SKF Microlog Analyzer dBX

User Manual
Part Number **15V-090-00102-100**
Revision **E – October 2024**
Software Version **1.2**

Read this manual carefully before using the product. Failure to follow the instructions and safety precautions in this manual can result in serious injury, damage to the product or incorrect readings. Keep this manual in a safe location for future reference.

SKF Sverige AB
Ålgatan 10D, 973 34 Luleå, Sweden
Telephone: +46 (0) 31 337 10 00

Product Registration

Please take a moment to register your product to receive exclusive benefits offered only to our registered customers, including technical support, tracking your proof of ownership and staying informed about upgrades and special offers. (Please visit our website for more details on these benefits.)

General Product Information

General information such as datasheets and catalogues are published on the Condition Monitoring Systems site on SKF.com. Supporting product information can also be downloaded from the SKF Technical Support self-service web portal.

Product Support Contact Information

Repair** and **Calibration Services – Under Warranty, please contact your nearest SKF Technical Support representative and raise a support case. For more details about the Warranty, see Appendix A.

Product Sales – For information on purchasing condition monitoring products, services and support on products out of warranty, please contact your local SKF sales office or distributor.

How to request Technical Support – Please open a support case using the Technical Support group's self-help portal at www.skf.com/cm/tsg. Once your support case is submitted, a technician will contact you to begin working on your issue. For urgent issues we are available at these times:

- Monday through Friday, 5:00 a.m. to 4:00 p.m. Pacific Time
  Phone: +1 800 523 7514 within the US or +1 858 496 3627 outside the US.

- Monday through Friday, 8:00 a.m. to 4:00 p.m. Central European Time
  Phone: +46 31 337 65 00.

- Monday through Friday, 7:30 a.m. to 4.30 p.m. India Standard Time
  Phone: +60 16 699 9506.

Table of contents

1 Product description

1.1 Introduction

Before using the software, please read and understand the separate Microlog Analyzer dBX Quick Start Guide.

If the equipment is used in a manner not specified by the manufacturer, both the safety and functionality of the equipment may be impaired.

SKF Microlog Analyzer dBX is a portable condition monitoring system that uses hybrid, keypad and touchscreen navigation based on a Microsoft Windows operating system.

The dBX Series Microlog system is used by machinery maintenance personnel who wish to collect and analyze vibration data from their rotating machinery, to help reduce maintenance costs and downtime.

A dBX Series Microlog System consists of three components:

- Microlog dBX Vibration Analyser
- Application modules installed on the Microlog Vibration Analyser
- A host computer with SKF @ptitude Analyst software and the Thin Client Transfer TCT.dBX or SKF @ptitude Observer software and the Data Bridge web service.

  Note: If the SKF @ptitude Analyst software is hosted on the cloud, only TCT.dBX is required on the host computer, along with the end point connection information to access the @ptitude Analyst Transaction Service.

The Microlog Analyzer dBX data collector is a portable, four channel data acquisition and storage terminal. It collects machinery vibration, temperature, and other condition monitoring measurements. Together with visual observations, Microlog Analyzer dBX allows for detailed machine condition analyses in a harsh industrial environment.

The Microlog Analyzer dBX, with its preinstalled and licensed modules, performs tasks required for machinery predictive maintenance.

It automatically collects both dynamic (vibration) and static (process) measurements from almost any source, it provides easy to use setup screens for quickly capturing data related to specific applications, such as balancing, etc., and it allows you to configure one or twelve measurements for automatic data collection at one sensor location.

1.2 Third party licences

Some pieces of licensed software such as open source or third-party libraries have been used when developing this product.

For any enquiries contact SKF's Technical Support Group TSG.

1.3 Startup password

Adding a startup password to your Microlog Analyzer dBX helps protect your data and secure your device. You can select to change, enable, or disable the password option from the Utility module.

1.3.1 Setting up a startup password

When the Microlog Analyzer dBX is powered on for the first time, the system requires you to set up a startup password.

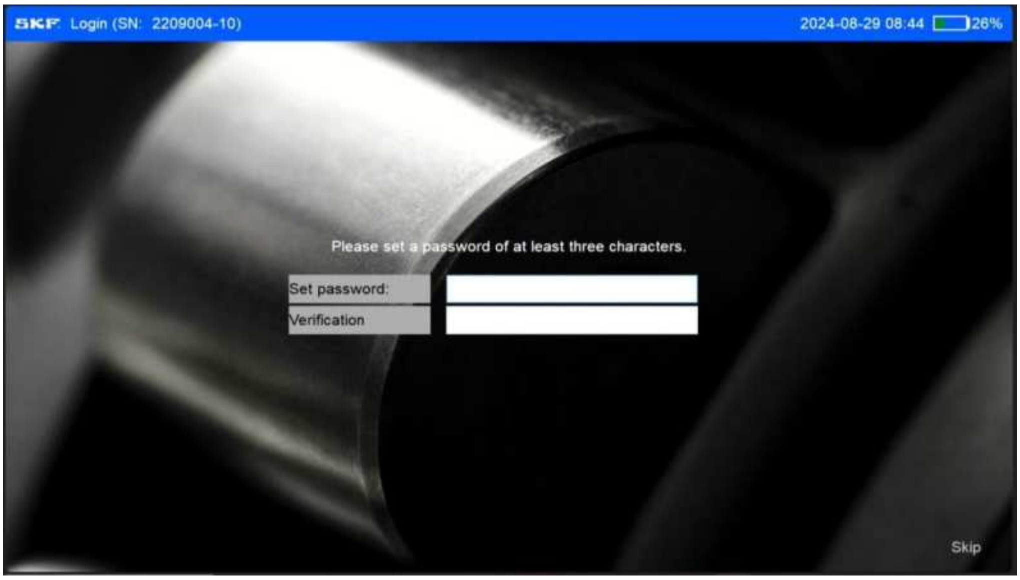


Figure 1 Login screen – Set password

1. Tap the **Set password** field to display the keypad and enter the password.

2. Re-enter the password in the **Verification** field.

Figure 2 Password keypad

Once the password configuration is complete, the system will require you to enter the password each time the device is powered on.

Tap the **Password** field to display the keypad and enter the password to log in.

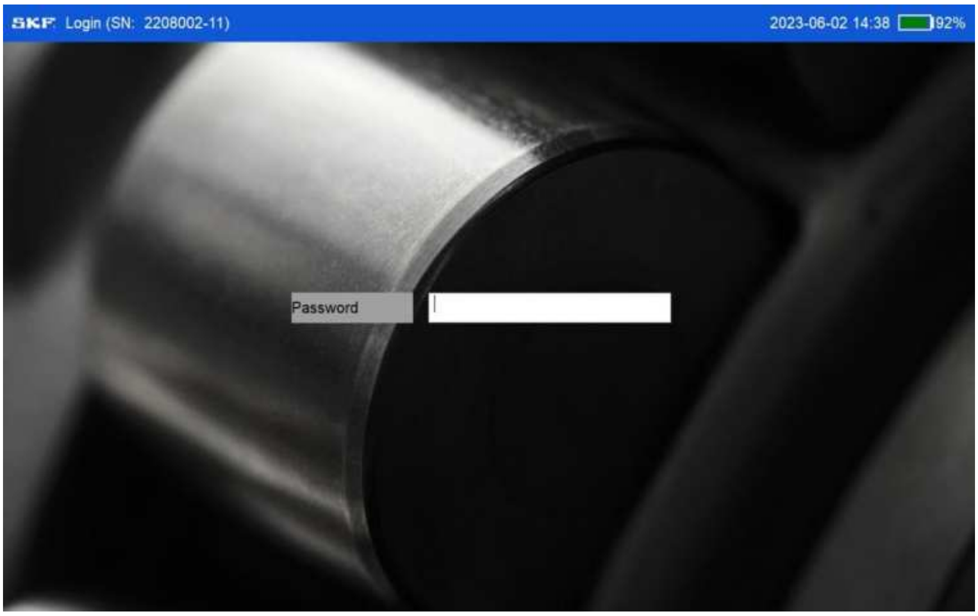


Figure 3 Log in with password

1.3.2 Skip the password option

To skip the password option and log in directly, tap **Skip** at the bottom-right corner of the display.
Note: If you select **Skip**, the password option will be disabled. To enable the password option, go to the Utility module.

1.3.3 Reset a forgotten password

1. Contact SKF's Technical Support Group TSG and provide the serial number (SN) located at the top left corner of the display to get an unlock code.
2. Use the unlock code to log in to the system again.
3. After an unlock code is used, reset the password settings from the Utility module.

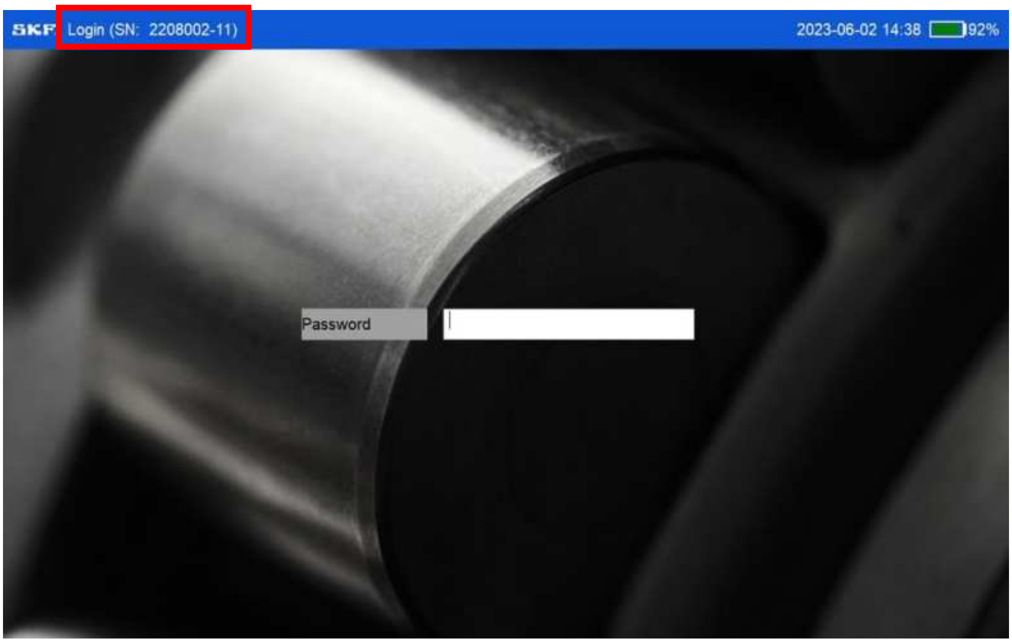


Figure 4 Serial number – example

1.3.4 Logout

To log out, click the logout icon in the top right corner of the home screen.

In the Utility module, you can set a timer for auto logout.

Note that setting a startup password is required to use the logout function.

1.4 USB Security

To protect the device from cyber-attacks via its USB connector, Microlog dBX implements USB port locking.

In normal circumstances, the USB port is blocked from operating with any of the following approved devices:

- USB mouse
- Approved USB transfer cable (CMAC 9010)
- Power-only devices such as USB torches
- Approved USB-HDMI output adaptors

Note: The only other type of USB device allowed is a flash drive/memory stick type.

Since this type of device can be used to deliver malware to a Windows-based computer system, the dBX implements a trust process which is triggered if the user attaches a USB memory key to the device, but only while the following applications are active:

- Analyzer (or any of its derivatives such as gE Enveloping)
- Balancing
- Utility
- dBX Documents

Upon inserting a USB memory key in the above circumstances, the device will display a prompt message: **< file “Trust USB” to be sent >**

Tap **Yes** to enable the key and continue or **No** to cancel. In the latter case, the USB device will not be available.

As a further security step, if a start-up password is set in the device, see Startup password, the user will be prompted to enter the password before the key becomes available to the application.

1.5 Battery

Microlog Analyzer dBX uses two batteries in sequence. The power manager uses the first battery and then switches to the second battery when first battery is drained.

Note: To balance usage time and maintain the health of both batteries, swap them every six months.

When both batteries are used, the percentage level is calculated by setting their combined full power as 100%.

The battery symbol in the top right corner shows the remaining battery level or indicates if the device is connected to a charger. The battery symbol colour changes based on the battery level.

Symbol	Description
	Battery level above 20%.
	Battery level is between 10% and 20%.
	Battery level is below 10%.
	Connected to a charger. **Note**: The charging symbol may take up to 20 seconds to appear after connecting the charger.

2 Data Collector module

2.1 Introduction

Before using the software, please read and understand the separate Microlog Analyzer dBX Quick Start Guide document.

The **Data Collector** module is used for collecting data with the Routes built from the SKF @ptitude Analyst or SKF @ptitude Observer software and downloaded to the **Microlog Analyzer dBX**.

Enter a Route from the Data Collector module, select a Point node and perform measurements for each Point. Upload the archive data to the SKF @ptitude Analyst or SKF @ptitude Observer for further analysis and creating reports.

To start Data Collector, using the navigation keys, navigate to the SKF Data Collector icon and press **OK** or double tap the icon.

Figure 5 Home screen – Data Collector icon

2.2 Route manager

2.2.1 Introduction

When you open Data Collector, you will see the **Route manager** view. The Route manager presents the available Routes stored on the device. Use the navigation keys or double tap on a Route to select it.

The Route information or instruction will be displayed on the right side of the display. Select Route **Instruction** or **Information** view by tapping one of the tags on the top.

Note: The Instruction information will be available if the downloaded Route contains instructions.

➢ A **Route** is a predefined sequence of measurements representing the Point of measurement on one or multiple pieces of equipment within a device or machines of a factory.

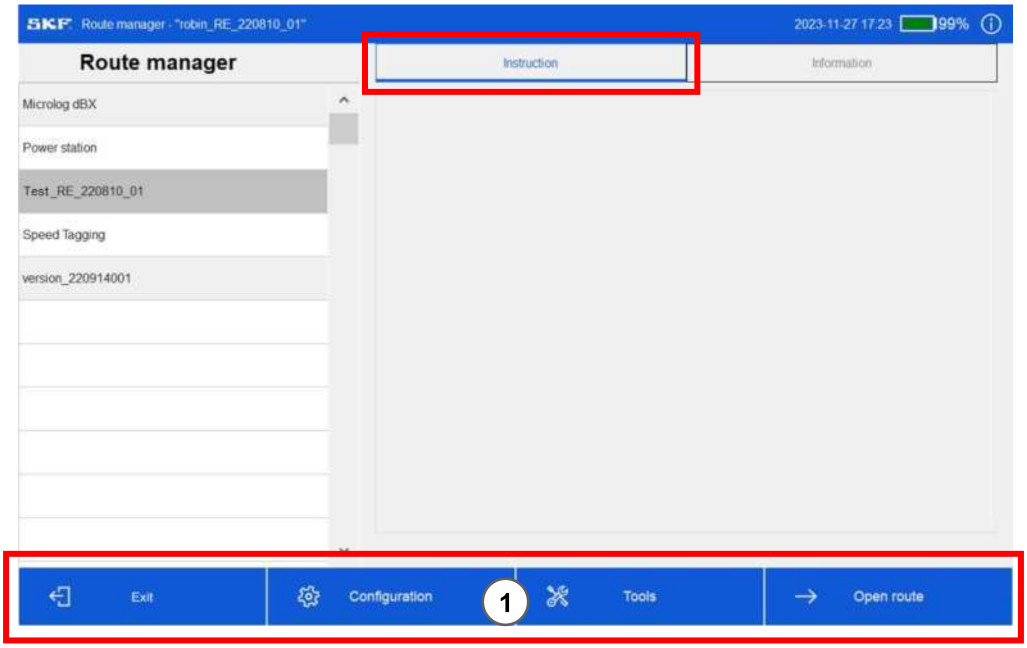


Figure 6 Route Instruction view

1. Menu buttons

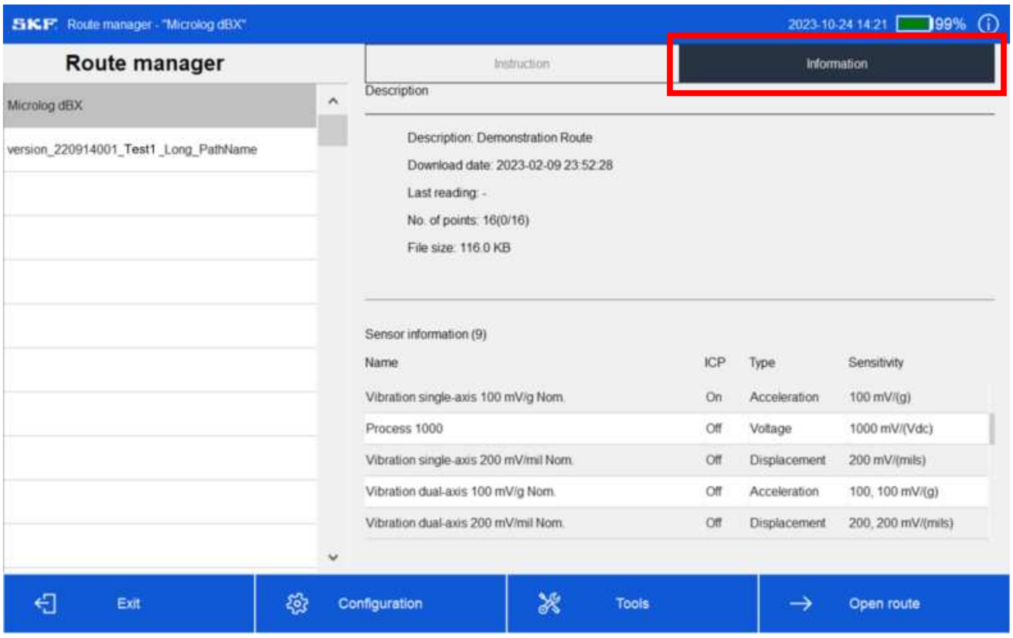


Figure 7 Route Information view

2.2.2 Open Route

To open a Route, use the navigation keys to select a Route and press the **OK** key or tap **Open route**.

If there is no username downloaded, it will open the Route directly.

For the username to be available, download it together with the Route. Every user/operator set is part of each individual Route.

If you log in with a username, it will be saved to the Route, giving the possibility to track which user performs the data collection.

Tap **Back** to return to the previous view.

2.2.3 Configuration

From the configuration view, you can set up the optional functions. Select from four sub-menus for setting up the related parameters **Axis**, **Display**, **Measurements,** and **Log option**.

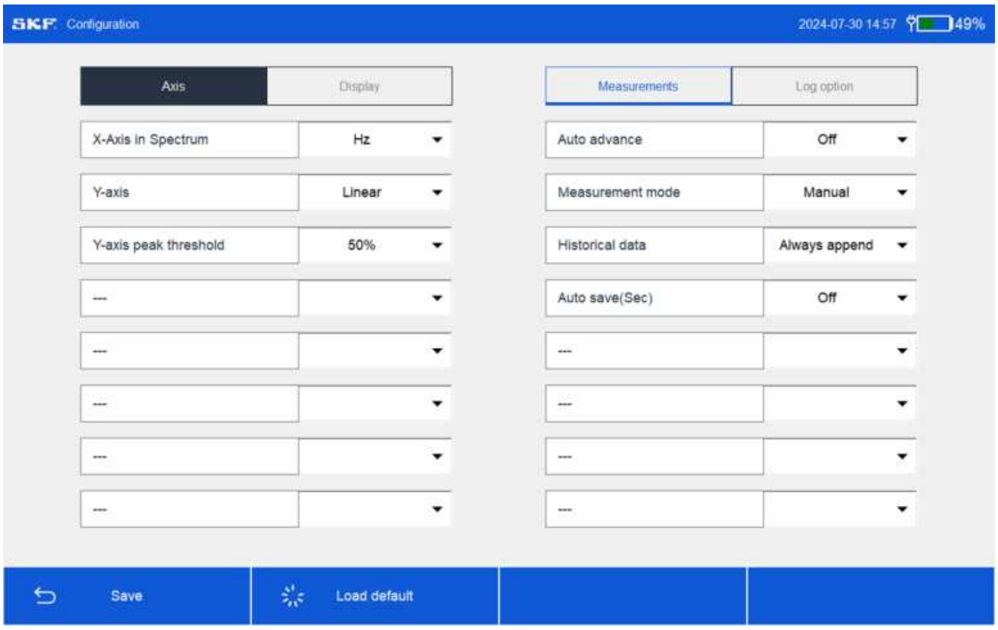
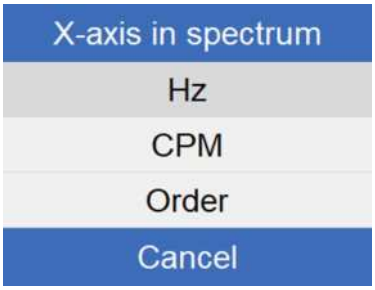
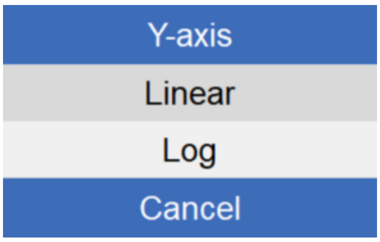


Figure 8 Configuration view

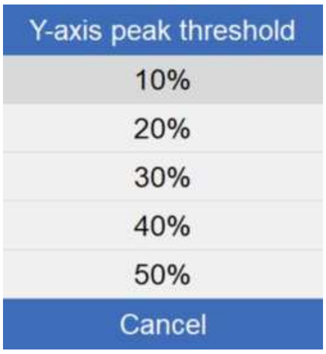
- **Axis**
  - **X-Axis in Spectrum** – Select **Hz**, **CPM** or **Order**. The selected unit will be displayed as the default X-Axis unit for Spectrum plots.



  - **Y-Axis** – Select **Linear** or **Log scale** as the Y-axis of Spectrum plots.

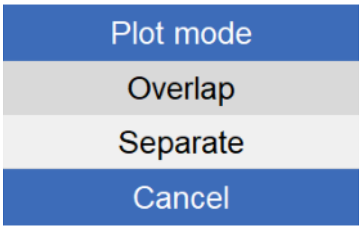


  - **Y-axis peak threshold** – Select a percentage number as the default Y-axis peak threshold used to select peaks when the cursor's jump peak function is enabled. Those peaks whose amplitudes are higher than the threshold level will be selected for the cursor to move on them.
    You can change the threshold level manually when working with a cursor.



- **Display**

  - **Plot mode** – Select **Overlap** or **Separate** as the default plot format. This option will set the default format of multiple channels as overlap or separate. You can change the setting from the display menu of a plot.



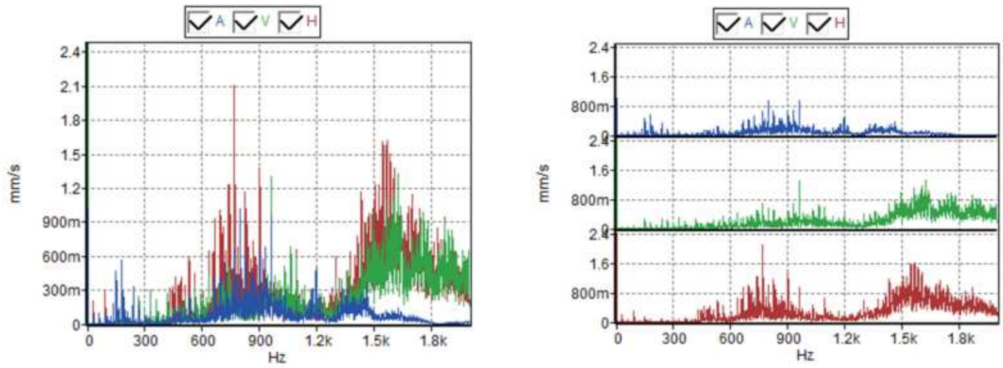


Figure 9 Overlap display (left), Separate display (right)

  - **Show 1X** – Select **On** or **Off** from the dropdown. Shows the running speed as a vertical dotted line in the FFT plot.

- **Show band alarm** - Select **On** or **Off** from the dropdown menu.
  On – Will show you the band alarm curves when the Route contains band alarm information.
  Off – Will hide the band alarm curves as default.
  You can change the setting from the display menu of a spectral plot.

Figure 10 Band alarm

- **Show Fault frequencies** – Select **On** or **Off** from the dropdown menu.
  On – Will show you the fault frequencies on a spectral plot when the Route contains fault frequencies information.
  Off – Will hide the fault frequencies as default. You can change the setting from the display menu of a spectral plot.

Figure 11 Fault frequency

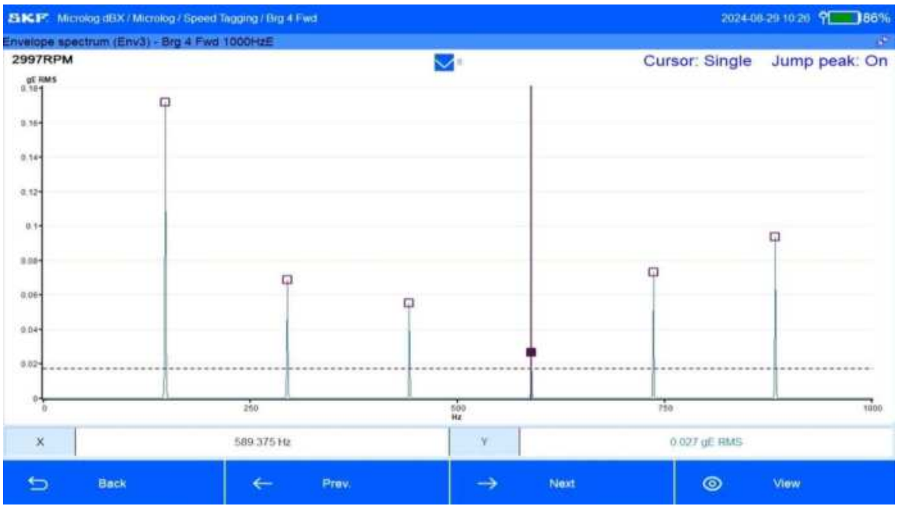


Figure 12 Highlighted peaks – example

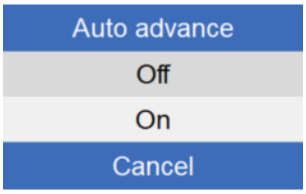
- **Orbit filter** – Set the default Orbit filter from the menu for orbit plots. You can also change the setting from the plot menu of an orbit plot.

