurate the frequency spectrum will be.

In EVIDAS, if you select a block size N, both the frequency resolution Δf and the number of lines L will be adjusted accordingly (Dependencies between block settings, 262).

Block duration *T*

The block duration T is the total acquisition time needed to record the measured values of one block, i.e., the time needed to record N measured values where N is the block size. The unit of the block duration is seconds [s].

The time to record a block depends on the block size N and the sample period Δt.

$$T = N \cdot \Delta t$$

Example: The time to record a block with N = 2048 measured values at a sample rate of f_s = 9600 Hz is

$$T = N \cdot \Delta t = N \cdot \frac{1}{f_s} = 2048 \cdot \frac{1}{9600\ \text{Hz}} = 0.21333\ \text{s}$$

Frequency resolution Δ*f*

The frequency resolution Δf is a measure for the accuracy of the FFT output and defines the width of one frequency bin on the x-axis.

It is determined by the block duration T.

$$\Delta f = \frac{1}{T}$$

For example, if the block duration T = 1 s, the frequency resolution Δf = 1 Hz. If the block duration T = 2 s, the frequency resolution Δf = 0.5 Hz.

Therefore, the longer the block duration T, i.e., the larger the block size N (assuming that the sample rate f_s is fixed), the more information is available for the FFT calculation and the smaller, i.e., better, the resulting frequency resolution Δf will be.

It follows that,

$$\Delta f = \frac{1}{T} = \frac{1}{N \cdot \Delta t}$$

In EVIDAS, if you select a frequency resolution Δf, both the number of lines L and the block size N will be adjusted accordingly (Dependencies between block settings, 262).

Example: The frequency resolution Δf for the above block duration T = 0.21333 s is

$$\Delta f = \frac{1}{T} = 4.6875\ \text{Hz}$$

Frequency span *F*

According to the Nyquist-Shannon sampling theorem, if a function y(t) contains no frequencies higher than B Hertz, it is completely determined by giving its ordinates at a series of points spaced 1/(2 · B) seconds apart.

Therefore, if *F* is the maximum frequency of a band-limited signal, the sample period Δ*t* must be 1/(2 · *F*) seconds apart to completely determine the signal.

$$\Delta t = \frac{1}{2 \cdot F}$$

This leads to

$$F = \frac{1}{2 \cdot \Delta t} = 0.5 \cdot f_s$$

The frequency span *F* defines up to which frequency the FFT will produce a result.

Example: If the measured values are acquired at a sample rate f_s = 9600 Hz, the frequency span *F* is

$$F = 0.5 \cdot f_s = 4800 \text{ Hz}$$

Number of lines *L*

The number of lines *L* is defined as the ratio of the total frequency span *F* in relation to the frequency resolution Δ*f*.

$$L = \frac{F}{\Delta f}$$

The better the frequency resolution Δ*f*, i.e. the smaller Δ*f*, the larger the number of lines, i.e., the information content of the frequency spectrum.

In EVIDAS, if you select a number of lines *L*, both the block size *N* and the frequency resolution Δ*f* will be adjusted accordingly (Dependencies between block settings, 262).

Example: For the above frequency span *F* = 4800 Hz and frequency resolution Δ*f* = 4.6875 Hz, it follows that

$$L = \frac{F}{\Delta f} = \frac{4800 \text{ Hz}}{4.6875 \text{ Hz}} = 1024$$

⇨ FFT settings tab, 167

13.3.2 Dependencies between block settings

Several of the FFT frequency settings are dependent on each other.

Sample rate f_s and frequency span *F*

The frequency span *F* is determined by the sample rate f_s.

$$F = 0.5 \cdot f_s$$

Example: If the measured values are acquired at a sample rate f_s = 9600 Hz, the frequency span is *F* = 4800 Hz. No frequencies higher than 4800 Hz will be displayed on the FFT chart.

Block duration *T* and block size *N*

With the sample rate f_s fixed, the block duration *T* is determined by the block size *N*.

$$T = N \cdot \Delta t$$

With

$$\Delta t = \frac{1}{f_s}$$

it follows that

$$T = \frac{N}{f_s}$$

Example: The time to record a block with *N* = 2048 measured values at a sample rate of f_s = 9600 Hz is

$$T = \frac{N}{f_s} = \frac{2048}{9600\ \text{Hz}} = 0.21333\ \text{s}$$

Frequency resolution Δ*f*, number of lines *L*, and block size *N*

If the sample rate f_s is fixed, and either the frequency resolution Δ*f*, or the number of lines *L*, or the block size *N* is given, the other two settings are determined.