- **Orbit cycle** – Select the between 1–16 orbit cycles for the orbit plots. You can also change the setting from the plot menu of an orbit plot.

- **Jump peak** – Select default value **On** or **Off** from the dropdown menu. Peaks are sorted by y-value and the top 10 peaks are highlighted. With the Jump Peak function is enabled, the cursor will jump from one peak to another when you move it. You can turn the Jump Peak function on or off by pressing the Up key twice while working with a plot.

- **Measurements**

  - **Auto advance** – Select **On** or **Off** from the dropdown menu.
    On – Move the selected Point to the next Point automatically when the measurement of a Point is completed.
    Off – Move the cursor manually to the next Point.



  - **Measurement Mode** – Select **Auto** or **Manual** from the measurement mode menu. When Auto is selected, the measurement will start directly after the **Start** / **Stop** key is pressed.
    When **Manual** is selected, it will only show the waveforms charts after the **Start** / **Stop** key is pressed, but it will not start the measurement until the **OK** key is pressed for confirmation.

  - **Historical data** – Select **Always append** or **Always overwrite** from the Historical data menu.
    When **Always Append** is selected, new measurement data is added to the historical data table, which stores up to 12 most recent records. If the table reaches its 12-record limit, the oldest record will be automatically deleted when a new data is stored.
    When **Always overwrite** is selected, the last measurement data will be overwritten by the new measurement data and only one data record is stored in the historical data table.

- **Auto Save (Sec)** – Select **Off**, **1**, **2**, **3**, **4**, or **5** seconds from the menu. When **Off** is selected, you must tap **Save** after each measurement. If you select a number of seconds, the measured data is saved automatically after the specified time.

Tap **Load default** to restore the settings to default values.

- **Log option**

  - **Auto log raw data** – At the beginning of each measurement, the data collector module acquires the raw data, which is a continuous signal record sampled at 102.4 kHz.
    The raw data can be converted into various measured functions to complete the measurement. You can replay the raw data file as an audio clip using a Bluetooth headset or perform post analysis to derive more information for diagnosis purposes.

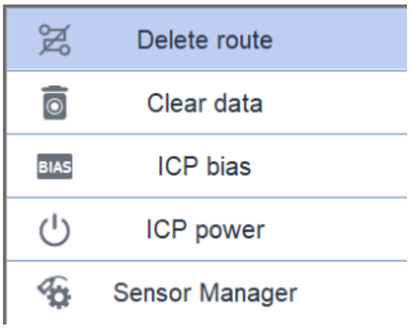
    Note: The raw data will occupy a large storage space, and its use should be avoided.

Select **Off**, **On**, **Alert** or **Danger** from the dropdown menu as the default raw data logging status after each measurement.

- **Off** – Never store the raw data.
- **On** – Always store the raw data.
- **Alert** – Store the raw data when the measured data is in alert status (Alarm 1).
- **Danger**: Store the raw data when the measured data is in Danger status (Alarm 2).

2.2.4 Tools

From the **Tools** menu, you can delete a selected Route, clear the archive data of a selected Route, check the ICP bias and power, or enable the Sensor Manager to edit the sensors used with your data collector.



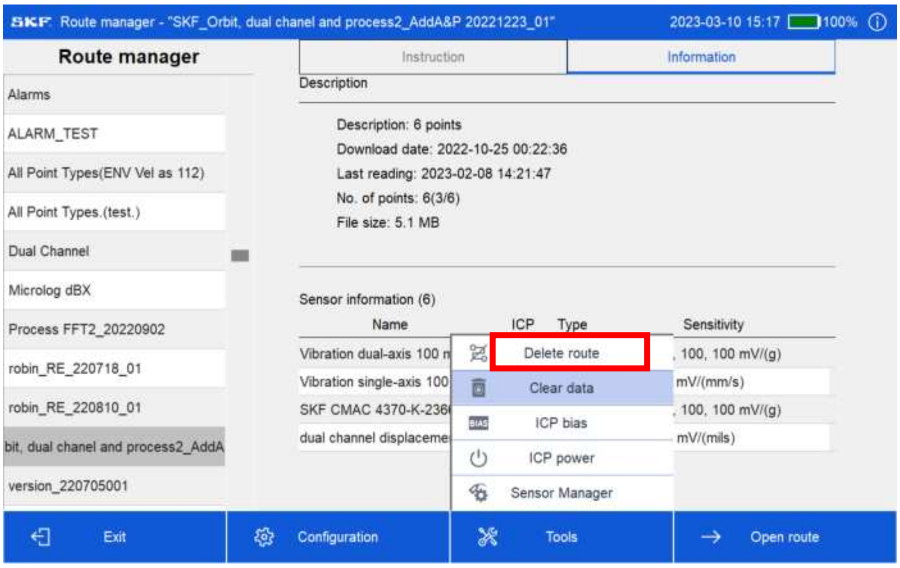
- **ICP bias check** – opens a new window, where you can connect your ICP sensors for testing.
  Select **Verify** to perform the checking, the result is shown in a window, which may indicate one of the following conditions:
  - **Short circuit (low bias)** – Usually indicates a faulty cable with a short loop.
  - **Normal (normal bias)** – The ICP circuit is working normally.
  - **Open circuit (high bias)** – If there is no sensor connecting to this channel, the bias check shows status **Open**. If an ICP sensor is connected to this channel, open circuit usually means a faulty cable.

- **ICP power check** – Select ICP Power® to check the ICP power supply of your Microlog dBX. To perform the checking, disconnect all sensors and select **Verify**. In normal conditions, the power supply of all channels should be around 25+/-1 V. If the result shows abnormal voltage levels, contact SKF's Technical Support Group TSG for assistance.

  Note: ICP® is a registered trademark of PCB Piezotronics Europe GmbH in Germany and other countries.

- **Delete Route**

1. Select a Route from the **Route manager list**.
2. Select **Tools** > **Delete route** to delete the selected Route.



3. A pop-up window with a numeric keypad will open. Enter the code shown in the title bar to confirm the deletion.

- **Clear data**

1. Select a Route from the **Route manager** list.
2. Select **Tools** > **Clear data** to clear the measured data in this Route.
3. A pop-up window with a numeric keypad will open. Enter the code shown in the title bar to confirm the data clear.

- **Sensor Manager** – With Sensor manager, you can add, delete, or edit sensors you use with the data collector. Select **Tools** > **Sensor Manager** to display the Sensor Manager window.

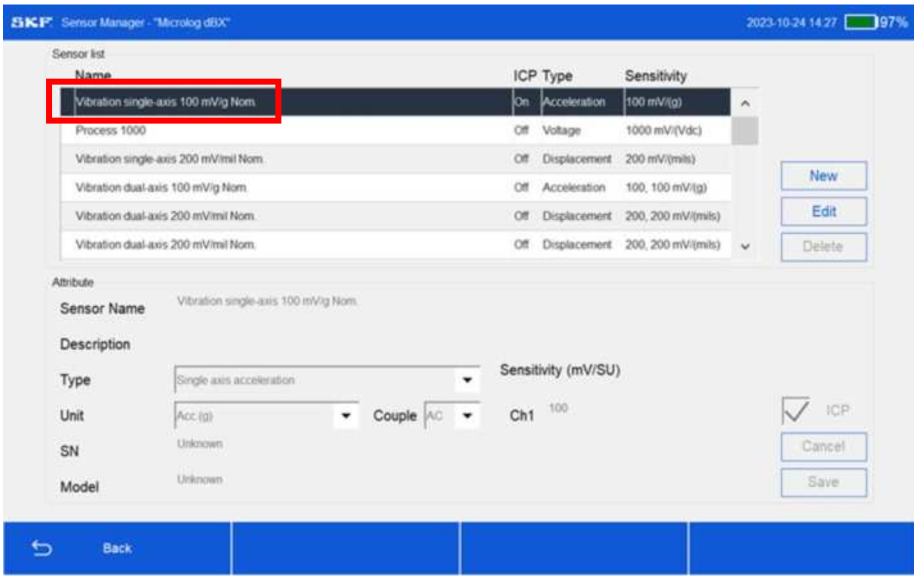
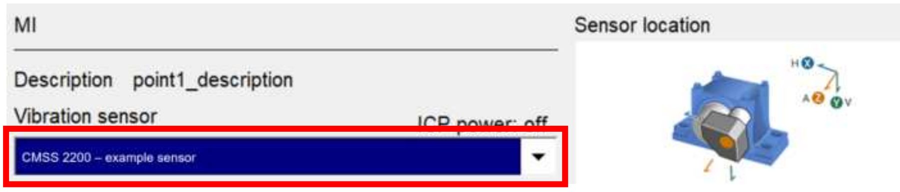


Figure 13 Sensor manager – example

- **Sensor List** – The upper part of the window shows the **Sensor List**, which contains the available sensors in this Route.
  Note that there are two categories of sensors. The first category includes the sensors from the Route, which are Route dependent and can only be edited but not deleted. The second category includes the custom-built sensors in this Data Collector, and they can be used by all Routes. The name of a custom-build sensor is marked with an asterisk (*) sign.

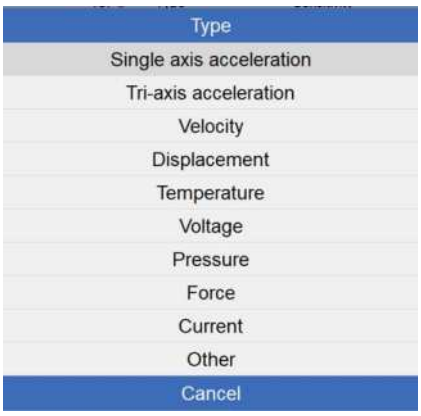
- **Attribute** – Select a sensor to view the sensor attributes under **Attribute** in the lower part of the screen.

- **New** – Tap **New** to add a new sensor to the list, then fill in the sensor information into each column of the **Attribute** section.
  The sensor information includes:
    - **Sensor Name** will only be shown in the sensor selection menu of a measurement Point.



    - **Description** – Fill in a description of the sensor when necessary.

    - **Type** – Select a sensor type from the pop-up menu.



    - **Unit** – Select the unit of the sensor. The available sensor units depend on the sensor type you have selected. The sensor unit will be used to specify the sensitivity as mV / (Sensor Unit).

    - **Couple** – Select **DC** or **AC** couple for the sensor. The AC couple tends to remove the DC offset of the measured signal, while the DC couple retains the DC offset. For example, if the shaft centreline is measured from an Eddy Current probe, select DC couple to retain the DC value.

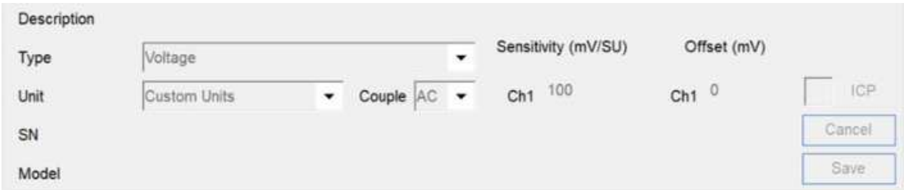
    - **Sensitivity (mV/Unit)** – Fill in the sensor's sensitivity value(s) mV / (Sensor Unit)

    - **SN** – Fill in the sensor's serial number

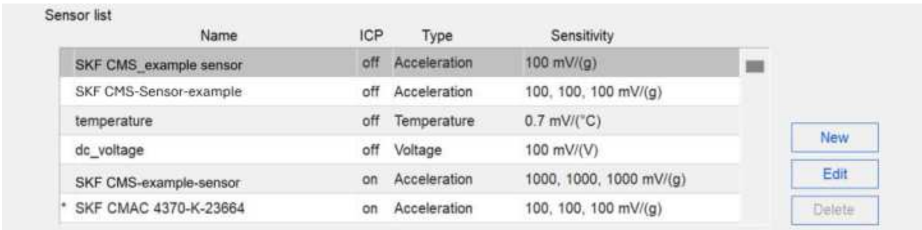
    - **Model** – Fill in the sensor's model name

- **Offset** – Fill in the offset value of a DC couple sensor in **mV**. The offset value defines the sensor's zero point. For example, if the offset is 1000 mV, then a measured value of 1000 mV corresponds to 0 Sensor Units.

- **ICP** – enable or disable the ICP power. If the ICP power is checked, the data collector will provide the ICP power to the sensor when measuring a signal.



Select **Save** to save the new sensor to the list or select **Cancel** to abort the procedure.



- **Edit** – Select to edit the attributes of the selected sensor. Click **Save** to save the changes.

- **Delete** – Select to delete the selected sensor.
  Note that those sensors created from the TCT software can't be deleted. Only the sensors created in the Data Collector can be deleted.

Tap **Back** to exit the Sensor Manager view.

Tap **Exit** to exit the Data Collector module.

2.2.5 Version information

Tap the information icon  at the top right corner to display the version information.

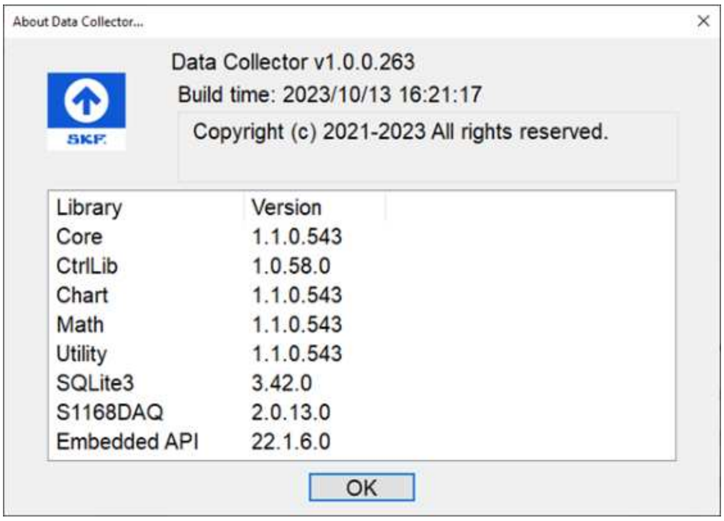


Figure 14 About window – example

2.3 Navigating a Route display

After you select and enter a Route from the Route Manager view, you'll see the Route display.
The Route display shows the Route hierarchy, node information, note table or the history data table. Use the navigation keys to move the cursor to a selected node.

2.3.1 Route Display of a Plant, an Area, or a Train node

When the cursor is on a plant, an area, a train or a machine, the display will show the Route hierarchy on the left side, the node information on the middle and the Notes table on the right side.

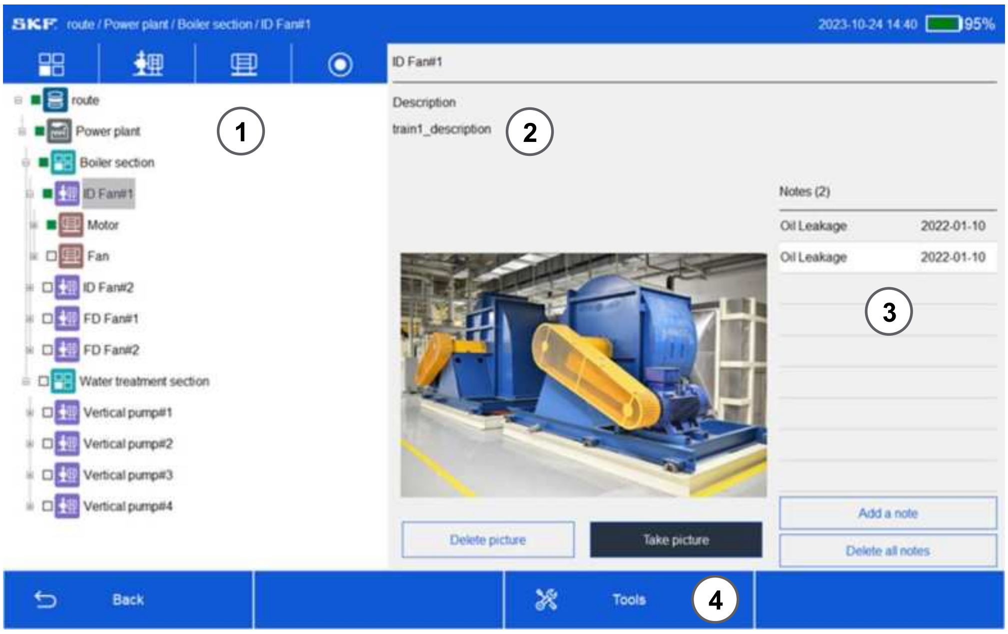


Figure 15 Display of a route when the cursor is in a Plant, an Area, or a Train

1. Route hierarchy
2. Node information
3. Notes table
4. Menu buttons

2.3.2 Route display of a Machine node

When the cursor is on a machine, the display will show you the Route hierarchy on the left side, the node information on the middle, the machine attribute (RPM, Pulses/Rev and NFC code) and the Note table on the right side.

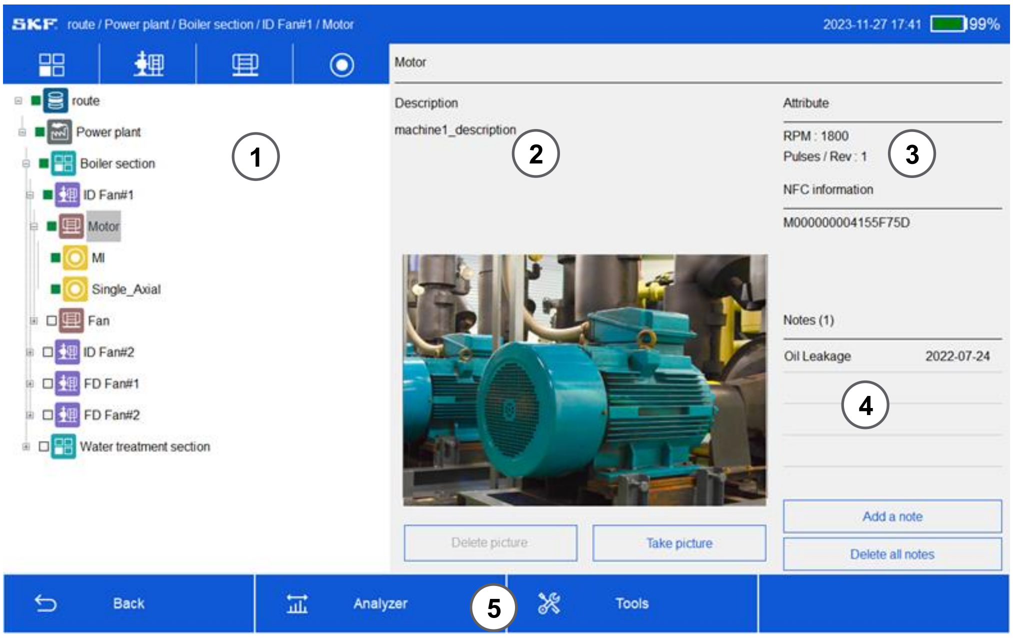


Figure 16 Display of a Route when the cursor is on a Machine.

1. Route hierarchy
2. Node information
3. Machine attribute (Rated speed, NFC information)
4. Notes table
5. Menu buttons

2.4 Working with the Route hierarchy

Use the navigation keys or the touch screen to move the cursor on the Route hierarchy to select a node and display the node property or perform measurements.

Microlog dBX provides only four levels of hierarchy in Data Collector:
Plant, **Area**, **Train,** and **Machine**.

If a Route contains machines with greater hierarchy levels than four, the host software will automatically use the lower four levels and omit the higher ones to preserve the hierarchy structure closest to the machine level.

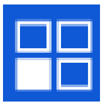
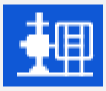
Sub-machines (@ptitude Observer only)
Microlog dBX does not handle sub-machines (machines within machines) since machines in the Microlog dBX may only contain Points, not other machines.

To bypass this, the host software will take all Points from sub-machines and program them as immediate children of the parent machine. Each Point will include the name of the parent sub-machine in its name.

This modification does not affect the host software - all measurements collected for Points within sub-machines will be uploaded to the correct location.

2.4.1 Expand or collapse a selected hierarchic level

Tap an icon on top of the Route hierarchy to expand or collapse the desired hierarchic level.

	Expand or collapse Area level.
	Expand or collapse Train level.
	Expand or collapse Machine level.
	Expand or collapse Point level.

With these icons of hierarchic levels, you can fast find an area, a train, a machine, or a Point from the Route hierarchy.

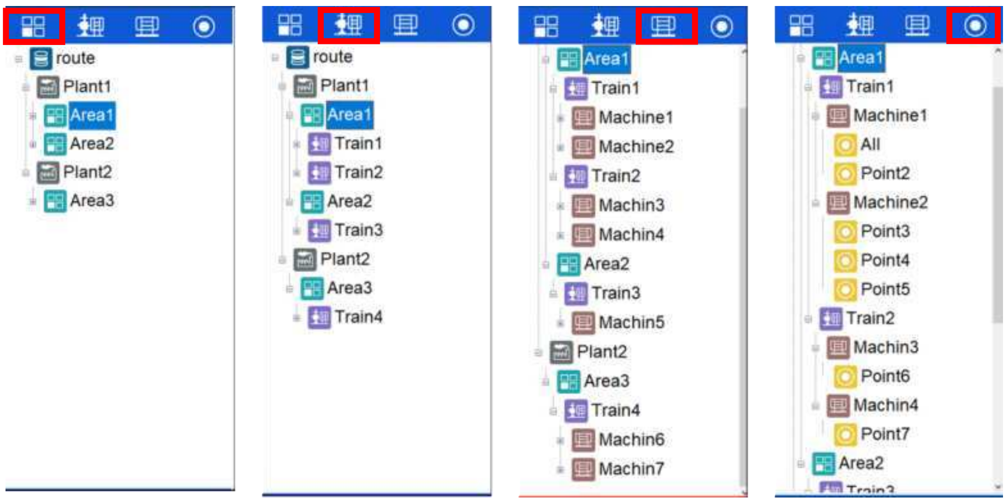


Figure 17 From right to left: Area, Train, Machine and Point level

- **Expand a node** – Move the cursor to a node which is indicated by a "**+**" icon and press the right navigation key to expand a node. To move to the next node, press the right navigation key again.
- **Collapse a node** – Move the cursor to a node which is indicated by a "**-**" icon and press the left navigation key to collapse a node. To move to the previous node, press the left navigation key again.

2.4.2 Read the alarm status from the Route hierarchy

At the right side of each node in the Route hierarchy, there is a square indicator showing the alarm status of that node.

You can read the alarm status by the colour and form of the indicator:

- **White** ☐ – No data.
- **Green** ☐ – All the latest measured data in this node or all its child nodes are in normal status. Note that if the measured data is not set with alarm level(s), it will be shown with a green indicator.
- **Green** ▲ – Skipped machine.
- **Yellow** ☐ – At least one of the latest measured data in this node or all its child nodes is in alarm 1 status.
- **Red** ☐ – At least one of the latest measured data in this node or all its child nodes is in alarm 2 status.

2.4.3 Alarm types

The Data Collector module supports the detection and display of various alarms:

- Above alarm
- Below alarm
- In window alarm
- Out window alarm

The alarm status is displayed on a bar chart with different colours:

- Green – normal
- Yellow – alert
- Red – danger

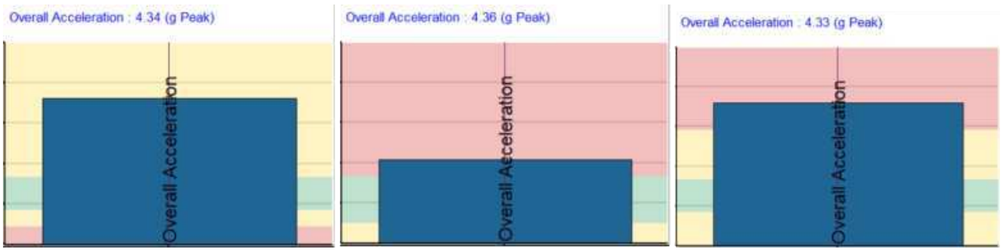
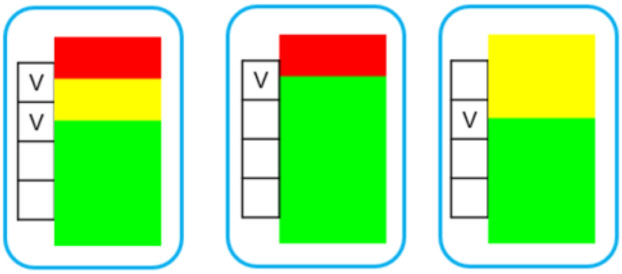
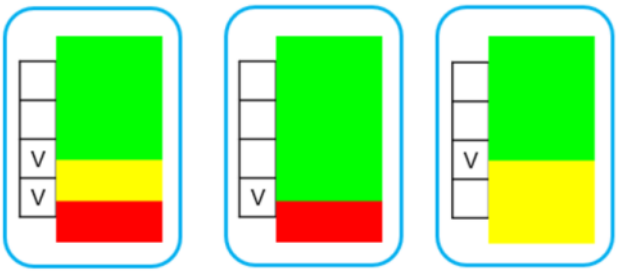


Figure 18 Bar plots with various types of alarms on the background

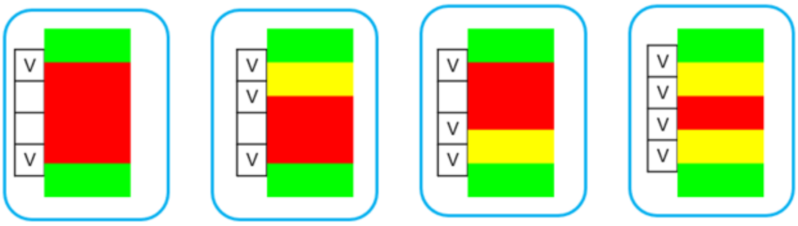
- **Above alarm** – A measured value higher than the alarm value(s) is detected as Alert (yellow) or Danger (red).