To verify this, let us say the block size *N* is given. The frequency resolution Δ*f* is then determined by

$$\Delta f = \frac{1}{N \cdot \Delta t} = \frac{f_S}{N}$$

Now, if the frequency resolution Δ*f* is thus determined, the number of lines *L* is derived as follows

$$L = \frac{F}{\Delta f} = \frac{0.5 \cdot f_S}{\Delta f}$$

And if the number of lines *L* is determined, then again the block size *N* can be derived as

$$N = \frac{f_S}{\Delta f} = f_S \cdot \frac{L}{0.5 \cdot f_S} = 2 \cdot L$$

Example: If the measured values are acquired at a sample rate of f_s = 9600 Hz, and the block size is *N* = 2048, the frequency resolution is

$$\Delta f = \frac{f_S}{N} = \frac{9600\ \text{Hz}}{2048} = 4.6875\ \text{Hz}$$

Then it follows that the number of lines is

$$L = \frac{0.5 \cdot f_S}{\Delta f} = \frac{4800\ \text{Hz}}{4.6875\ \text{Hz}} = 1024$$

And thus the block size is again

$$N = 2 \cdot L = 2048$$

⇨ FFT settings tab, 167

13.4 Windowing

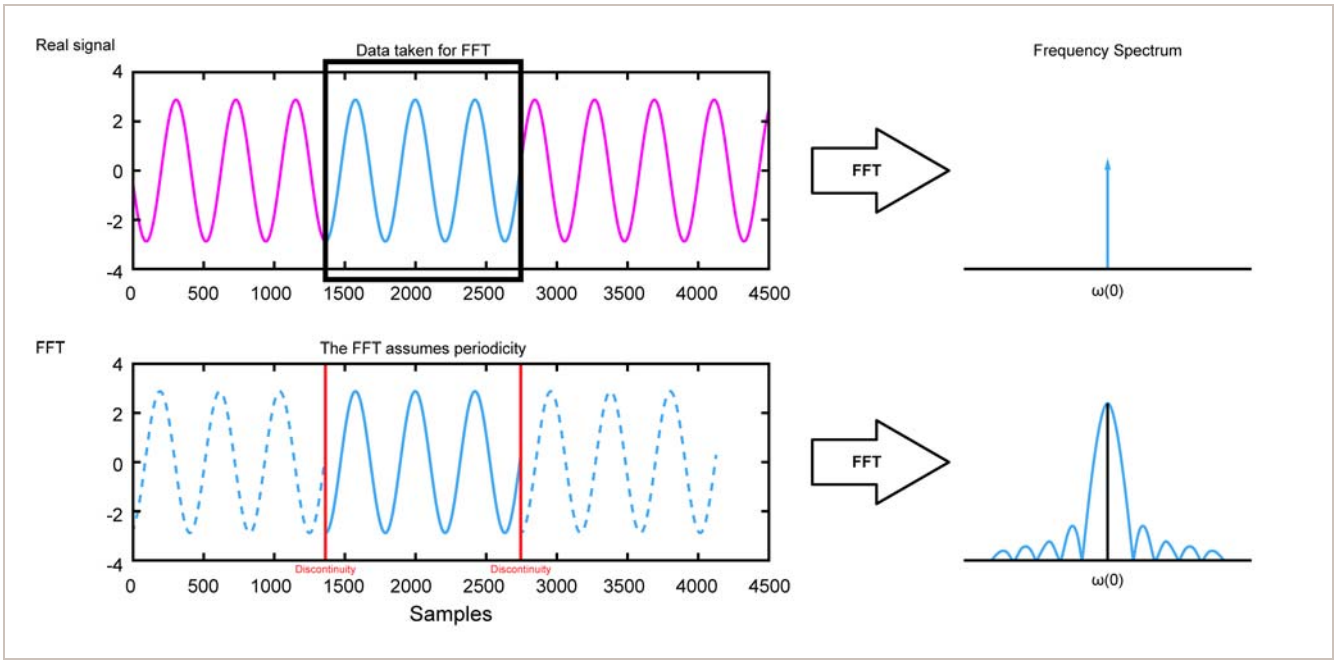
Windowing weighs the measured values of a sample block prior to the FFT calculation in order to reduce the discontinuities at the edges of the sample block and thus to improve the accuracy of the frequency spectrum obtained by the FFT calculation.

Spectral leakage

The FFT calculation assumes that the measured values recorded during the block duration *T* represent just one period of an infinitely repeating periodic signal.