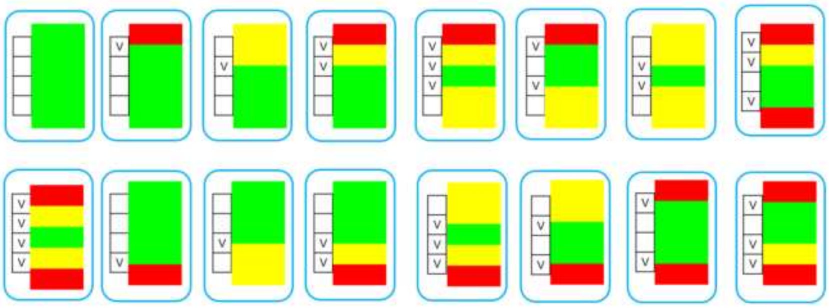
- **Below alarm** – A measured value lower than the alarm value(s) is detected as Alert (yellow) or Danger (red).



- **In window alarm** – A measured value inside the specified range is detected as Alert (yellow) or Danger (red).



- **Out window alarm** – A measured value outside the specified range is detected as Alert (yellow) or Danger (red).



2.4.4 Take a picture of a node

If there is a picture associated with the selected node, it will be displayed in the middle of the Route display.

Tap **Take a picture** to activate the camera on the back of your Microlog dBX, take a picture of the area, train or machine and save it to the database.
The saved picture files are in JPEG format. On upload of the Route, they will be saved to a folder on the host computer.

Note: If you delete a Route and upload it again, the saved images will be lost. To avoid this, do a Route reset instead. Route reset is possible from Microlog dBX and TCT.dBX.

Refer to the TCT.dBX Instruction Manual for more information on how to configure where photos and Non-Point notes will be saved.

2.4.5 Working with Note(s) at Non-Point nodes

When the cursor is on a Non-Point node, you can add a note or several notes with a piece of text or a picture to describe the abnormal behaviour of the machine.

- **Add note** – Add note to an area, a train, or a machine. Note is part of the archive data and is time dependent. A note is either pre-coded from the

@ptitude Analyst or created from the Data Collector module. You can add note(s) to a machine as a memo of the machine condition.

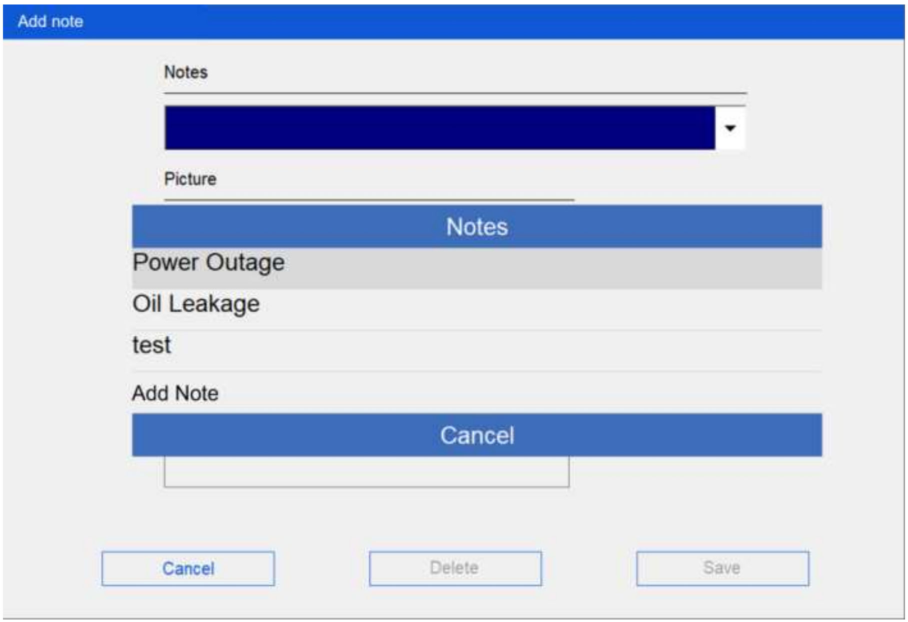


Figure 19 Add note

Tap **Notes** to display the list of pre-coded notes, move the cursor to select a coded note and save it. Or you can select **Add Note** to display the keypad and input a string to create a new note. The created Note will be saved to the note list for future use.

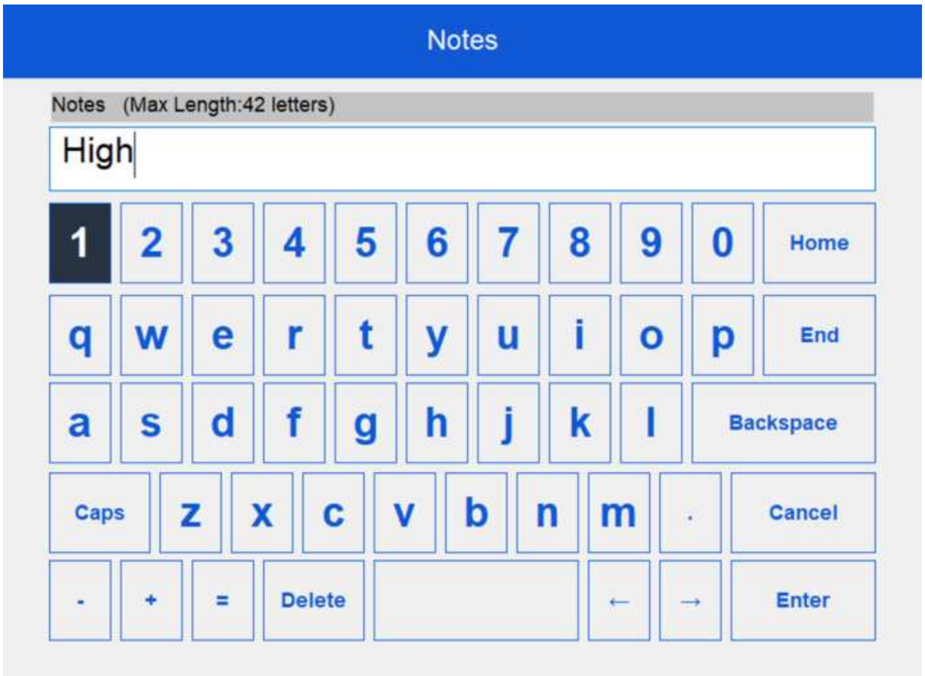


Figure 20 Add note keypad

Tap **Take picture** to activate the camera and take a picture. The picture will be saved as part of the note record. Tap **Delete picture** to delete a picture.

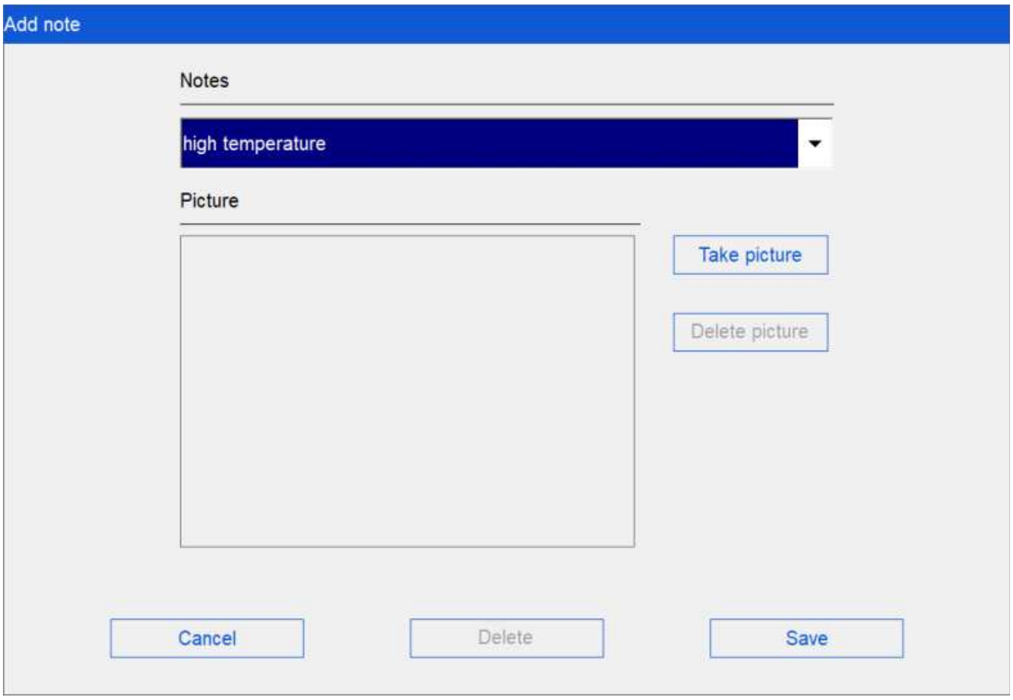
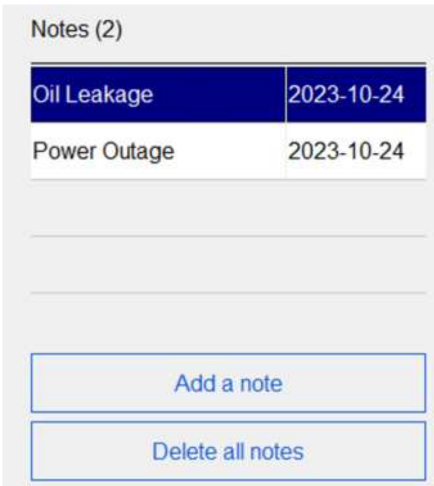


Figure 21 Add note

Tap **Save** to save the note and close the window.

- **Delete a note** – Tap the note to enter the edit note page and select **Delete**.
- **Delete all notes** – Tap to delete all saved notes.



2.4.6 Using the NFC reader

With the built-in RF identification using near-field communication NFC located on the rear, you can quickly locate a machine by scanning the NFC tag installed at the corresponding location.

Move the cursor to a machine to check whether there is an NFC code in the **Attribute** section.

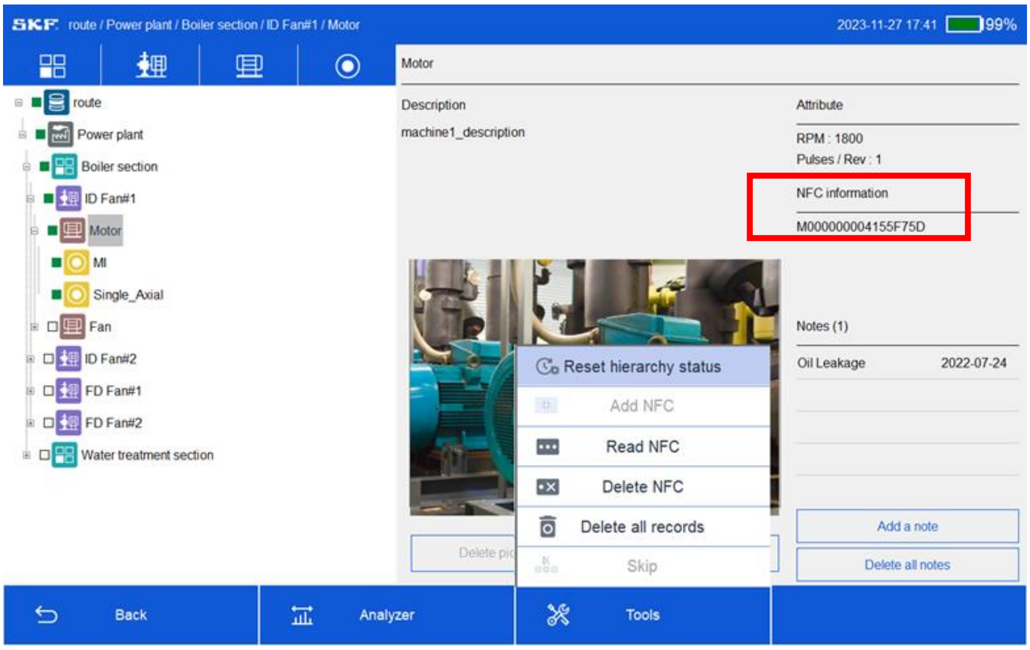


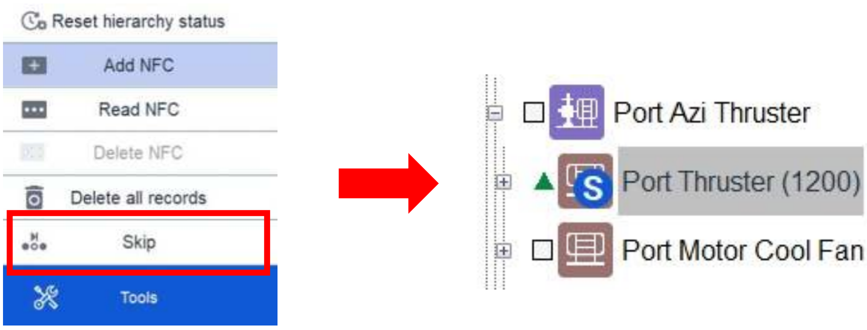
Figure 22 NFC information

If the NFC code is stored in the specific node, you can scan the corresponding NFC tag and move the cursor to that node. Select the following options from the **Tools** menu to work with the NFC feature:

- **Add NFC** – Add an NFC code to the specific node. Select this function to activate the NFC reader. Scan the NFC tag, which is installed to the physical location of the node and store the NFC code to the Route.
- **Read NFC** – Scan the NFC tag and move the cursor of the Route hierarchy to the corresponding node.
- **Delete NFC** – Delete the NFC code which is stored in this node.

2.4.7 Skip a measurement

Place the cursor on a machine node in the Route hierarchy and select **Skip** from the **Tools** menu to skip the measurement. It will prompt you to add a note for this skip action.



The skipped machine and all its child nodes will be marked with a triangle indicator to show the skip status. This function is usually used when the machine is not operating.

2.4.8 Route display of a Point

When the cursor is on a Point or a direction of a Point, the display will show you the hierarchic database on the left side, the Point and sensor information in the middle and the archive data table on the right side.

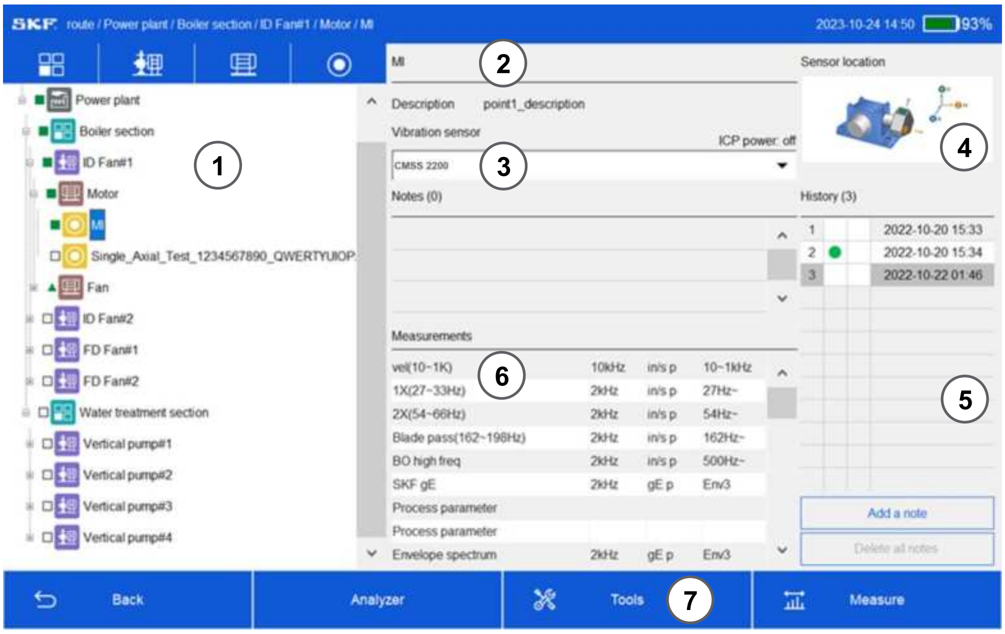


Figure 23 Display of a Route when the cursor is on a Point

1. Route hierarchy
2. Node information
3. Sensor selection
4. Sensor plot
5. History data table
6. Measurement panel, see MPA section
7. Menu buttons

2.4.9 Multiple Point Automation (MPA) and Route Points

Points downloaded in a Route which are members of an MPA (Multiple Point Automation) group will be displayed as a single Point in the Route hierarchy.

When you click **Measure** for a Point made from an MPA group, a raw data sample will be taken as normal, and then the raw data will be processed into measurements for all the Points in the group.

The Measurements panel in the middle of the Route screen will show the measurements for each of the Points that formed the group, and when data is collected, these will appear as separate plots in the measurements screen.

2.4.10 Working with Note(s) at a Point node

The note(s) of a Point is part of the data record.
As you move the cursor on the history data table, it will show you the corresponding record note(s) in the node information table in the middle of the display.

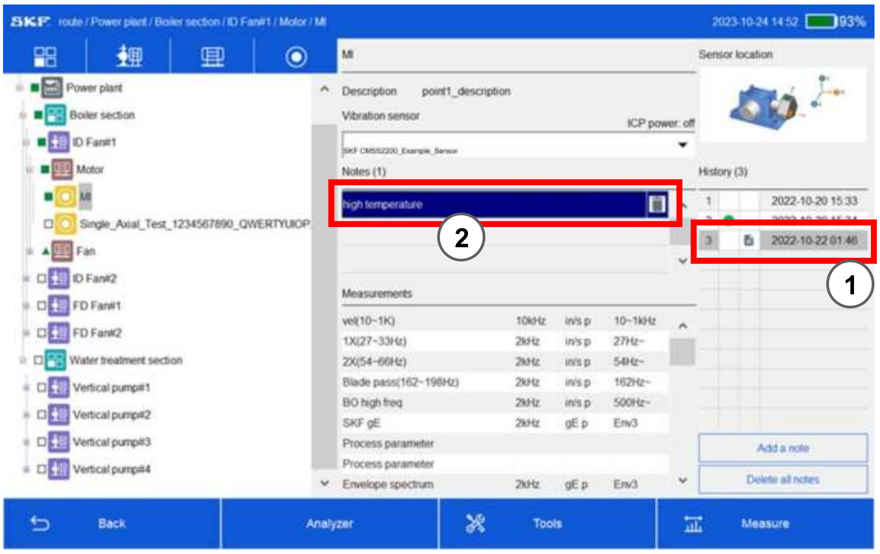


Figure 24 Notes – example

1. Note inside
2. Note content

Follow one of the methods below to add a note to a data record. See Working with Note(s) at Non-Point nodes for detailed description.

Add a note after a measurement

After taking a measurement and before saving the new record, you can add a note to it from the **Add note** menu.

Figure 25 Add note

Add a note to a selected record from the History data table
Browse through the **History** data table, select a record and tap **Add a note** to add a new note to this record.

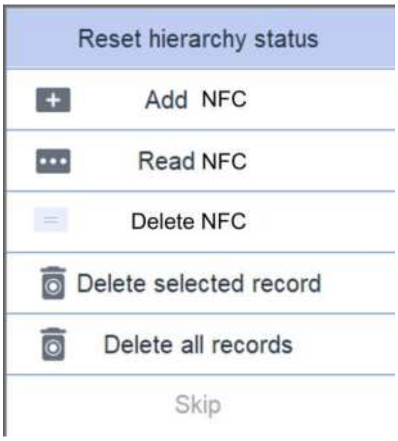
Tap **Back** to return to the **Route manager** view.

2.5 Data measurements

The following chapter describes how to install a sensor and how to take measurements within a Route.

2.5.1 Reset hierarchy status

When you start a new measurement task with a selected Route, select **Reset hierarchy status** from the **Tools** menu of the Route.



This option will display **Reset hierarchy status since YYYYMMDD** with a time stamp pre-set as the current date and time. You can also input a different time stamp manually if wanted.

Tap **Enter** to confirm the reset.

The hierarchy status of the data measured before the time stamp will be cleared.

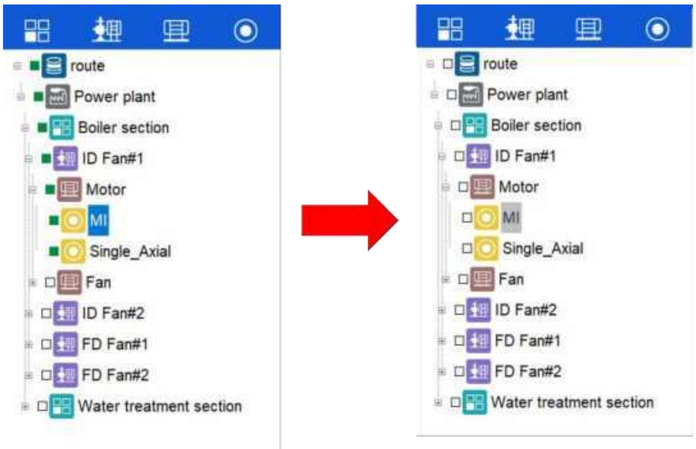


Figure 26 Reset hierarchy status

Note: It is recommended to reset the hierarchy status before performing a new measurement task to see if a Point is measured or not.

2.5.2 Sensor connection

Before taking a measurement, connect a suitable sensor to your Microlog dBX and install it in the correct position and direction.

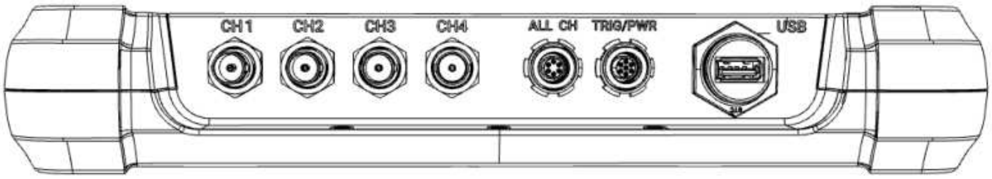


Figure 27 Back view connections

- **Uniaxial accelerometer** – Connected to Channel 1 (Connect to the All CH Fischer connector with a proper cable/connectors).
- **Dual axis sensors** – Always connect to Channel 1 and Channel 2.
- **Triaxial accelerometer** – Connected to Channel 1, 2 and 3 for measurements of axial, horizontal and vertical directions. (Connect to the All CH Fischer connector with a proper cable and connectors).
- **DC sensor** – Connected to channel 4. (Connect to All CH Fischer connector via an appropriate cable).
- **Speed sensor** – For AC type speed sensor, always connect a speed sensor to the PWR/TRIG connector with a proper cable/connectors.
For the DC-type speed sensor, connect to Channel 4.

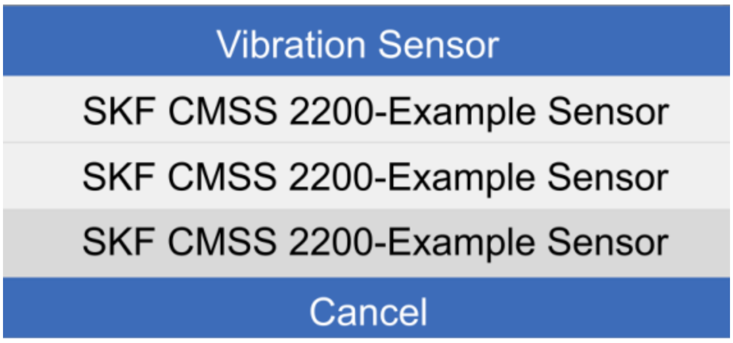
- An AC type sensor generates a pulse per revolution signal. It includes the signal from an SKF laser tachometer or an ECP sensor (Eddy current probe).
  Note: If the peak value of the speed signal is lower than 2.5 V, it will not trigger the speed measurements.

2.5.3 Sensor selection

Select a sensor from the sensor selection menu before performing a measurement.

Tap **sensor selection** to display the list of available sensors and use the navigation keys to select one.

Note: Only sensors of the same type will be displayed in the list.



2.5.4 Sensor types and sensor installation

DC sensor
A DC sensor Point can be identified from the illustrating icon at the up-right corner. The DC-type signal includes temperature, pressure, flow, voltage, current and others. It is always connected to Channel 4 from the BNC connector or the All CH connector with an appropriate cable.

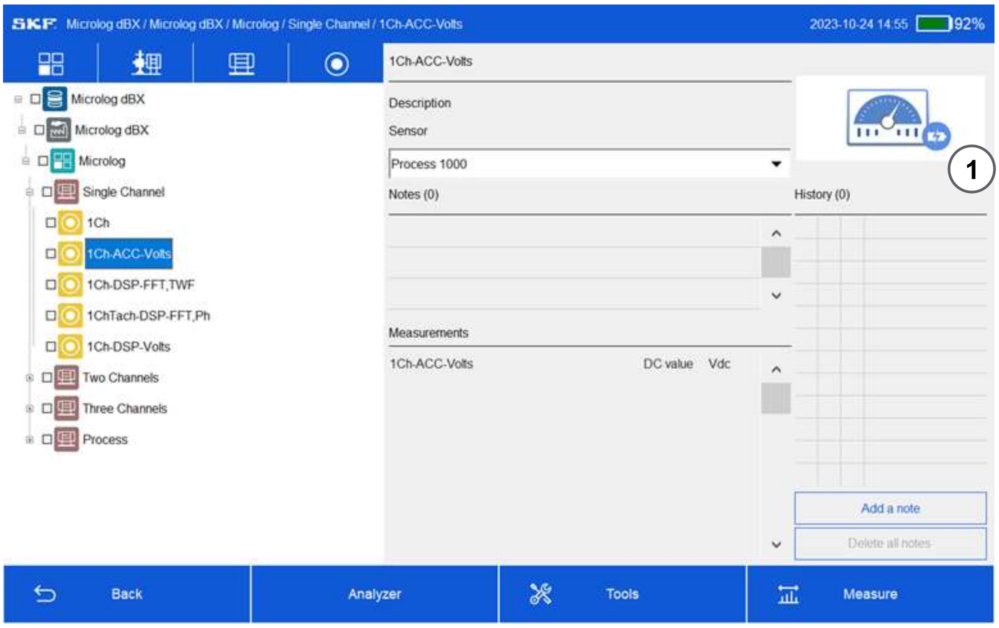


Figure 28 Process type – example

1. Plot illustration of a DC sensor

Manual parameter

For a manual parameter Point, you will need to input a numerical value manually after you tap **Measure**. The manual parameter Point can be identified from the keypad icon at the up-right corner.

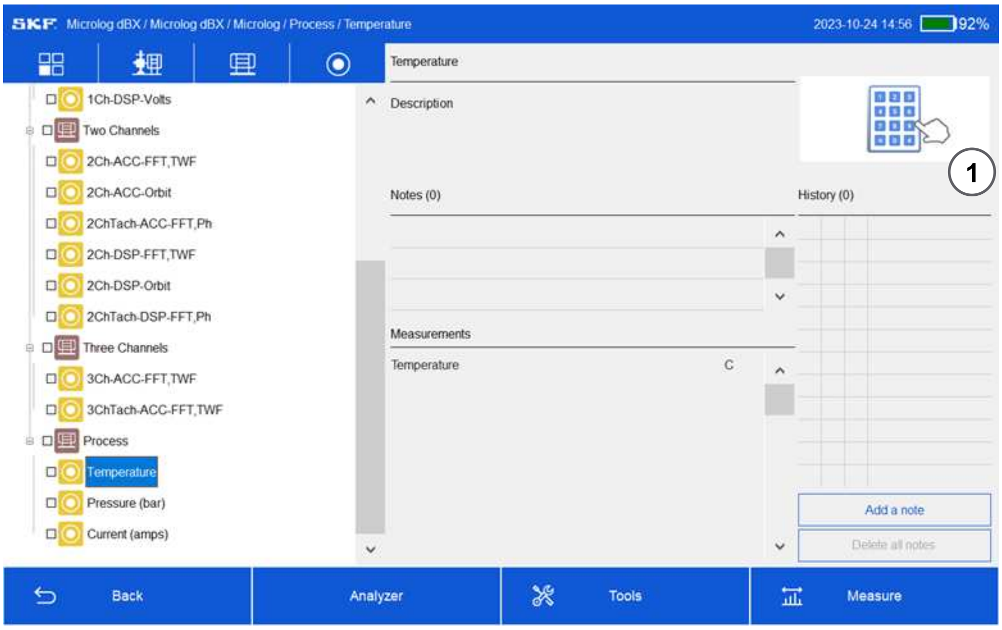


Figure 29 Process types – example

1. Manual parameter input – example

Uniaxial sensor

A uniaxial sensor contains neither location nor directional information inside. Usually, the information on the sensor location and direction is indicated by the name of the Point.

For example, a Point named MOV may indicate that the sensor is installed in the vertical direction of the Outboard Point of a Motor.

You can tell whether it is a Uniaxial Point from the plot illustration in the up-right corner.

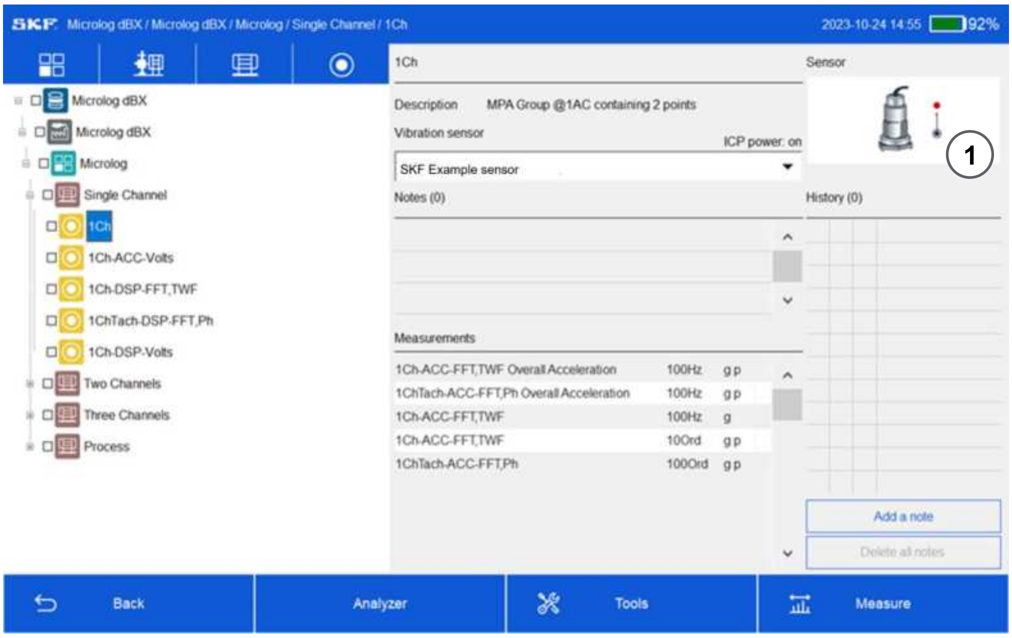


Figure 30 Process FFT – example

1. Plot illustration of a uniaxial sensor

Dual axis sensor

When a dual axis sensor is used, it will show a plot illustration of the dual axis sensor in the top right corner. A dual axis sensor is a pair of sensors with which you can measure the Orbit of a rotating shaft in addition to the other general functions.

When a Point is set up to measure Orbit, the locations of the two sensors will be shown at the top of the sensor plot illustration.

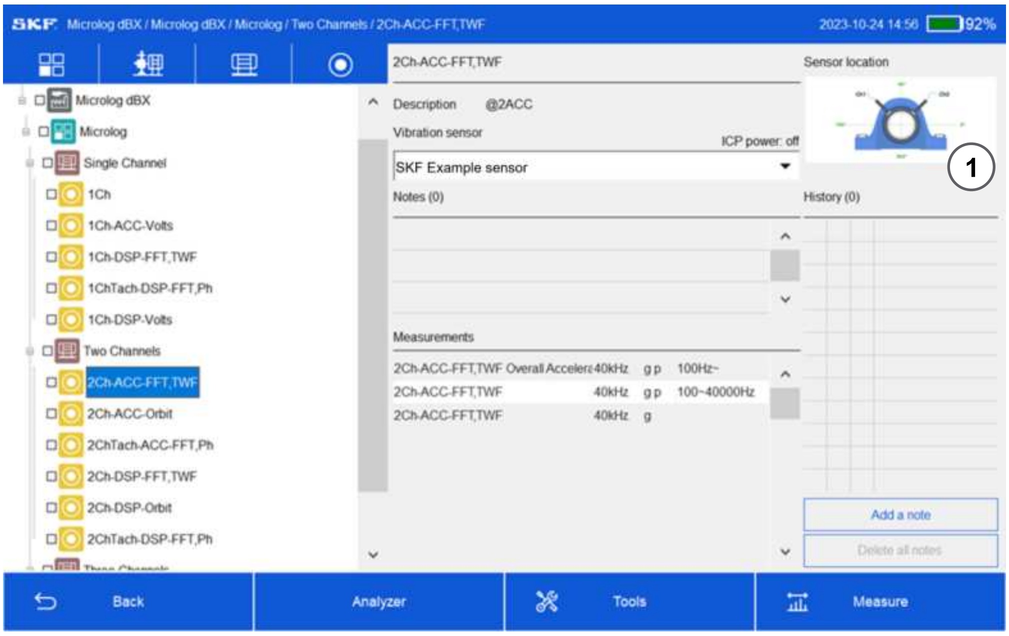


Figure 31 Dual channel – example

1. Plot illustration of a dual axis sensor

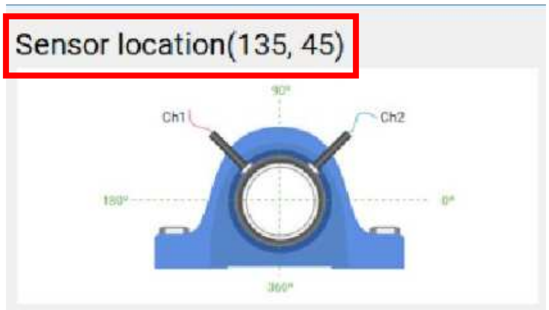


Figure 32 Locations from a dual axis sensor used for orbit measurements

Triaxial accelerometer

When a triaxial sensor is used, it will show an illustrating plot of a triaxial sensor in the top right corner. A triaxial sensor contains location information and direction because the measured data from the 3 axis needs to be assigned to horizontal, vertical, and axial directions in a proper way.

You can always follow the default setting to install the sensor. In case the sensor installation is not possible due to some restriction, you can change the sensor installation to match your way of installation.

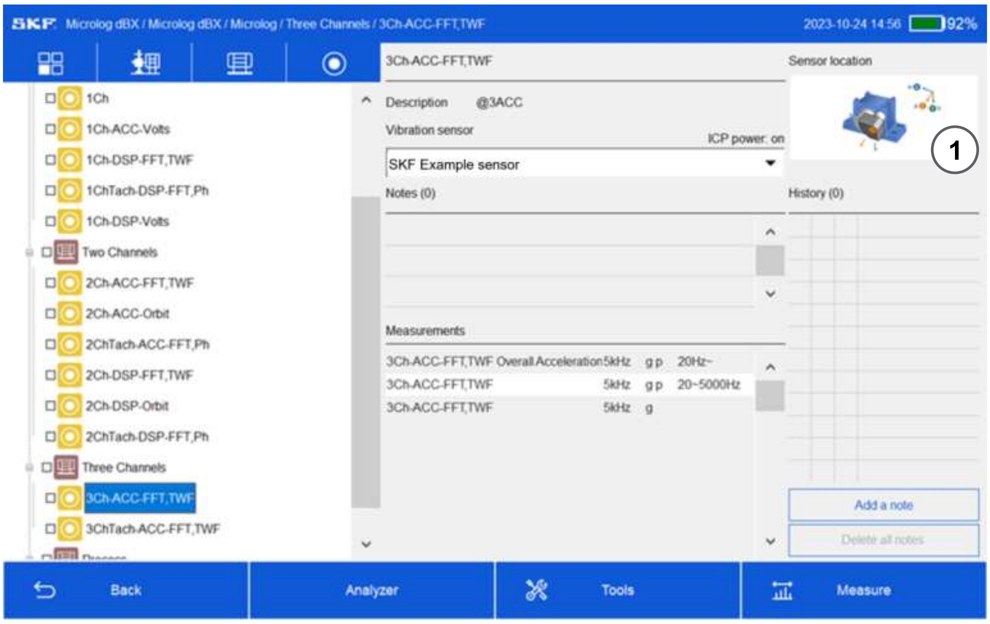
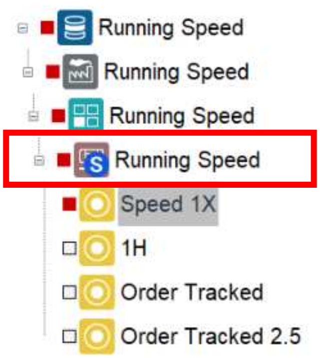


Figure 33 Plot illustration of a triaxial sensor

2.5.5 Speed measurement

When a machine node is marked with an **S** symbol, that implies there is a speed Point(s) under this node. Measure the speed Point(s) before measuring the other Points.



2.5.6 Measure speed from an AC-type sensor

For a speed Point which measures the speed from an AC-type sensor, such as an SKF laser tacho sensor or an ECP sensor, it will show the following display.

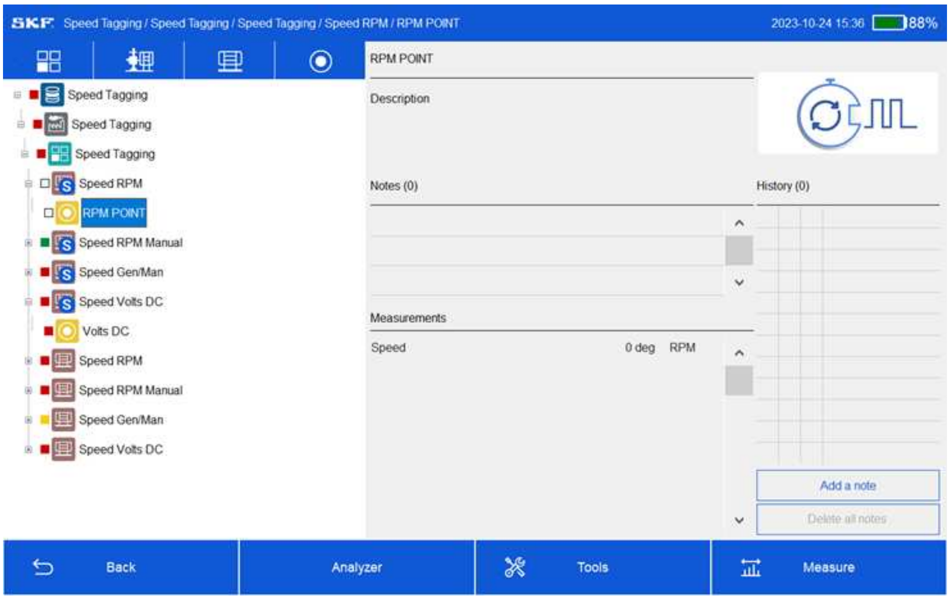


Figure 34 Speed tagging from AC-type sensor – example

Note that the AC-type speed signal is always connected to the PWR/TRIG connector. Move the cursor to the speed Point under the machine and press the **Start** / **Stop** key to display the speed measurement window.

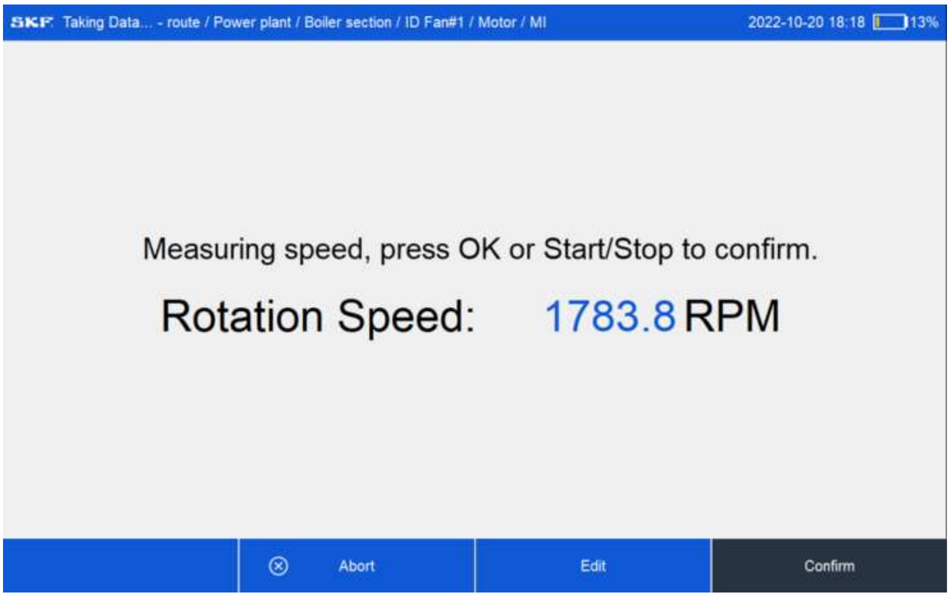


Figure 35 Measuring speed – example

Tap **Confirm** or press the **OK** key to complete the speed measurement.

Tap **Abort** to abort the measurement. You can also select **Edit** from the speed measurement display to enter a speed value manually.

2.5.7 Measure speed from a DC-type speed sensor

If a Point is set to measure the speed via a DC-type speed sensor, it will show the following display.

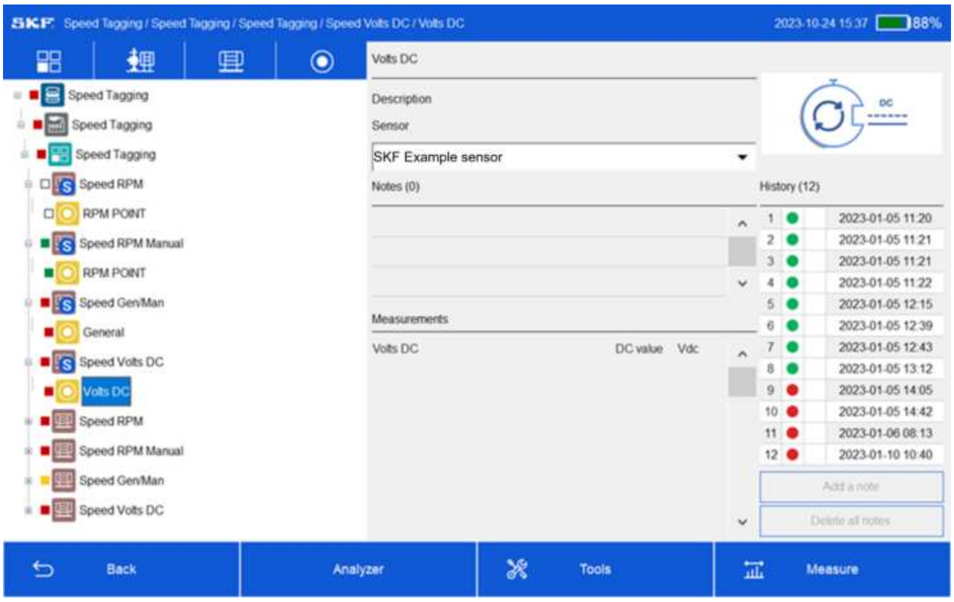


Figure 36 Speed tagging from DC-type – example

Connect the signal to Channel 4 and tap **Measure** to start the measurement.

2.5.8 Measure speed via manual input

If a Point is set to measure the speed via manual input, it will show the following display.

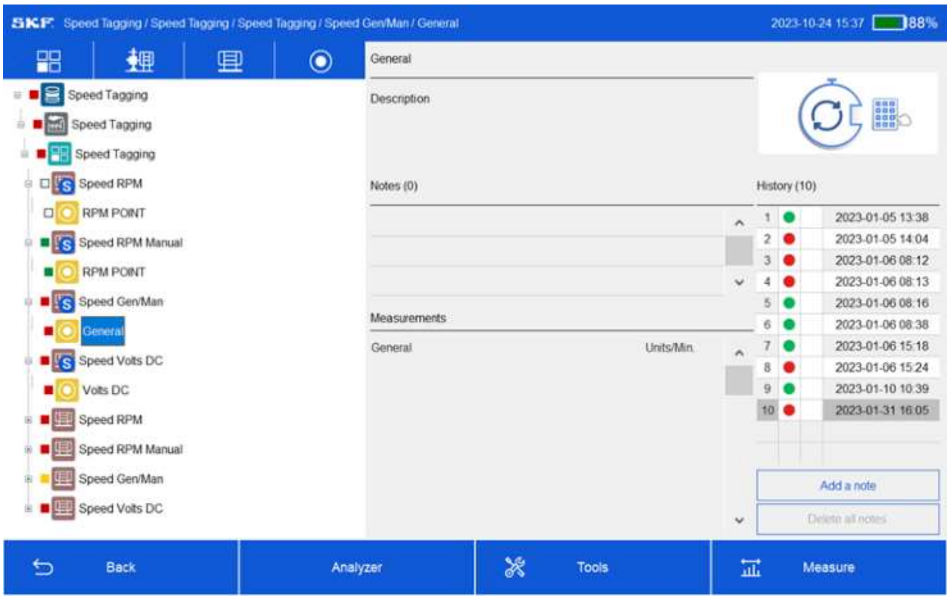


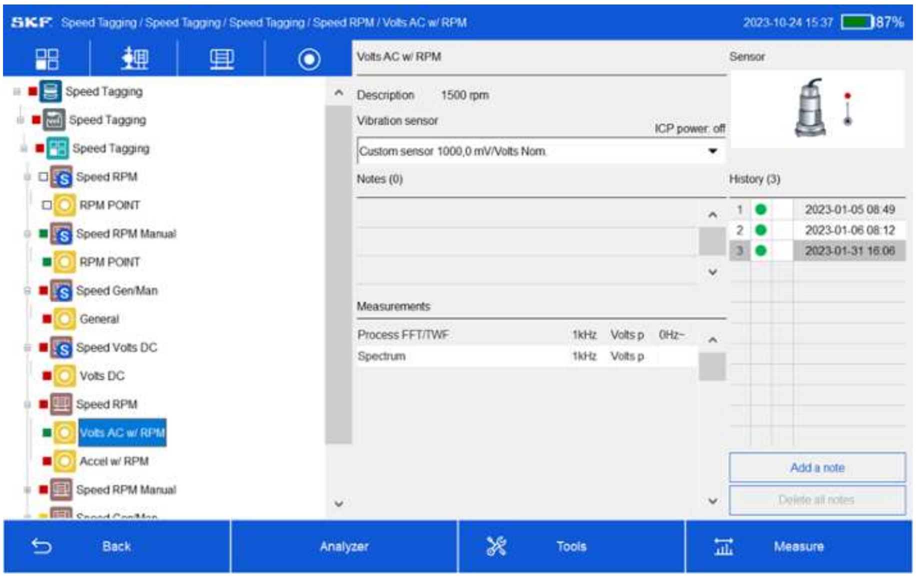
Figure 37 Measure speed – manual input – example

Tap **Measure** and enter the speed value in the numerical keypad located in the up-right corner of the display.

2.5.9 Measure speed using speed tagging

If a speed Point is set as speed tagging, it will measure the vibration spectrum from which you can tag the 1X as the rotation speed.

1. Connect the vibration signal to Channel 1 and then tap **Measure** to start the measurement.

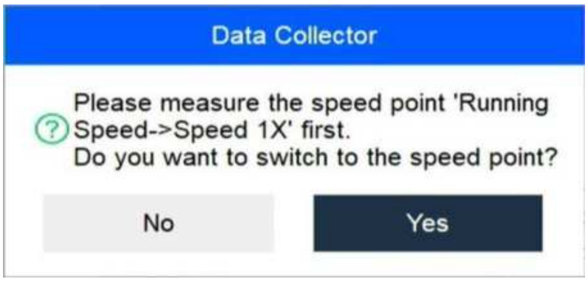


2. After the vibration measurement, the cursor of the spectrum plot will stay at the maximum peak. Move the cursor to the 1X position if the 1X is not at the maximum peak and tap **Save speed** to save the cursor value as the speed.

2.5.10 Speed ratio and ratio speed

When the speed Point(s) is measured, the speed value will be applied to other measurement Points of the same train by multiplying the pre-set speed ratio to generate the ratio speed(s). Note that the speed Point(s) always need to be measured first.

Otherwise, it will prevent you from measuring other points and display the following warning message.



2.5.11 Rotation speed

The rotation speed of a machine is presented in the top left corner of a spectral plot or a waveform plot.

Figure 38 Rotation speed of a machine

The speed value may come from three sources:

- **Rated speed** – The rated speed is part of the machine attributes, which are defined by the host software. If there is no measured speed for the machine, the rated speed will be shown at the speed corner.
- **Measured speed and ratio speed** – The measured speed(s) is derived from the measurement of a speed Point. The other measurement Points under the same machine train can convert it into different speed values by applying the speed ratios. This feature makes it easy to derive individual speed values when there is a transmission mechanism in a machine train. When a Point contains a measured speed or ratio speed, it will replace the rated speed.
- **Tagged speed** – To tag a speed value, move the cursor to a selected frequency on a spectral plot and select **Set as 1X** from the **View** menu. The tagged speed value is used for that measurement Point and applies to other measurement functions under this machine. If you select **Clear 1X** from the **View** menu, the speed value will restore to the rated speed.

2.5.12 Order tracking Points

If a Point contains order tracking measurements, it will require you to measure the speed with an AC-type speed sensor first.
The measured functions derived from the order tracking algorithm include **Order spectrum**, **Phase spectrum**, **Amplitude**, **Phase** and **Orbit**.

Alternatively, if an AC-type speed signal is not available, but the reference speed is measured from a speed tagging Point in advance, the software will conduct an order normalisation algorithm to provide an order spectrum with no phase information.

If neither the AC-type speed sensor nor the speed tagging Point is available, the order tracking measurement will be aborted.

2.5.13 Acquisition of raw data

Move the Route hierarchy cursor to a measurement Point and press the **Start** / **Stop** or the **Measure** key to start a measurement.

2.5.14 Measurement modes

Auto

If the measurement mode is set as **Auto** from the Configuration window, the measurement will start right after the **Start** / **Stop** key is pressed. The Data Collector module will acquire the raw signal and show it with waveform plots and percentage status of the measurement progress.

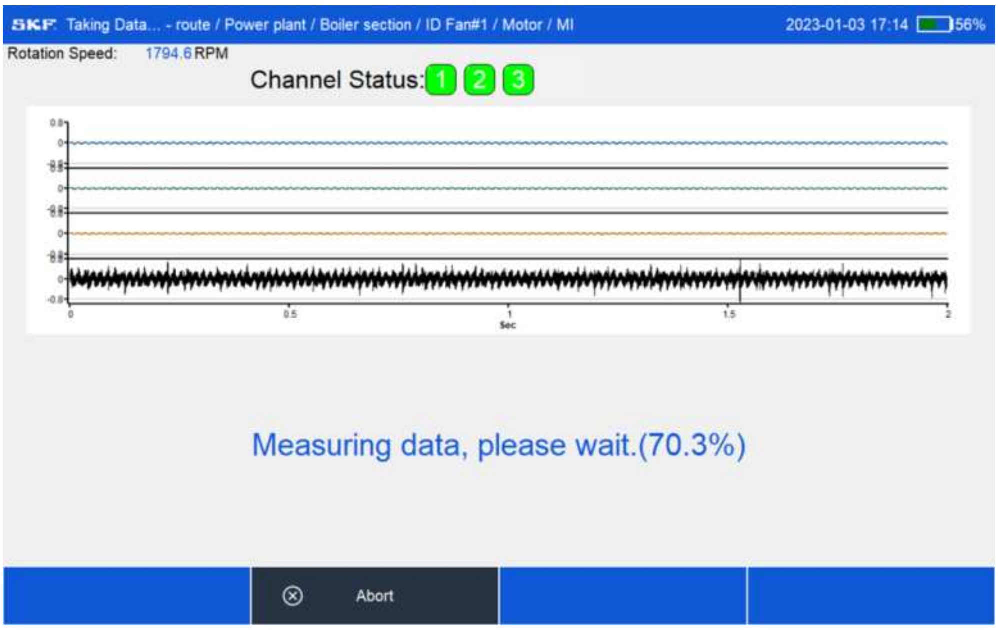


Figure 39 Measurement example – Auto

Manual

If the measurement mode is set as **Manual** from the Configuration window, it will display a live view of the measured signal after the **Start** / **Stop** key is pressed.

Note that the measurement will not start until the **OK** key or **OK** button on the screen is pressed. Using the manual mode, you can observe the signal before starting a measurement.