However, in practical applications the block usually contains an incomplete number of cycles, which leads to discontinuities at the edges of the sample block, i.e., when placing one block next to a copy of it, the last measured value of the block does not correspond to the first measured value of the next (copied) block.

In the FFT, these discontinuities show up as frequency components not present in the original signal, i.e., the frequency spectrum calculated by the FFT does not represent the spectrum of the original signal, but a smeared version of it. It appears as if energy from one frequency leaks into other frequencies. The phenomenon is therefore known as spectral leakage.



▢ Due to discontinuities at the edges of the sample block, a sine signal is not represented as a single sharp peak, but smeared across neighboring frequency bins.

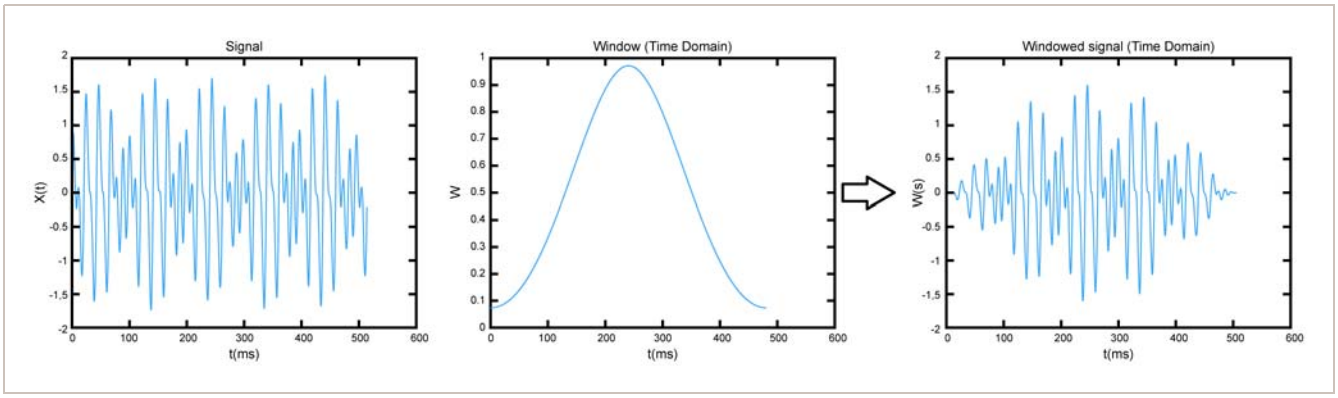
An FFT calculation analyzes only a short slice of the signal and therefore outputs the frequency spectrum of a theoretical signal, that is composed of infinite repetitions of that slice of 2^n measured values.

Window functions

Window functions are used to reduce the amplitude of the discontinuities at the edges of the sample block. The measured values of the signal are multiplied with the window function whose amplitude tapers towards zero at the edges.

The width of the window must match the block duration *T*, i.e., the window function operates on all measured values within a block.

The result is a windowed signal with very small or no discontinuities and therefore reduced spectral leakage.



You can choose from among different types of window functions to reduce spectral leakage.

Each window function is designed to optimize the FFT output for a specific signal type.

If you do not know which signal to expect, the Hann(ing) window is a good starting point and is sufficient in 95% of all cases.

Window function	Description	Best for signal type
Rectangular	This is the window function with the highest amount of spectral leakage. It has a value of 1 over the whole block duration and is equivalent to no window at all.	Transient signals that have a duration shorter than the block duration.
Hann(ing)	Bell-shaped curve.	Transient signals that have a duration longer than the block duration. General-purpose window function.
Hamming	Modified version of the **Hann(ing)** window.	Transient signals that have a duration longer than the block duration. General-purpose window function.
Blackman	Provides a high rejection of signals outside its main lobe.	Single tone measurements.
Blackman-Harris	Modified version of the **Blackman** window.	Single tone measurements.
Flattop	This window function has the best amplitude accuracy, but a poor frequency resolution.	Use this window when amplitude accuracy is important. However, it cannot be used to detect nearby frequency components.

If a window function is applied, it reduces the amplitude of the time domain signal, especially at the edges of the sample block. The resulting loss in signal power is called *coherent gain* (Comparison of window functions, 266).

The FFT output must be corrected to compensate for this loss of signal power.

Comparison of window functions

When selecting a window function, consult the following table for their respective features.