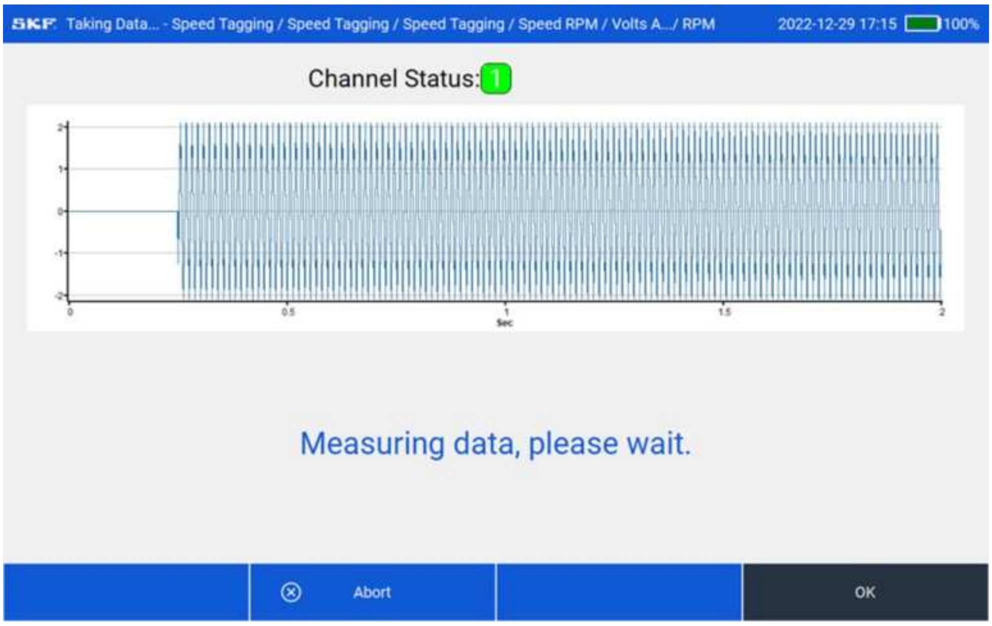


Figure 40 Measurement example – Manual

2.5.15 During data measurement

During data measurement, the time waveform is displayed with the progress percentage number at the bottom.

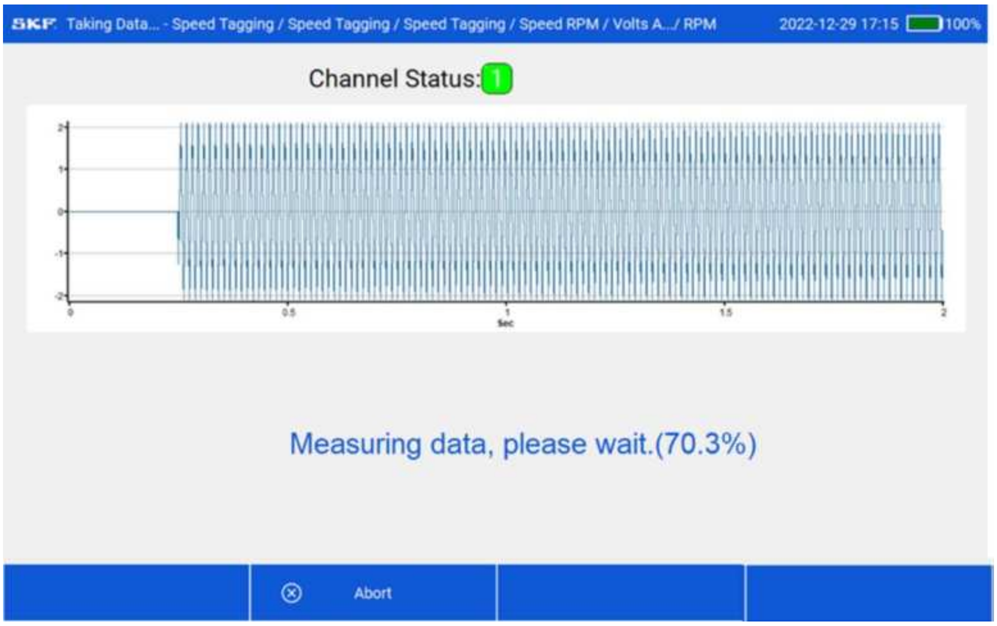


Figure 41 Single axial measurement

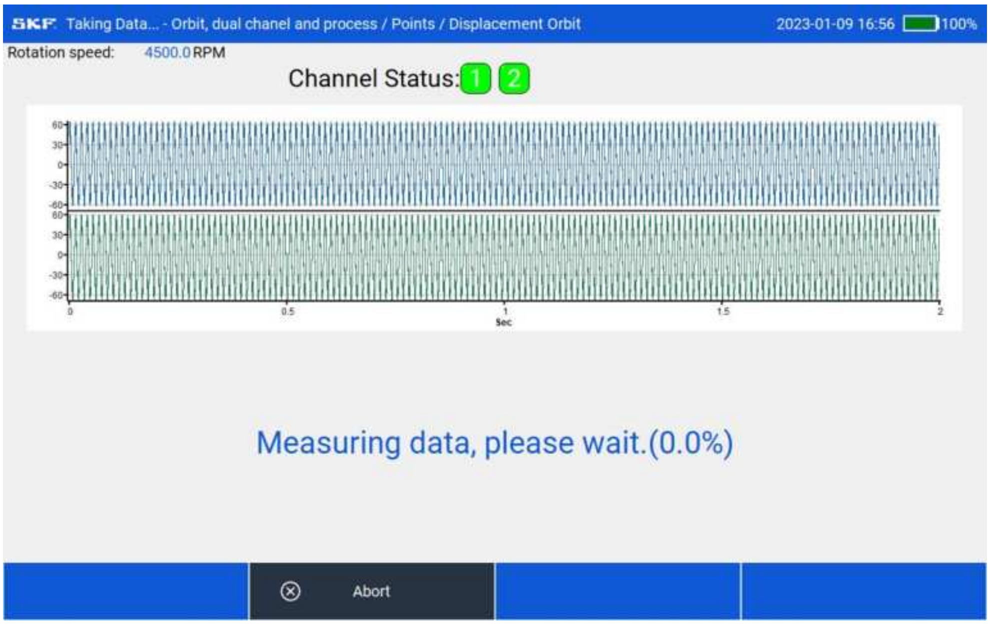


Figure 42 Dual channel measurement

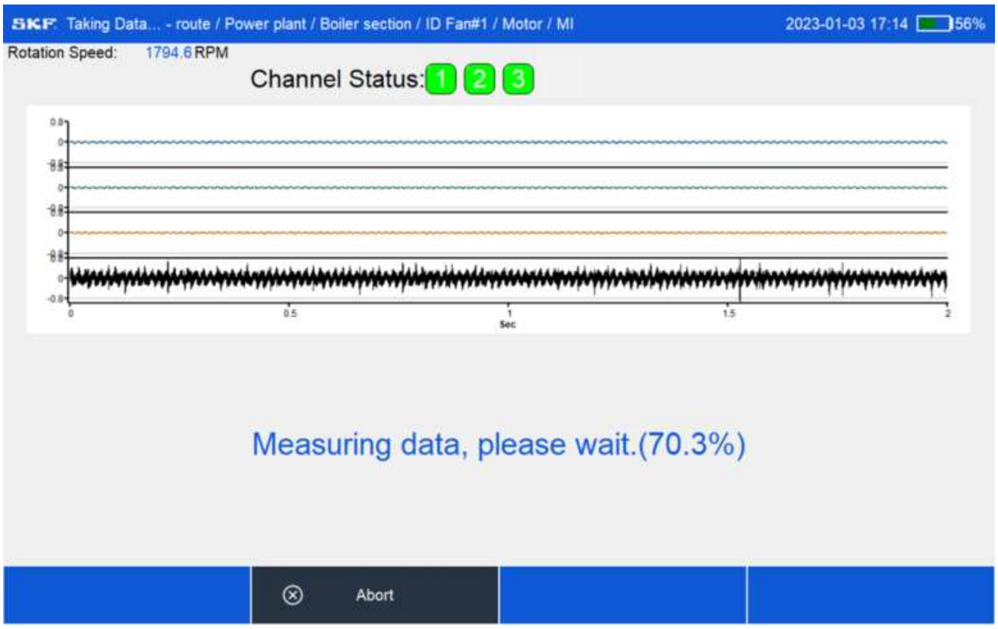


Figure 43 Triaxial measurement

2.5.16 Channel status during a measurement

The channel status is shown on top of the display with channel status icons. The channel status shows the channels which are in use:

Channel Status: 1 2 3	3 channel measurement
Channel Status: 1 2	Dual channel measurement
Channel Status: 1	Single channel measurement

2.5.17 Measurement abort

When an abnormal ICP bias or overloaded signal is detected during the measurement, the software will abort the measurement automatically and display a message showing the errors.

Figure 44 Overloaded signal – example

2.5.18 Complete a measurement

The time it takes to complete the raw data acquisition depends on the measurement's setup. For example, it takes around five seconds to complete a measurement with the following setup:

- Power spectrum x1
  - Bandwidth = 2 kHz
  - Resolution = 3200 lines
  - Average = 5 times, with 50% overlapping
- Time waveform x1
- Overall velocity x1
- gE (true peak-to-peak) x1
- gE envelope spectrum x1
- Band overall x5

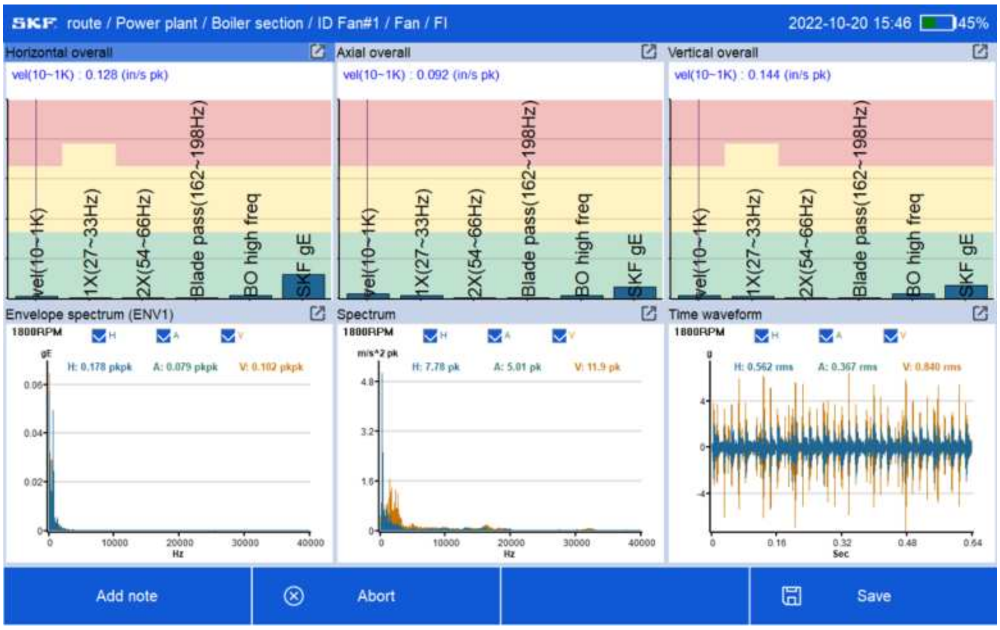


Figure 45 Measurement example

After the raw data acquisition is completed, it will be converted into various pre-set functions in the Route.

Tap **Save** to complete this measurement or select **Abort** to abort this measurement. To add a note to this record, tap **Add note** before saving it.
When the measurement results in more than six plots, tap **Next** to enter the next page for more information.

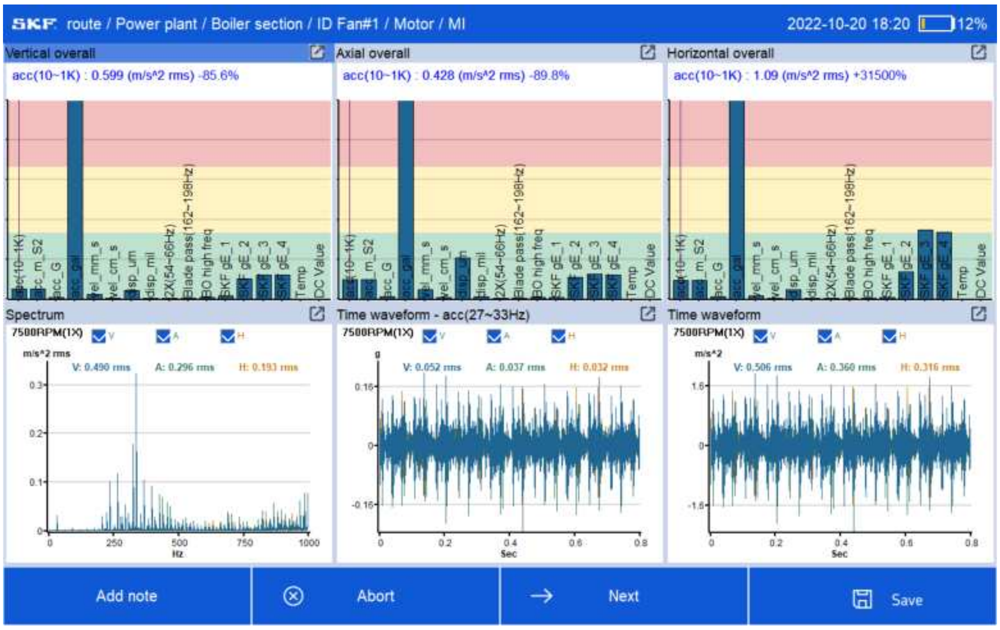


Figure 46 Measurement example

2.5.19 Manual parameter

The manual parameter must be entered manually. Tap the input field to display the keypad and input the data.

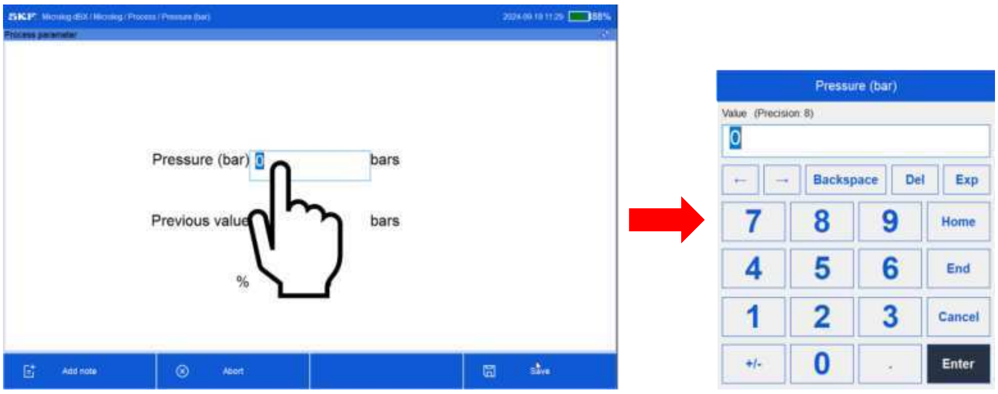


Figure 47   Manual parameter input

2.6 Non-Route measurements

The Non-Route measurement feature allows you to take a measurement which is not previously defined in the Route and downloaded to the Microlog dBX. The Non-Route measurement is useful when additional measured data is required to perform further analysis of the root cause of a vibration problem.

2.6.1 Activate a Non-Route measurement

Move the cursor of the route hierarchy to a machine or a Point, and the Analyzer button will display on the main menu. Tap the Analyzer button to activate a Non-Route measurement for that node.
Note that the hierarchy information of the selected node, either a machine or a Point, will be saved together with the measured data. Ensure the cursor is at the correct node where the Non-Route measurement will be taken.

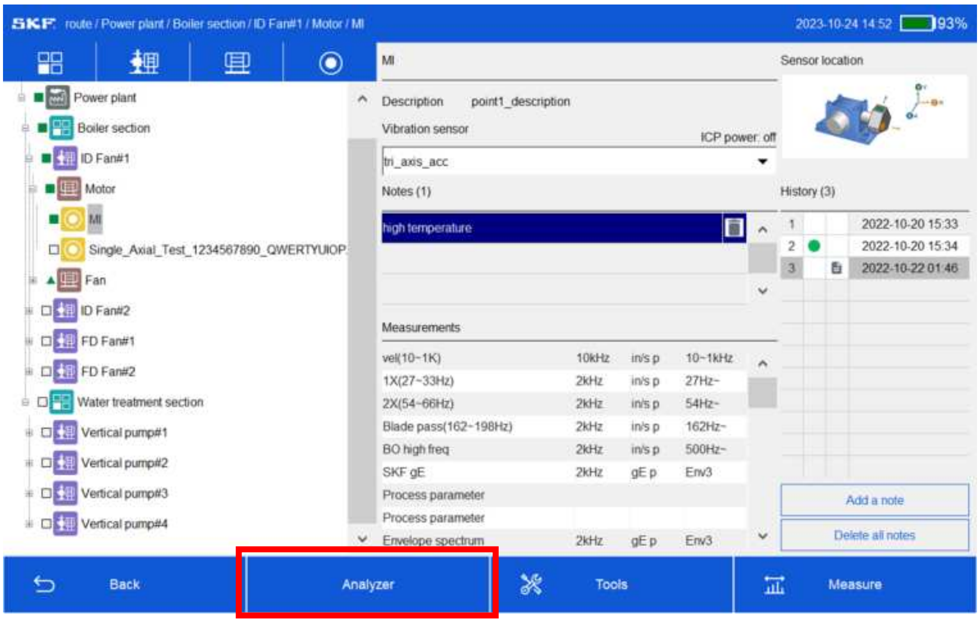


Figure 48 Analyzer menu button

Tap the **Analyzer** button, and the software will prompt a dialogue box showing that it will leave the Data Collector module and enter the Analyzer program for collecting a Non-Route measurement.

Tap **Yes** to enter the Analyzer module or select **No** to abort

2.6.2 Take a Non-Route measurement from a template

When the Non-Route measurement is activated, the software will prompt you to select a testing template or the Analyzer from the following display.

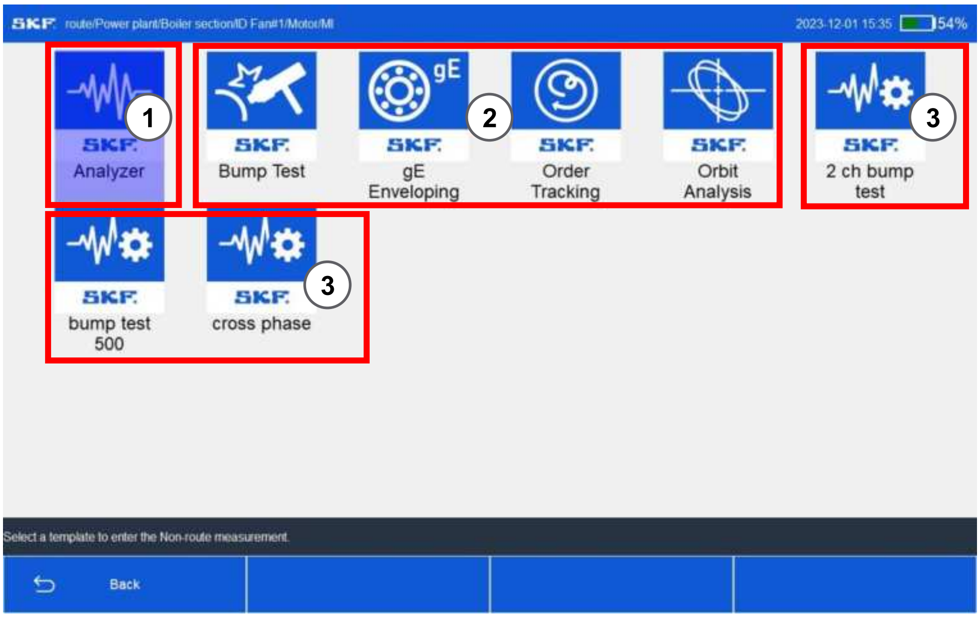


Figure 49 Templates for Non-Route measurements

1. **Analyzer** – Tap to enter the Analyzer module and take a non-template measurement.

2. **Standard templates** – Tap to enter the Analyzer module with pre-set parameters of a standard template.

3. **User-defined templates** – Tap to enter the Analyzer module with pre-set parameters of a user-defined template.

Select a standard or user-defined template to quickly perform a test with pre-set parameters.

2.6.3 Standard and user-defined test templates

- **Standard templates** – Provided by SKF and are pre-set with the most common measurement parameters for taking a measurement quickly. The standard test templates are shown with specific icons.

  The standard templates include **Bump Test**, **gE Enveloping**, **Order Tracking** and **Orbit Analysis**. For more information about these standard templates, refer to Perform test from a pre-set template for detailed instructions.

- **User-defined templates** – In addition to the standard templates, you can create and use your user-defined templates from the Analyzer module. These user-defined test templates are project files and are saved in the following folder: *D:\dBX\Microlog dBX\Analyzer\Data_Collector*.

As the Non-Route measurement is activated, the software will search for Analyzer project files in that folder and show them on the template display together with the standard templates. User-defined templates appear on the screen with the same icon.

2.6.4 Take a Non-Route measurement without template settings

If none of the templates are suitable for your non-route measurement, you can select Analyzer for taking a Non-Route measurement. Selecting Analyzer will run the Analyzer module with the default settings for a new project, and you'll need to go through the setup process to create a new test project. For detailed instructions, refer to the dBX Analyzer module chapter.

2.6.5 Complete a Non-Route measurement

After taking the Non-Route measurement from the Analyzer module,
select **File** > **Save data and return** to complete the measurement and return to the Data Collector module.

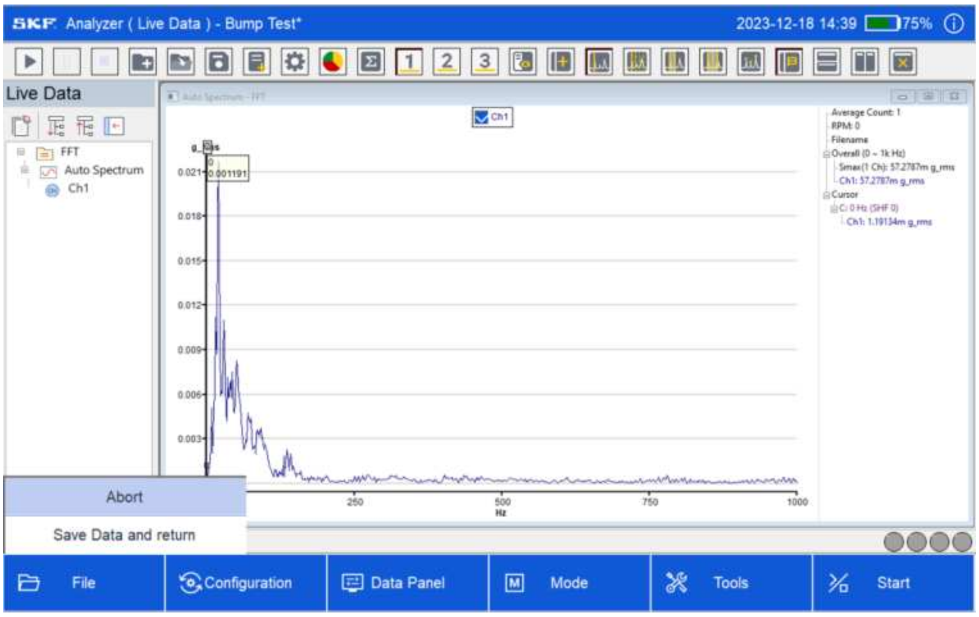


Figure 50 Save data

The software will prompt you to save the data to a file in the Analyzer format. You can review the saved data from the Analyzer module later.
It will also save the data to a **UFF** (Universal file format) file in a specific folder. The UFF file will be uploaded to the host software on your computer when the archive data is uploaded to the computer.

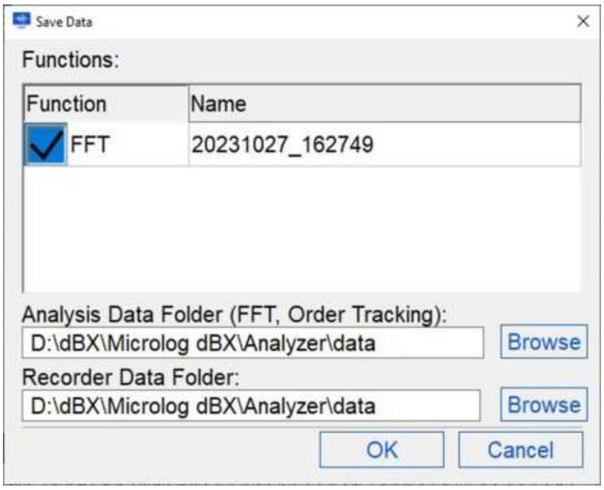


Figure 51 Save data – example

Note that only measured functions that are supported by the host software, SKF @ptitude Analyst, will be uploaded to the computer. Supported functions: **Analyzer**, **gE**, **Bump test** and **Orbit**.
Not supported measured functions can be reviewed from the Analyzer software.

2.6.6 How to create a user-defined template

To create a user-defined test template for Non-Route measurement:

1. From the home screen, double tap the SKF Analyzer icon or select a test template.
2. Set up or modify the measurement parameters and the display layouts.
3. Select **File** > **Project** > **Save as** to save this project.
4. Select the following folder path: *D:\dBX\Microlog dBX\Analyzer\Data_Collector*.

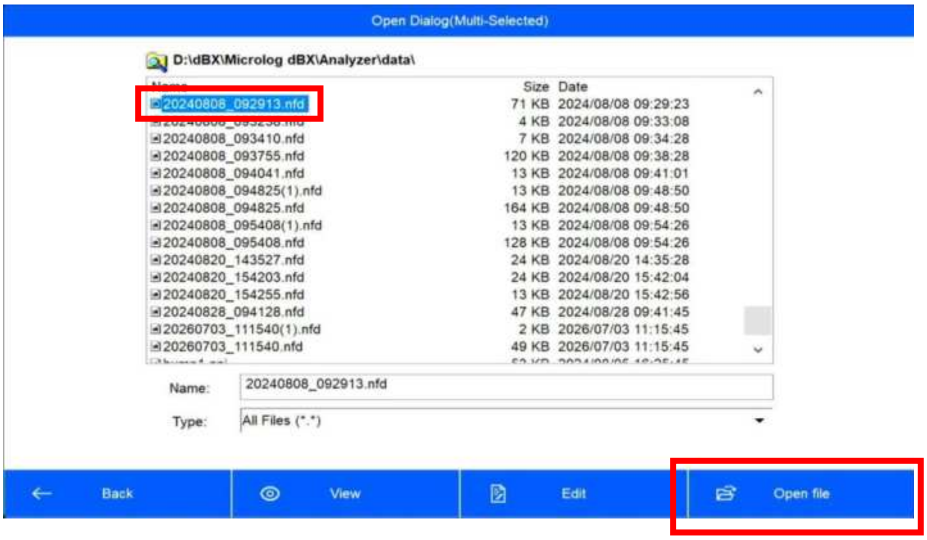
Next time you activate the Non-Route measurement, these saved projects will be shown as a user-defined template.
For more information, refer to the dBX Analyzer module chapter.

2.6.7 How to review Non-Route data

To review a Non-Route data using the Analyzer module, follow this instructions:

1. From the home screen, double tap the SKF Analyzer icon.
2. From the main menu, select **File** > **Data File** > **Add** to open the **Select Disk** dialog.

3. From the **Open Dialog** window, locate and select the data file. Tap the **Open** button to open the file.



4. Now you can locate the added data from the **History** panel. Select a function and press **OK** to display it on the plot area.

For further instructions, refer to the dBX Analyzer module chapter.

2.7 Multiple Point Automation (MPA in a Flash)

The Microlog Analyzer dBX allows the user to configure up to 12 measurements for automatic data collection at one measurement Point. Using the same sensor, the user needs to press only one button to sequentially collect all pre-configured MPA measurements.

MPA Point rules
Points in an MPA group should adhere to the following:

- There must be more than two Points in an MPA group.
- All Points must have the same coupling.
- The same sensor must be used for all Points.

2.7.1 MPA Route setup

To set up an MPA Route

- In your host software, such as @ptitude Analyst, access the **Point Setup** screen and use the **Description** field to identify MPA Points. The first characters in the Description field identify the MPA group to which the configured Point belongs. For MPA Points, the first character in the **Description** field must be "@". The "@" character must be followed by one or more alphanumeric characters that identify the group of MPA Points.

The characters following the "@" in the **Description** must be identical for each Point in the MPA group. For example,

@MT1 VELOCITY,
@MT1 ACCEL
@MT1 ACC ENV

are legal **Descriptions** for three MPA Points collected at the same location on a motor.

For three additional Points collected at a second location on the motor,

@MT2 VELOCITY,
@MT2 ACCEL
@MT2 ACC ENV

might be appropriate **Descriptions**.

MPA codes can be of any reasonable length, so the following setup is acceptable:

@MT31137 Motor NDE H VEL,
@MT31137 Motor NDE H ACCEL
@MT31137 Motor NDE H ACC ENV

Note that when MPA Points are downloaded to the Microlog dBX, they are combined into a single route Point. You may see the names of the original Points and their measurement parameters in the Measurements panel when the Point is selected.

The name of the single route Point containing MPA Points is determined as follows:

- If there is a common root to the Point names, it is used as the start of the MPA Point name.
- If there is no common root name, the name of the first point is used as the start of the MPA point name.
- The name of the MPA group is then appended to this name.

In the above example, this would result in an MPA Point named Motor NDE H - @MT31137

If more than 12 Points are defined, Microlog dBX collects data for the first 12 MPA Points, and the remaining MPA Points are ignored.

2.8 Data review

Move the cursor to a Point which has history data stored and press **OK** to enter the archive data table. Press the **C** key or the **Back** button to go back to the Route hierarchy.

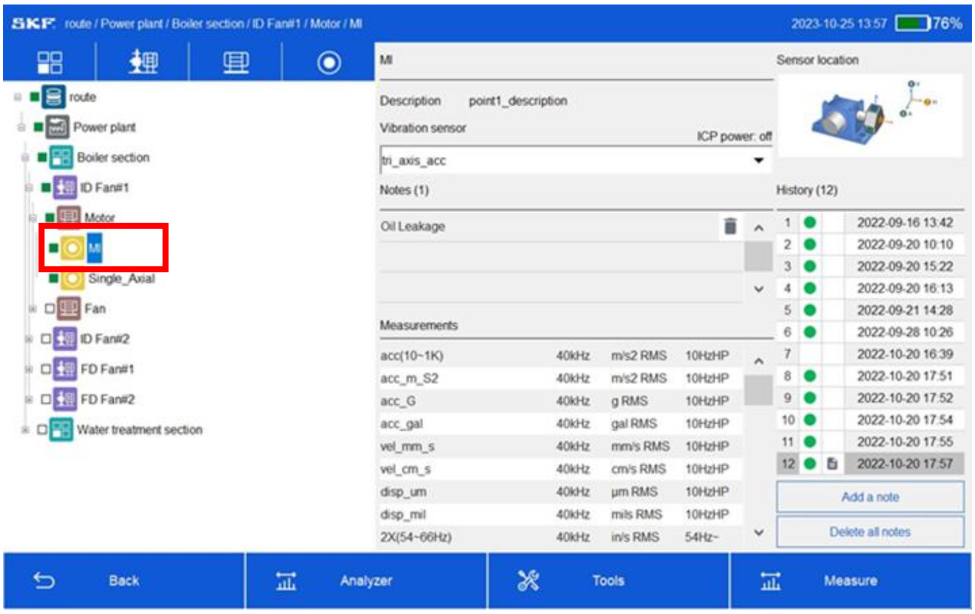


Figure 52 Archive data

When the cursor is on the history data table, use the navigation keys to move the cursor to the selected date and time. Press **OK** to enter the archive data display.

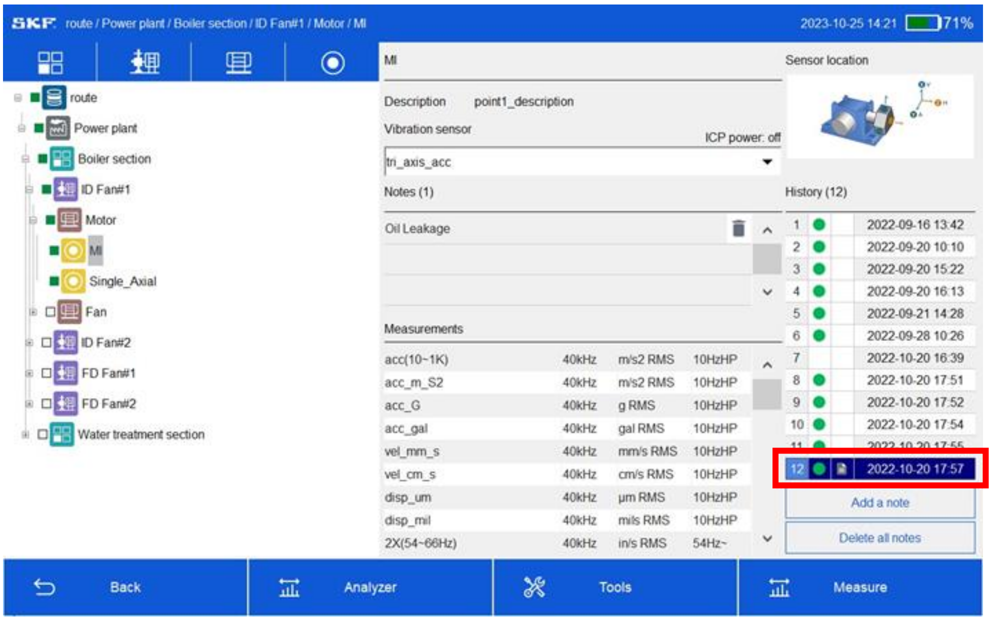


Figure 53 Date and time of a history data record

2.8.1 Multiple plots and single-plot display

In a multiple plot display, up to six plots at a time can be shown. Overall plots are prioritised, and the remaining plots presented are ordered depending on the setup.

If there are more plots, select **Next** to enter the other pages. You can display a selected plot in a single-plot display by maximizing it for in-depth data review.

- **Active plot** – You can always access an active plot which has a blue title bar on the top. The inactive plots have grey title bar colour. Tap on a plot to select and make it active.

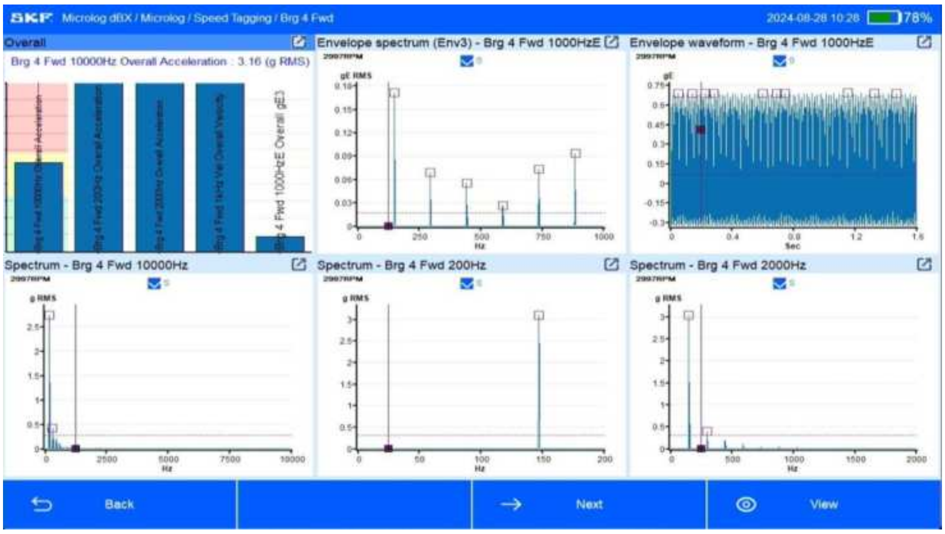


Figure 54 Active and inactive plots – example

Alternatively, press the **C** key to enable the active plot selection. It will change the title bar colour of the active plot into red. Then you can use the navigation keys to move the red bar to another plot and press **C** to disable the selection mode.

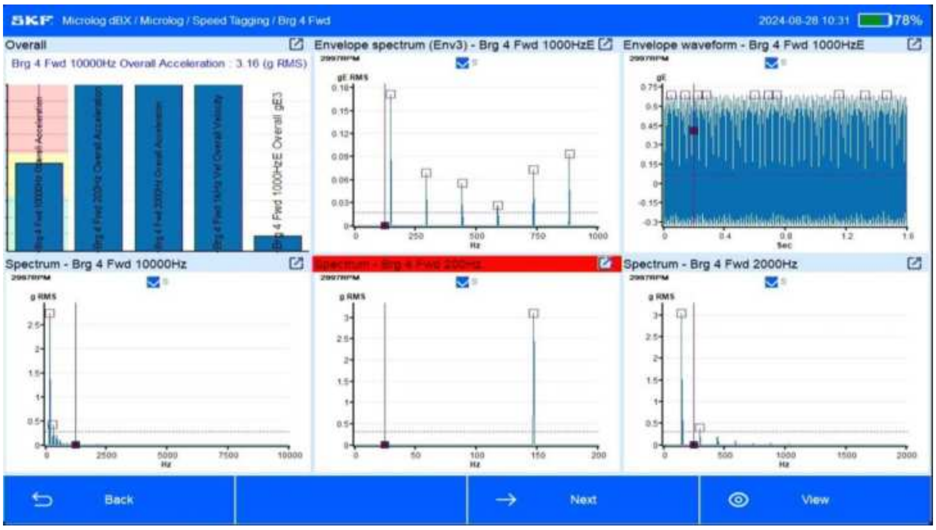


Figure 55 Red colour title bar example

- **Maximise or restore an active plot** –Press the **OK** key to maximize an active plot and press **OK** again to restore it.
  Alternatively, tap the **Maximise** caption button on the right side of the title bar to maximize or restore a plot.

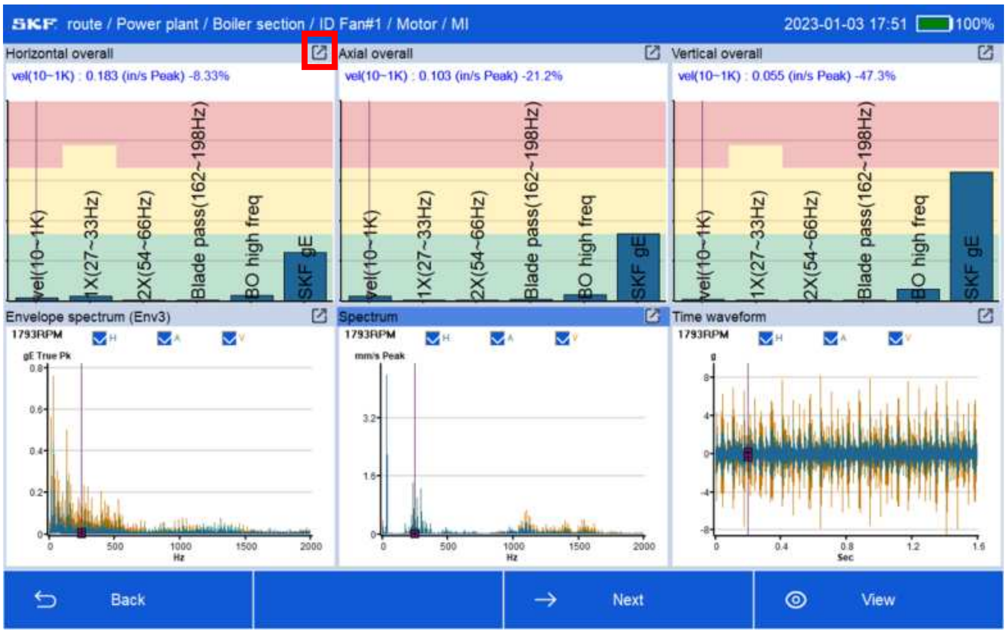


Figure 56 Maximise plot icon

Figure 57 Minimize plot icon

- **View menu and right click menu** – To display a plot with different settings or formats, use the **View** menu. You can also use the right-click key to display the pop-up menu for option selection. Note that the contents of the

Right click menu and **View** menu are identical.

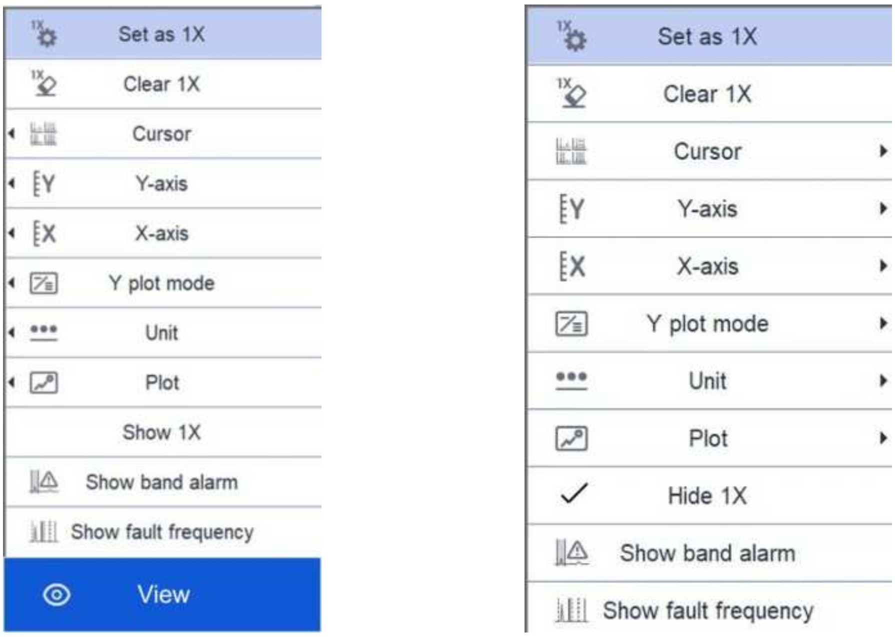


Figure 58 View menu (left) Right-click menu (right)

2.8.2 Overall plot

The Overall plot contains all the single-value data, such as overall, band overall, gE level, temperature, etc.

- **Bar plot** – Each value is displayed as a bar with its label on it. Note that all the bars are normalised by their Alarm 1 values, hence they are dimensionless data shown as percentages on the plot. With this arrangement, you can read the alarm condition of each value very quickly. The readings higher than 300% of the alarm 1 have their bars clipped. To read the real value of each bar, move the cursor using the navigation keys or tap on a bar to show it on the cursor value field.

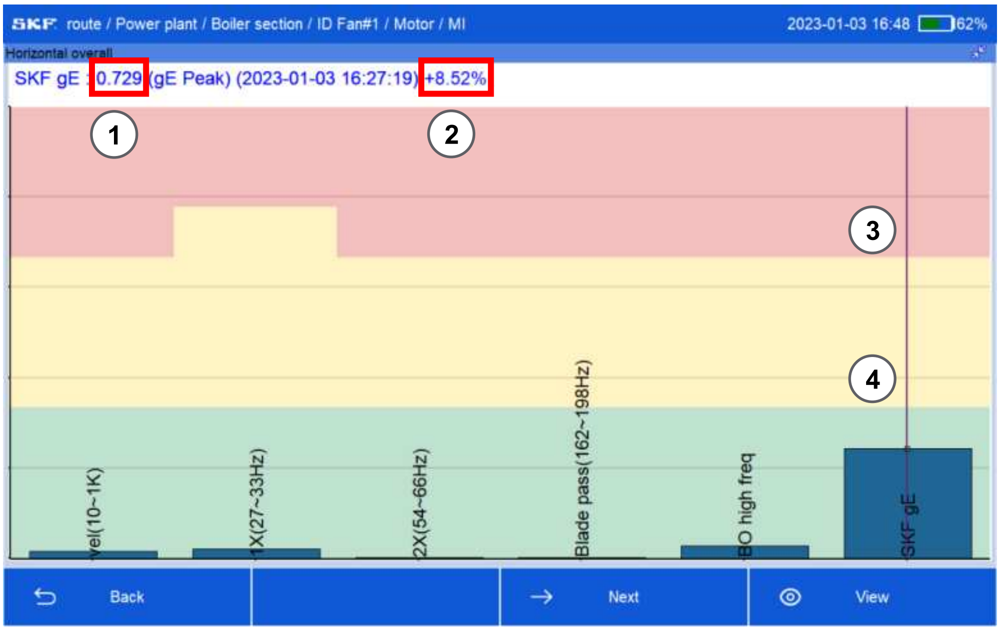


Figure 59 Bar plot – example

1. Cursor value
2. Percentage
3. Danger
4. Alert

- **Trend plot** – Move the cursor of a bar chart to a selected bar, then tap **View** to select **Trend plot** from the menu.

To restore to bar plot display, select **Overall plot** from the **View** menu.

Figure 60 Trend plot

2.8.3 Waveform plot

Displays time waveform or gE waveform data with various layouts and formats. Select different options from the **View** menu.

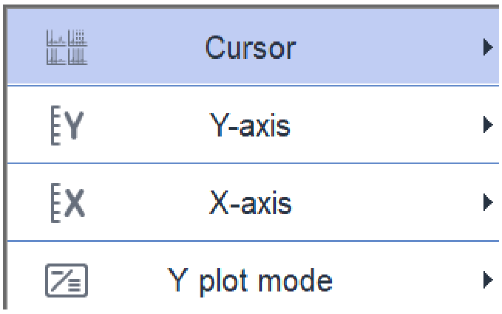
- **Overlap and separate display** – Time waveforms of the multiple directions are shown in one overlapped plot.

Figure 61 Time waveforms – Overlapped view

Select **Separate** from the **View** menu / **Y plot** mode to change it to a separate plot.

Figure 62 Three separate plots example

- **Time waveform plot view menu**



  - **Cursor** – Select Cursor type.
  - **Y-axis** – Set the scale of Y axis as auto or fixed.
  - **X-axis** – Set the scale of X axis as auto or fixed.

2.8.4 Spectral plot

There are two types of spectral data **Spectrum** and **gE spectrum**. Select different options from the **View** menu to display spectral data with various layouts and formats.

2.8.5 View menu of spectral plot

For detailed information on spectral cursors and how to use them, refer to Working with cursors.

- **Set as 1X** – Move the cursor to a selected frequency and set it as 1X. It will set the cursor frequency at 1X (rotational speed) of this spectrum.

  Note that the tagged 1X value overwrites the ratio speed of this Point and is applied to all the other measured functions under this Point.

Figure 63 Set as 1X (Right), Clear 1X (Left)

- **Clear 1X** – Clear the 1X setting and restore back to the rated speed.
- **Cursor** – Select cursor type.
- **Y-axis** – Set the scale of Y-axis as Auto scale, Fixed scale, Linear scale, or log scale.
- **X-axis** – Set the scale of X-axis as Auto or Fixed scale. Display the X-axis unit in Hz, CPM or Order.
- **Y-plot mode – Separate and overlap** – Displays the spectral data of the multiple directions in separate plots or an overlapped plot by selecting one from the **View** / **Y-plot** mode menu.

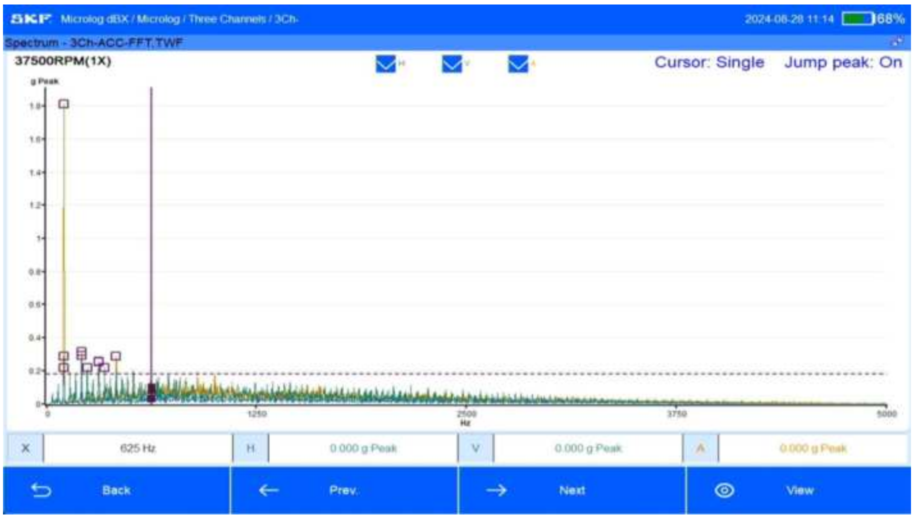


Figure 64 Spectral plot – example 1

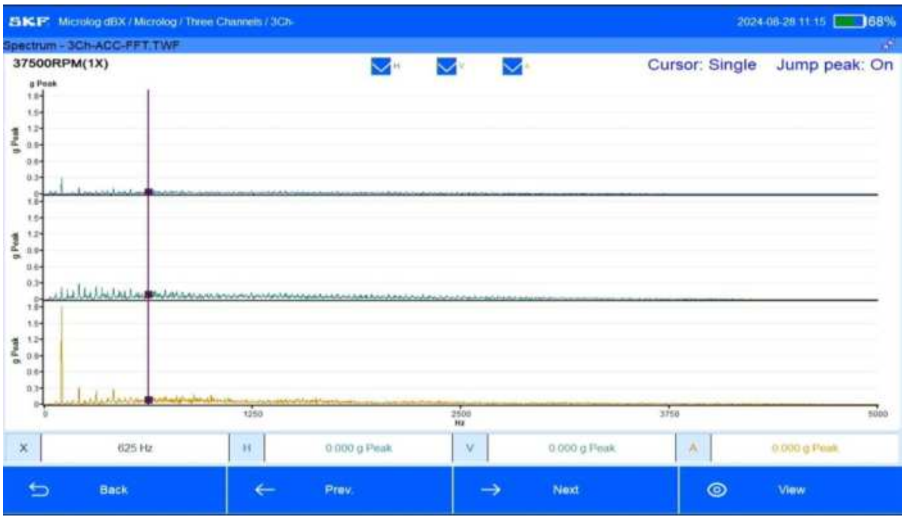


Figure 65 Spectral plot – example 2

- **Unit** – Perform a unit conversion for the plot. For example, if the data is measured in velocity, you can select to display it in other velocity units: mm/s, in/s, or cm/s.
  Note that the converted units are used for display purposes only, and it will not overwrite the measured data.

- **Plot** – Display the historical spectra data in waterfall plot.

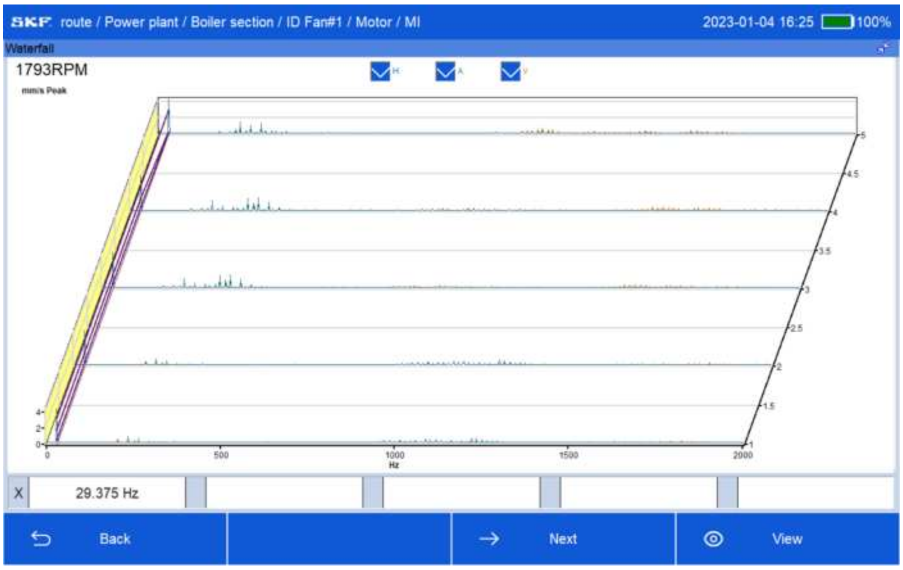
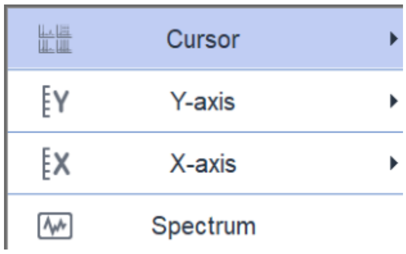


Figure 66 Waterfall plot – example

- **Show / Hide 1X** – Select to show or hide the 1X.
- **Show / Hide band alarm** – Select to show or hide the band alarm.
- **Show / Hide fault frequency** – Select to show or hide the fault frequency.