Window function	Highest Side-Lobe Level [dB]	Side-Lobe Roll-Off-Rate [db/Oct]	Coherent Gain	Normalized Equivalent Noise Bandwidth [bins]	-3 dB Bandwidth [bins]	-6 dB Bandwidth [bins]	Max. Amplitude Error [dB]
Rectangular	-13	-6	1.00	1.00	0.8845	1.21	-3.92
Hann(ing)	-31	-18	0.50	1.50	1.4382	2.00	-1.42
Hamming	-43	-6	0.54	1.36	1.3008	1.81	-1.75
Blackman	-58	-18	0.42	1.73	1.6800	2.35	-1.10
Blackman-Harris	-92	-6	0.36	2.00	1.8962	2.72	-0.83
Flattop	-93	-6	0.22	3.77	3.7200	4.58	< -0.02

⇨ FFT settings tab, 167

13.5 Averaging

Averaging is used to control the noise in the signal in order to identify the stationary frequencies that are otherwise hidden in the noise band. As noise is considered to be a random component of the signal and hence having a zero mean, it can almost fully be eliminated by averaging.

EVIDAS offers the following averaging options:

Averaging option	Description
Exponential averaging	Exponential averaging uses all frequency spectrums calculated within a time period you define and averages the amplitudes per frequency across those spectrums, whereby more recent frequency spectrums are taken into account more prominently than older ones. After the time period you define, e.g., 10 seconds, the influence of historical data on the average drops below 1%.

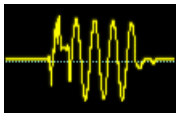
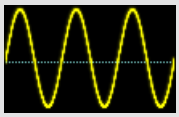
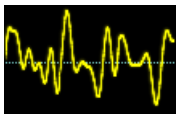
⇨ FFT settings tab, 167

13.6 Frequency analysis options

You can choose between the following frequency analysis options:

Analysis option	Description	Unit of y-axis
FFT Spectrum (peak)	Displays amplitude over frequency, i.e., the frequency distribution of the signal. The FFT Amplitude Spectrum *AS* [V_{peak}] is determined by $AS[V_{peak}] = \frac{1}{CoherentGain} \cdot \frac{\lvert FFT \rvert}{N}$ The FFT output is thus normalized by the block size *N*, and the loss of energy caused by windowing is compensated with the *CoherentGain* factor of the respective window function (Comparison of window functions, 266).	[V_{peak}]
Linear Spectrum (RMS)	Displays RMS over frequency. The linear spectrum *AS* [V_{rms}] is determined by $AS[V_{rms}] = \frac{1}{\sqrt{2}} \cdot AS[V_{peak}]$ The factor $\frac{1}{\sqrt{2}}$ is only applied for non-DC components.	[V_{rms}]
Power Spectrum (PS)	Displays the energy of a signal over frequency. The power spectrum *PS* is determined by $PS = 2 \cdot AS^2$ for use with already compensated windowed data.	[V^2] = [W]
Power Spectral Density (PSD)	Displays the strength of the variations in energy over frequency, i.e., at which frequencies variations are strong and at which frequencies variations are weak. The Power Spectral Density *PSD* is determined by $PSD = \frac{PS}{NENBW \cdot \Delta f}$ where *NENBW* is the normalized equivalent noise bandwidth of the respective window function (Comparison of window functions, 266) and Δf is the frequency resolution of the FFT.	[V^2/Hz] = [dBm/Hz]

Use the following table to find the analysis option that fits your application.

Signal type	Your application	You can use
 **Transient**	Depending on requirements and window.	Power Spectral Density (PSD)
	Depending on requirements and window.	Power Spectrum
 **Periodic**	Observe the spectrum of the signal.	FFT Spectrum
	Observe the spectrum of the signal.	Linear Spectrum (RMS)
	Observe the power of the signal.	Power Spectrum
 **Random**	Observe the power of the signal.	Power Spectrum
	Normalized measurement of the power per unit bandwidth.	Power Spectral Density (PSD)

⇨ FFT settings tab, 167⇗

14 Bibliography

The following HBM publications are reference works on strain gauges and can be downloaded from the HBM website **www.hbm.com**.

- Hoffmann, K.: An Introduction to Stress Analysis using Strain Gauges, Hottinger Baldwin Messtechnik GmbH, Darmstadt, 1987.
- Hoffmann, K.: Practical Hints for the Application of Strain Gauges, Technical Note VD 84005e, Hottinger Baldwin Messtechnik GmbH, Darmstadt.
- Horoschenkoff, A., S. Klein, and K.-H. Haase: Structural integration of strain gages, Hottinger Baldwin Messtechnik GmbH, Darmstadt, 2006.

Index

D

E

F

G

H

I

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z