2.8.6 View menu of waterfall plot



- **Cursor** – Select a cursor type from X, Y, and X+Y.
- **Y-axis** – Set the scale of Y-axis as auto or fixed.
- **X-axis** – Set the scale of X-axis as auto or fixed.
- **Spectrum** – Return to the single spectra plot.

2.8.7 Show or hide band alarm

Show or hide the band alarm curves on the spectral plot. Note that the band alarm display is available only when the plot mode is set as **Separate**.

Figure 67 Show / Hide alarm curves

2.8.8 Show or hide fault frequency

Show or hide the fault frequencies on the spectral plot.

Figure 68 Show / Hide fault frequency – example

2.8.9 Complex spectral plot

A complex spectrum contains both amplitude and phase information. The Data Collector module has two complex spectrum functions available:

- **Order spectrum** – The Order spectrum is measured with a digital order tracking algorithm. It provides the amplitude and the phase data concerning the speed sensor.
- **Phase spectrum** – The Phase spectrum is measured with a trigging signal from the speed sensor. It provides the amplitude and phase data. Unlike the Order spectrum, the phase spectrum is derived from a regular FFT calculation. Phase spectrum is only available as part of Order Tracking Point.

Figure 69 Cross Spectrum – Amplitude (upper screen) and Phase plot (lower screen)

Move the cursor to a selected frequency and zoom in to see the details. You can see the phase reading from the cursor value table.

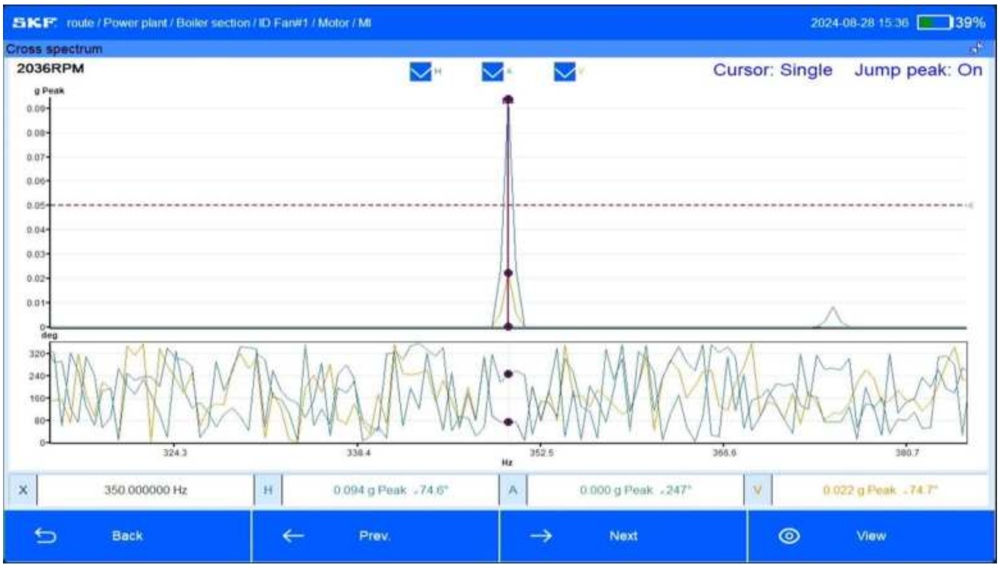
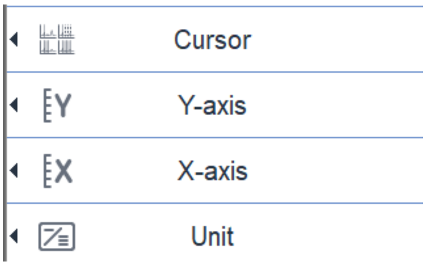


Figure 70 Cross Spectrum – Zoom

2.8.10 View menu of a complex spectral plot



For more information, see the description in section Complex spectral plot. Note that the band alarm curves, fault frequency display and waterfall plot of a complex spectral plot is **not** available from the **View** Menu.

2.8.11 Orbit plot

Display orbit plot with various options and settings. Select different options from the **View** menu.

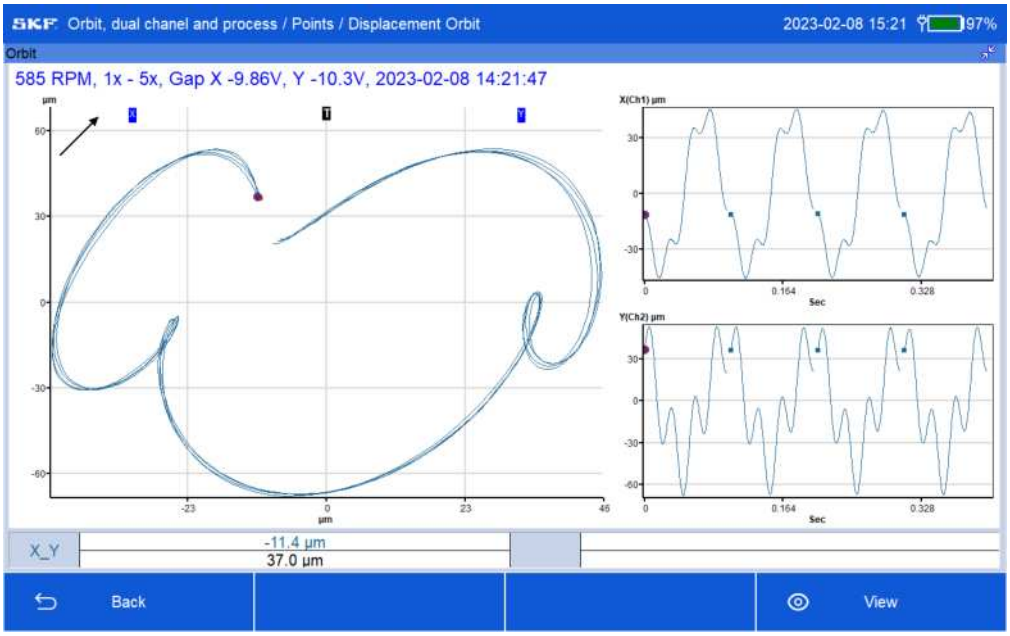
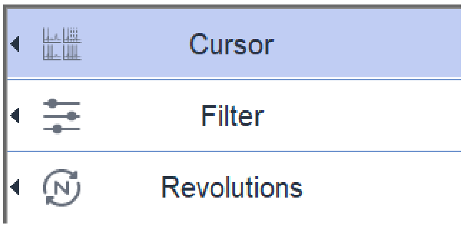


Figure 71 Orbit plot – example

On top of the orbit plot, the following parameters are shown Speed, Filter, DC Gap Value, and Cursor values.

- **View menu of Orbit plot**



  - **Cursor** – Select type of cursor from Point or Revolution. Select Revolution cursor to highlight one revolution at a time.
  - **Filter** – Select a filter from the options below.

Select **Custom** to set the minimum and maximum custom filter values.

- **Revolutions** – Select the number of revolutions for the plot.

2.8.12 Manual parameter

Display a manual parameter in trend plot or single Point format by selecting an option from the **View** menu.

Figure 72 View manual parameter – example

2.8.13 Amplitude and phase plot

Note: This plot can only be used when programmed from @ptitude Observer.

The amplitude and phase plot shows the amplitude and phase data of a specified order, such as 1X or 2X, on a polar plot.
Amplitude and phase plot is only available as part of Order Tracking Point.

Figure 73 Amplitude phase – example

- **View menu of amplitude and phase plot** – Select to display trend plot or single Point plot.

Figure 74 Amplitude phase trend – example

- **Show or hide a direction on a plot** – To show or hide a direction on a spectral or waveform plot, tap the check box at the top of the plot.

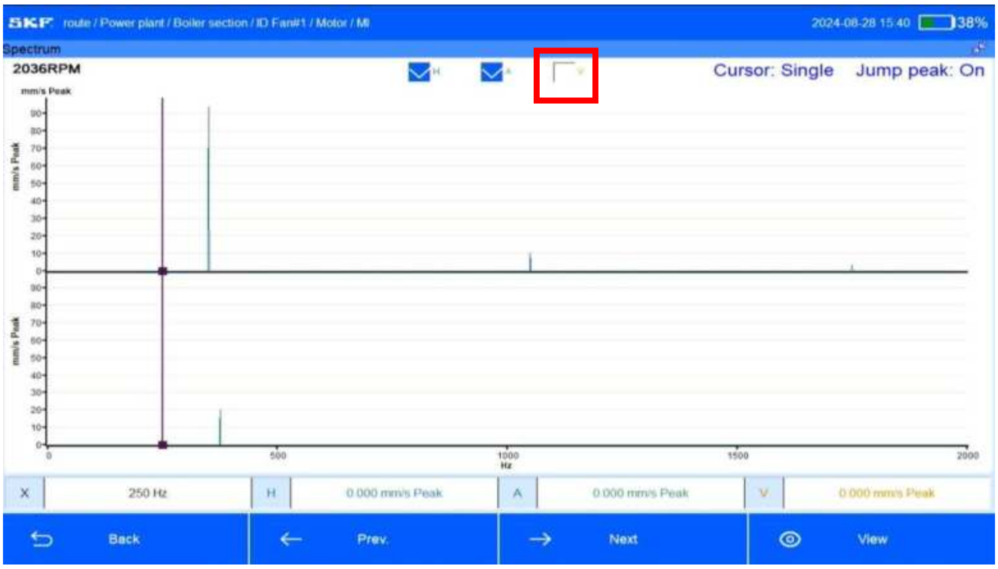


Figure 75 Show direction on a plot

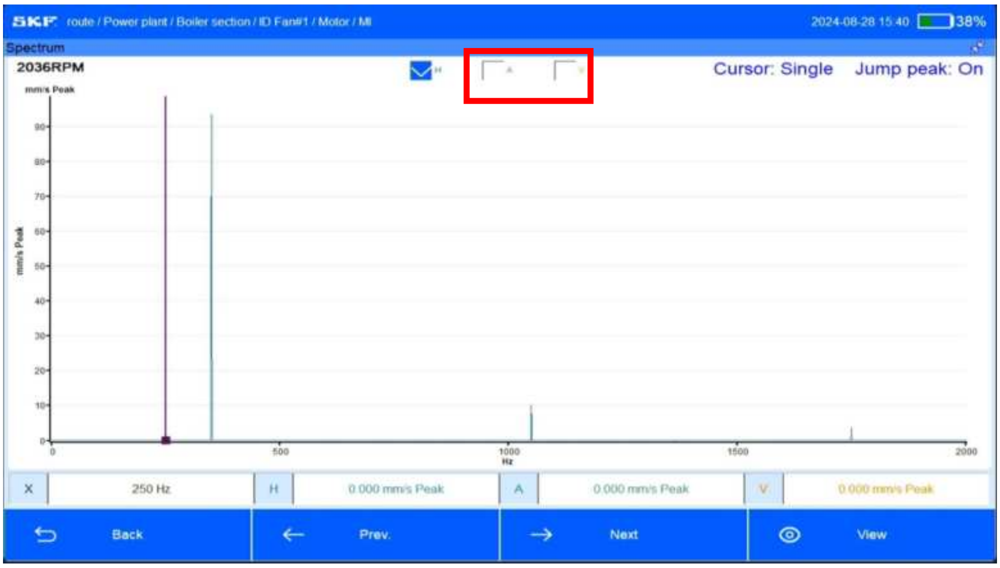


Figure 76 Hide direction on a plot

2.8.14 Working with cursors

Select a cursor

Select the type of cursor for navigating a spectral plot. Use the  key for quick toggling of the cursor type or select a cursor from the **Cursor** option in the **View** menu.

Move a cursor

Use the left and right keys to move a selected cursor or tap on a plot to locate the cursor to the tapped point. You can also slide your finger on a plot to move the cursor.

Types of cursors:

- **Single cursor** – Displays only one cursor on the plot.

Figure 77 Single cursor – example

- **Harmonic cursor** – Displays the harmonic orders of the cursor position.

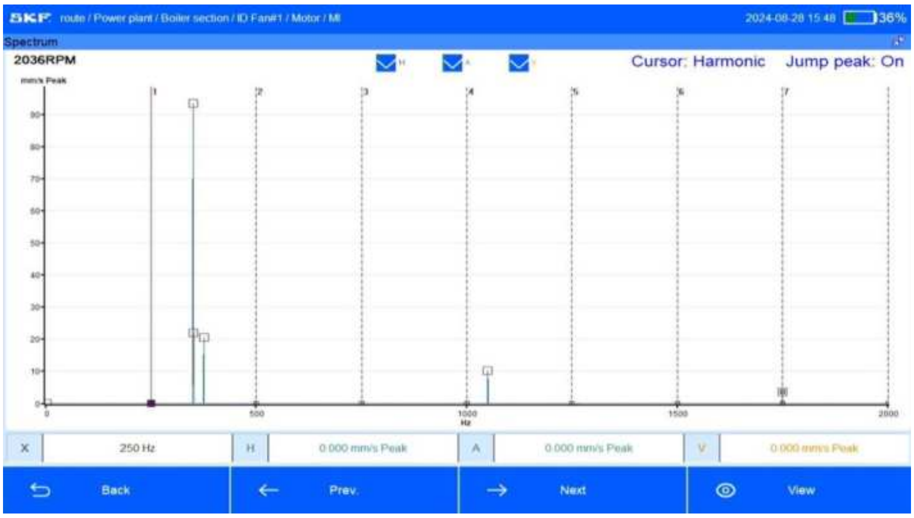


Figure 78 Harmonic cursor – example

Press the **C** key to make it possible to move the main cursor to each harmonic location and see the cursor value of each harmonic component. Press **C** again to return to a normal mode of a harmonic cursor.

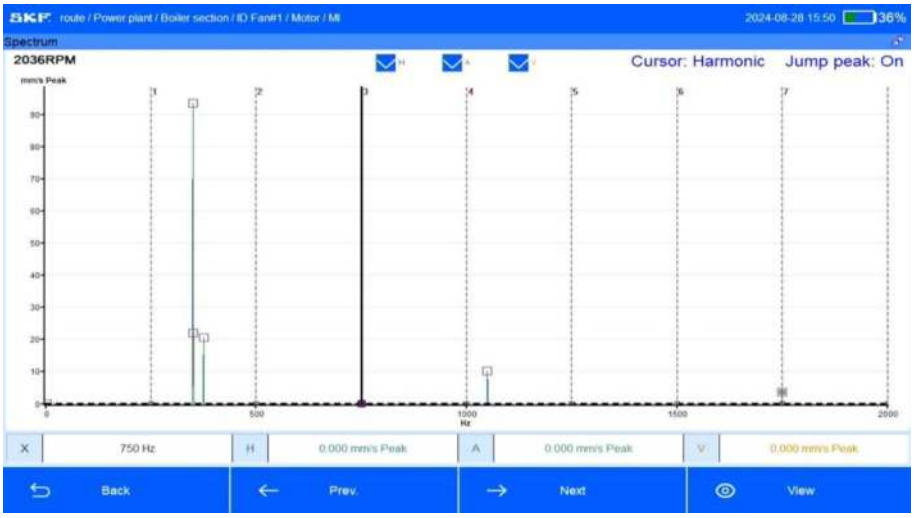


Figure 79 Harmonic cursor – example

- **Dual cursor** – Displays two cursors on a plot. Tap the **D1** or **D2** icon to select the left or the right cursor and move it. Tap at the middle of the dual cursor to move the complete Dual Cursor.

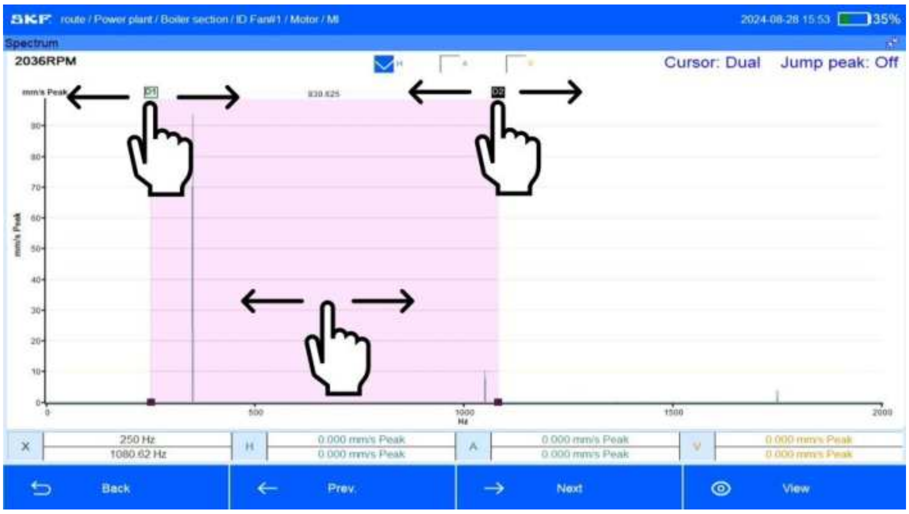


Figure 80 Dual cursor

- **Side band cursor** – Displays side band cursors. Tap the **S1** or **S2** icon, then use the left or right key to move the side cursors. Tap the **S** icon and use the left or right key to move the centre cursor.

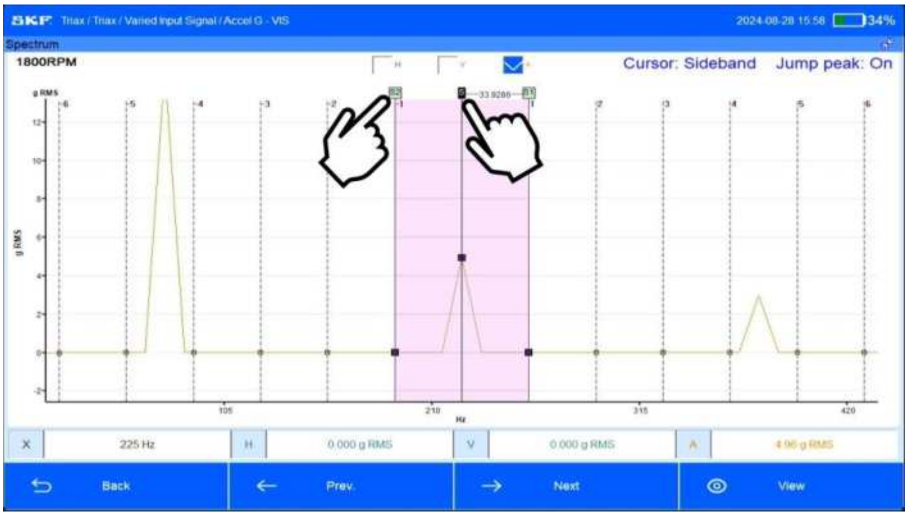


Figure 81 Side band cursor

- **Jump peak** – Make the cursor always stay on a peak. When you move the cursor, it will jump from one peak to another. **Press the Up key twice** to enable or disable the move peak function.

2.8.15 Zoom in, zoom out, auto scale and pan

- **Zoom in** – Press the zoom in key to zoom in the plot along the X-axis from the point where the cursor is on. You can also place two fingers on the plot and move them apart to conduct the zoom-in operation.

- **Zoom out** – Press the zoom out key to zoom out the plot along the X-axis from the point where the cursor is on. You can also place two fingers on the plot and move them together to conduct the zoom out operation.

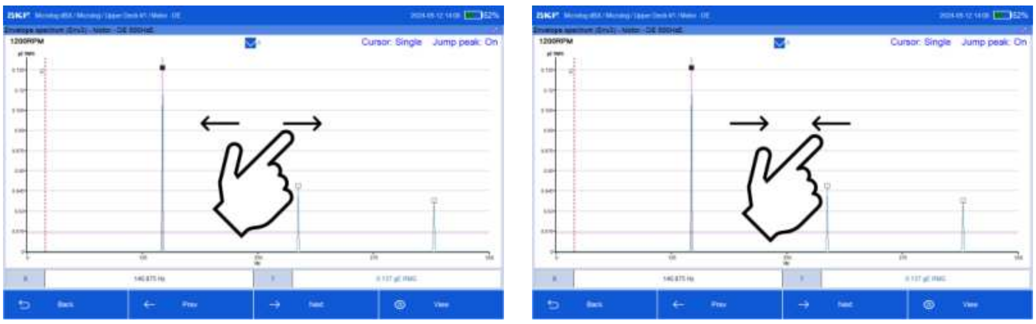


Figure 82 Zoom in and zoom out a plot

- **Auto scale** – Double tap at the legend area of the X-axis to auto-scale the X-axis. Double tap at the legend area of the Y-axis to auto-scale the Y-axis.

Figure 83 Double tap auto scale – example

- **Pan** – Use two fingers to drag the plot to perform the Pan function and move the data trace to the desired position.

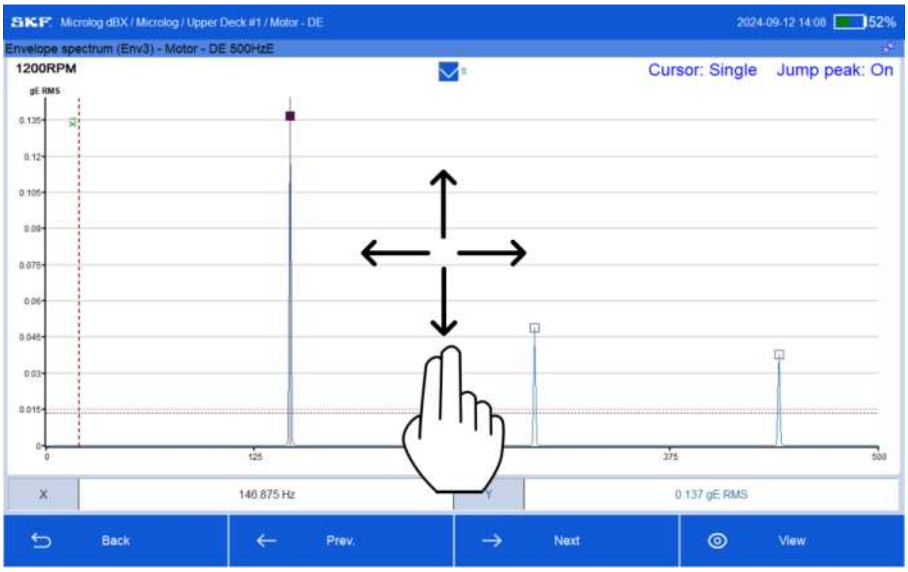


Figure 84 Pan function – example

Tap **Back** to return to the Route display.

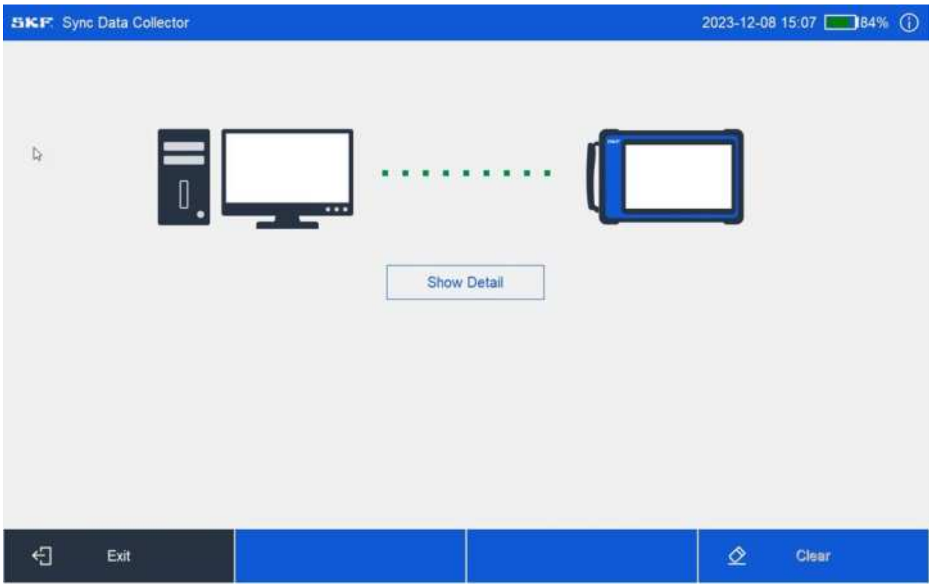
3 Sync Data Collector module

3.1 Connect Microlog dBX to a computer

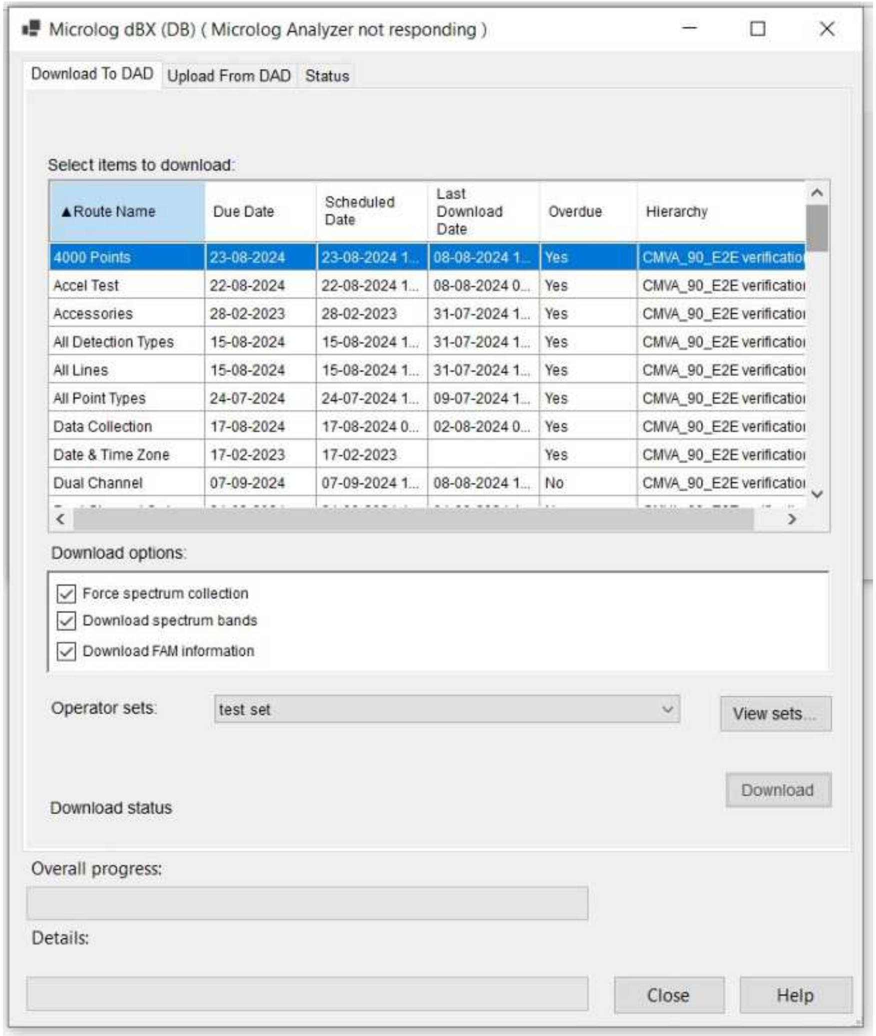
> *Before using the software, please read and understand the separate Microlog Analyzer dBX Quick Start Guide.*

1. Using the provided **CMAC 9010** (Data Link GT168) USB cable, connect your Microlog dBX to the computer's USB port.
   Note: The connection **cannot** be established by using a standard USB cable.

2. Double tap the **SyncDataCollector** icon from the home screen.



3. On your computer, open the **Analyst TCT dBX** (Thin Client Transfer).
4. Select **Transfer** > **Microlog dBX…** and check the Microlog Analyzer status If the connection has failed, it will show an error message.

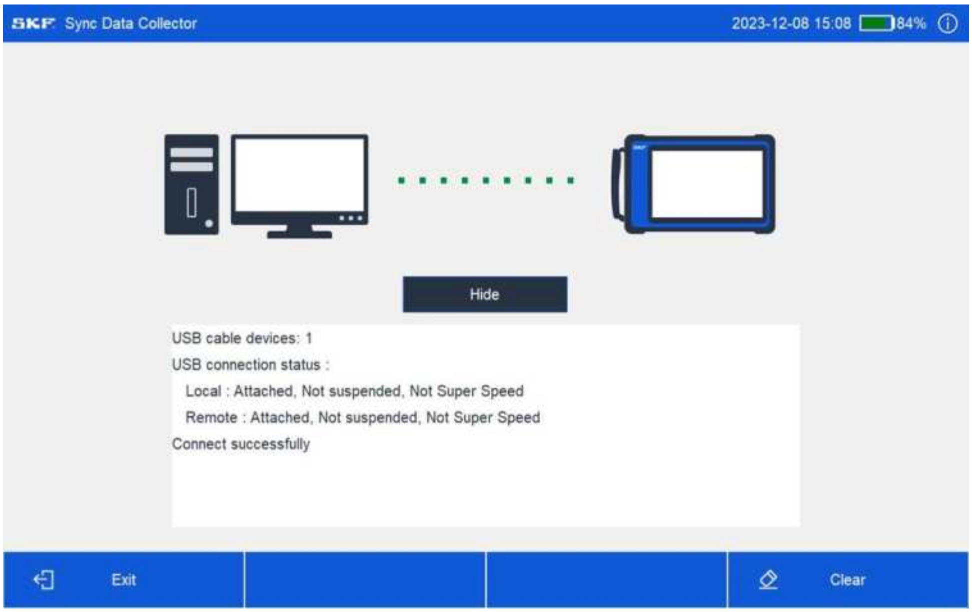


> ⚠ *For more information about the **Analyst TCT dBX**, read the separate SKF @ptitude Analyst Thin Client Transfer dBX User Manual.*

> ⚠ *For more information about the **Data Bridge**, read the separate SKF @ptitude Observer Data Bridge User Manual.*

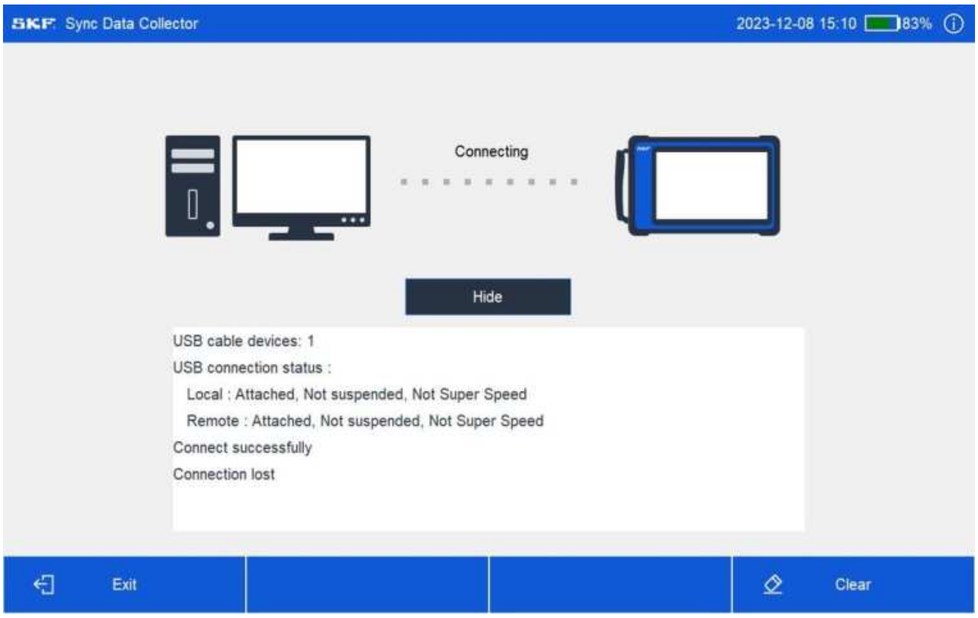
3.2 Connection status

When the connection is successful, it will show a scrolling green dotted line between the Microlog dBX and the computer and a **Connect successfully** message will appear in the information section.



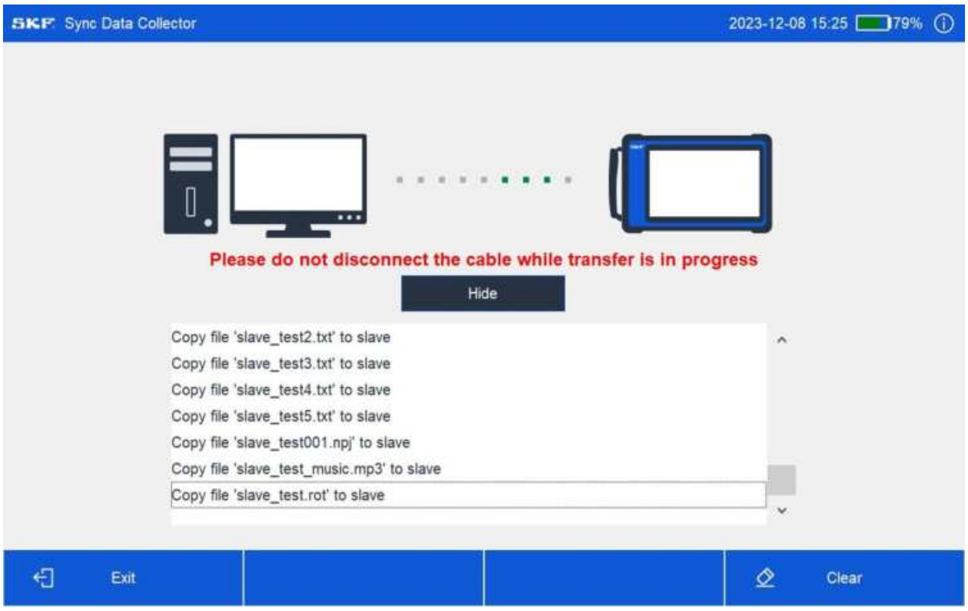
Tap **Hide** to toggle the information section.

If the connection is failed, the green dash line will turn into a grey colour, and a **Connection lost** message will appear in the information section.



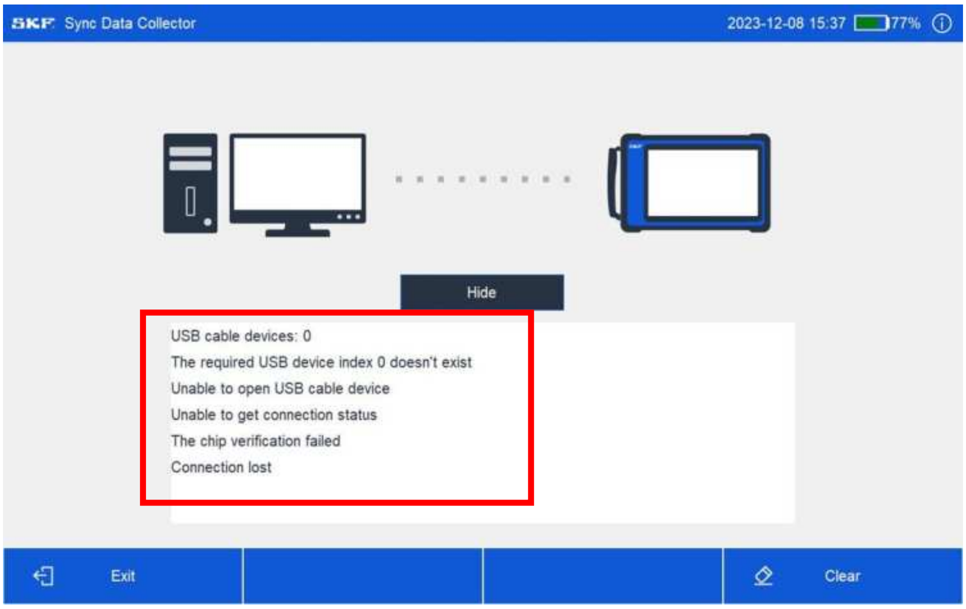
3.3 Data transferring status

When data transfer is in progress, a warning message is shown in the middle:
Please do not disconnect the cable while transfer is in progress



3.4 Error message

If connection fails, check the error message, and follow the suggested remedies.



Error Message	Remedy
Failed to enumerate USB cable devices	Check if the USB cable is plugged in
The required USB device index %d doesn't exist	Unplug and re-plug the USB cable
Unable to get information of device 0	Unplug and re-plug the USB cable
Unable to open USB cable device	Unplug and re-plug the USB cable
Unable to get connection status	Connection missing or failed, unplug and re-plug the USB cable
GetFolder: failed -30	Upgrade TCT.dBX to the latest version

4 Balancing module

4.1 Introduction

> *Before using the software, please read and understand the separate Microlog Analyzer dBX Quick Start Guide.*

The **Balancing** module is used for balancing various types of rotors on different machine types. It contains several balancing software features that fast and easily balance the rotors through a step-by-step process.

To start the Balancing module, using the navigation keys, navigate to the SKF Balancing icon and press **OK** or double tap the icon.

Figure 85 Home screen – Balancing icon

4.2 Home screen

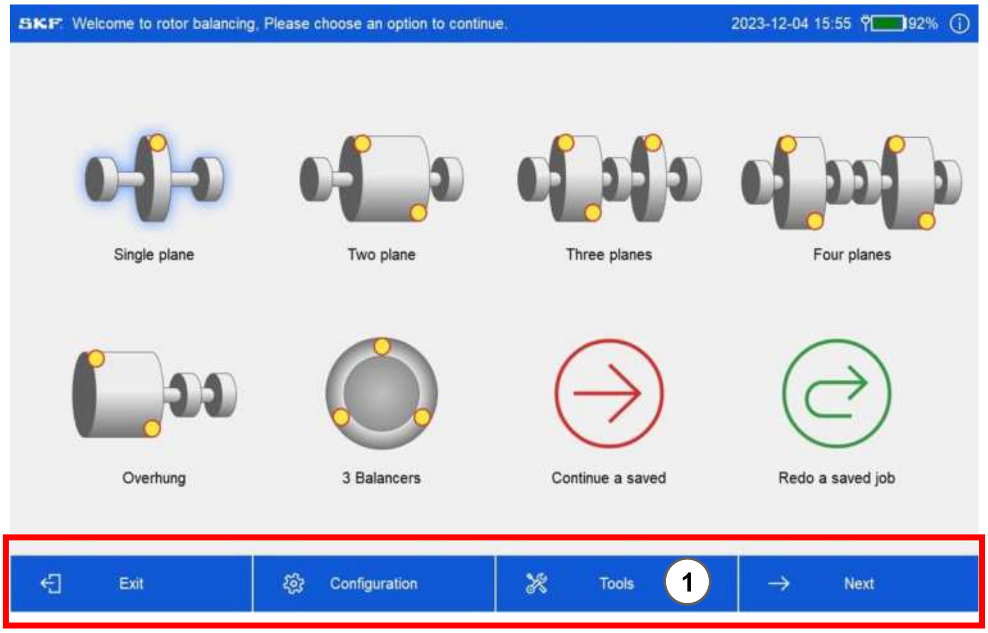


Figure 86 Balancing module home screen

1. Menu buttons

On the rotor balancing home screen, the following software and options are available:

- **Single Plane** – Single plane balancing on a new rotor, including the trial weight process and calculating balancing coefficients.
- **Dual Planes** – Dual plane balancing on a new rotor, including the trial weight process and calculating balancing coefficients.
- **Three planes** – Three planes balancing on a new rotor as shown in the picture, including trial weight process, and calculating the balancing coefficients.
- **Overhung** – Overhung rotor balancing on a new rotor, including the trial weight process, and calculating balancing coefficients.
- **Four planes** – Four planes balancing on a new rotor as shown in the picture, including trial weight process, and calculating the balancing coefficients.
- **3 Balancers** – Start new balancing job on three permanently mounted balancing weights as shown in the picture, including trial weight process, and calculating the balancing coefficients.

- **Continue a saved job** – Open a saved file and continue an unfinished balancing job with saved coefficients and historical data. It is one of the key features when you are in the middle of a large job.

- **Redo saved job** – Open an existing file where the balancing coefficients are saved.

- **Exit** – Tap to exit the Balancing module.

- **Configuration** – From this menu the following options are available:

  - **System Units** – Select **Metric units** or **Imperial units**.

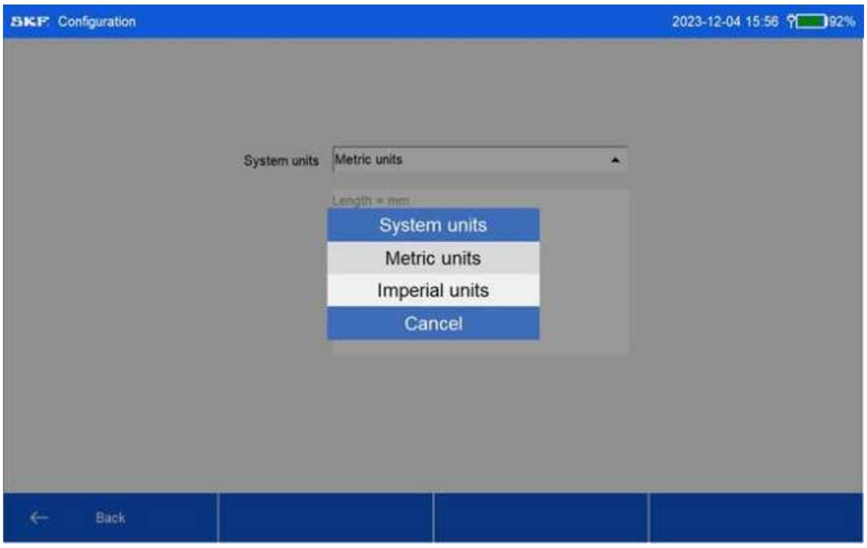


Figure 87 Configuration view – System units

  - **Load setup** – Tap to open a new window where you can select a setup file and load the setup parameters for your balancing job. Note that from the extension name of a setup file, you can tell what type of balancing parameters are stored in it. The corresponding balancing types for different extension names are:

    - *.bals1* – Single plane balancing
    - *.bals2* – Two plane balancing
    - *.bals3* – Three plane balancing
    - *.bals4* – Four plane balancing
    - *.balso* – Overhung balancing
    - *.balst*: – 3 balancer balancing

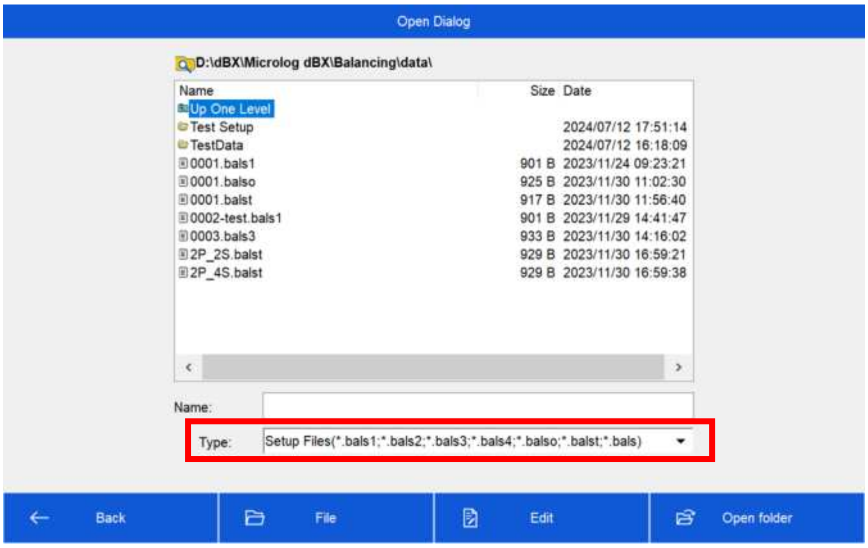


Figure 88 Different extension setup files – example

Tap the **Type** dropdown menu to choose the balancing job type.
After selecting the type of balancing job, you can load a setup file. Only files of the same type will be displayed. For example, selecting Single plane balancing will show only .bals1 files.

- **Tools** – From this menu two options are available:
  - **ICP Bias Check** – Check the power and the bias voltage of your ICP sensors' circuit loop.
    Select ICP Bias Check and new window will open with request to connect all you ICP sensors to the input sensor.
    Select **Verify** to perform the check, and the system will show the results in a window. The following conditions may appear:
    - **Short circuit** (low bias) – Indicates a bad cable with short loop.
    - **Normal** (normal bias) – The ICP circuit is working properly.
    - **Open circuit** (high bias) – If no sensor is connected to this channel, the bias check shows **Open**. If an ICP sensor is connected to this channel, an open circuit usually means a broken cable.

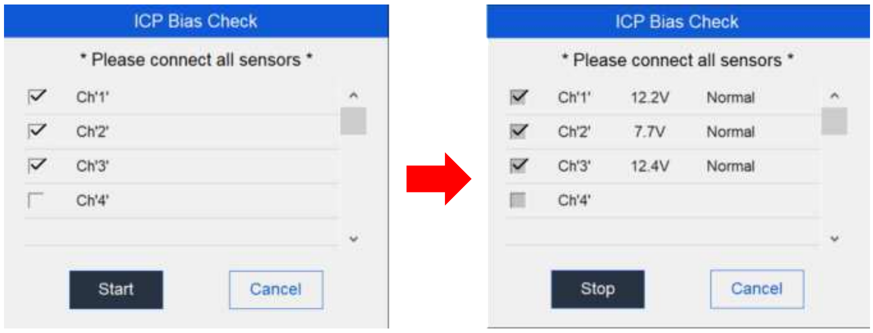


Figure 89 ICP Bias Check – example

- **ICP Power Check** – Check the ICP power supply of your Microlog dBX. To perform a check, disconnect all sensors and select **Start**. In normal conditions, the power supply of all channels should be around 25.6 V.
  If the result shows abnormal voltage levels, contact SKF's Technical Support Group TSG.

Figure 90 ICP Power Check – example

- **Version information** –Tap the information icon at the top right corner to display the version information.

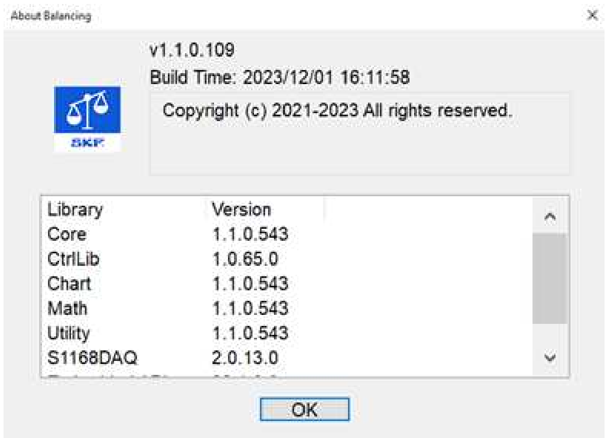


Figure 91 Version information – example

4.3 Balancing setup

After selecting a balancing job (single plane, dual planes, etc.) from the main screen, in the newly open view, various parameters for each type of balancing setup are shown.

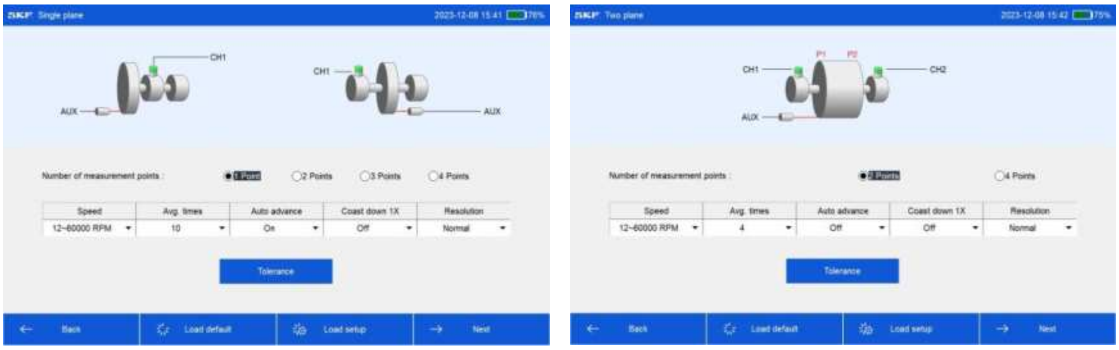


Figure 92 Single plane setup (left), Dual plane setup (right) – example

For accurate results, the sensor positions (**CH1**, **CH2**, **CH3**, **CH4**) on each display **must** be mounted on the machine as shown in the figure. Placing the sensor in the wrong position will generate incorrect balancing coefficients and be potentially dangerous.

- **Load Default** – Load the default saved setup. For more information, see section Sensor setup.

- **Load setup** – Opens the setup dialog where previously saved setups are stored. Select one setup from the list and tap **Open** to load the setup. For more information, see section Sensor setup.

- **Number of measurement Points** – When using a single Point for balancing, the measurement will rely entirely on only one sensor on each balancing plane to generate the balancing coefficients.

  In most cases, this is adequate to balance rotors. However, it may be necessary to use multiple Point balancing for more accurate balancing results. multiple Point balancing will be performed automatically once the user selects more than the minimum number of measurement Points for a given plane.

  For example, if you select **2 Points** in single plane balancing or **4 Points** in a dual or overhung rotor, you will use multiple Point balancing mode.

- **Multiple Point balancing mode** – When you select more than the minimum Points to balance, the balance algorithm will minimise all the vibration measured with the least square approach. This means that when both horizontal and vertical directions are measured in single plane balancing, it will try to minimise the vibration levels in both directions.

- **Speed** – Select a speed range for your balancing task. The default 12–60 000 RPM is appropriate for most balancing tasks.

- **Auto Advance** – Allows you to select **Off** or **On** start during the balancing measurement process. When **On** is selected, it will automatically enter the next measurement step.
  If **Off** is selected, press **Next or OK** navigation key to advance to the next balancing step.
- **Coast down 1X** – This function is for plotting the 1X (first order) vibration signal during coast down on your rotating machine. This plotted information is very useful in understanding your machine's behaviour during different operating RPMs. For more information, see section Heavy spot estimation.

  During your balancing process, you are balancing at a static RPM. With Coast down selected in the **ON** position, it will automatically enter the coast down measurement procedure once decreasing rotation speed is detected and show the results on the polar plots.

  Use this function to view the heavy spot (1X) on the polar plot.

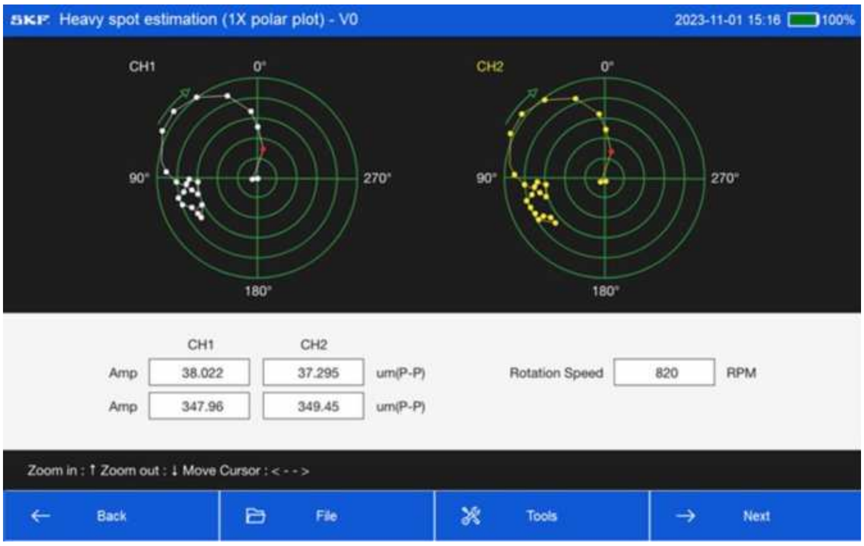


Figure 93 Heavy spot estimation – example

With the polar plots of the coast down procedure, you can easily find out the location of heavy spots even before the trial weight is applied.

- **Resolution** – Set the resolution of the tracking filters as
  - **Low**
  - **Normal**
  - **High**

- 0.03 Order
- 0.015 Order
- 0.008 Order
- 0.004 Order

Usually, you can set the resolution as **Low** or **Normal**. If the vibration signal is affected by other signals close to the balancing speed, you may use higher resolution to separate the 1X vibration from other signals.

- **Average Times** – Select between **2**, **4**, **10**, **20** times or manual stop.

4.4 Tolerance setup

Select **Tolerance** from the home screen to open the tolerance setup window.

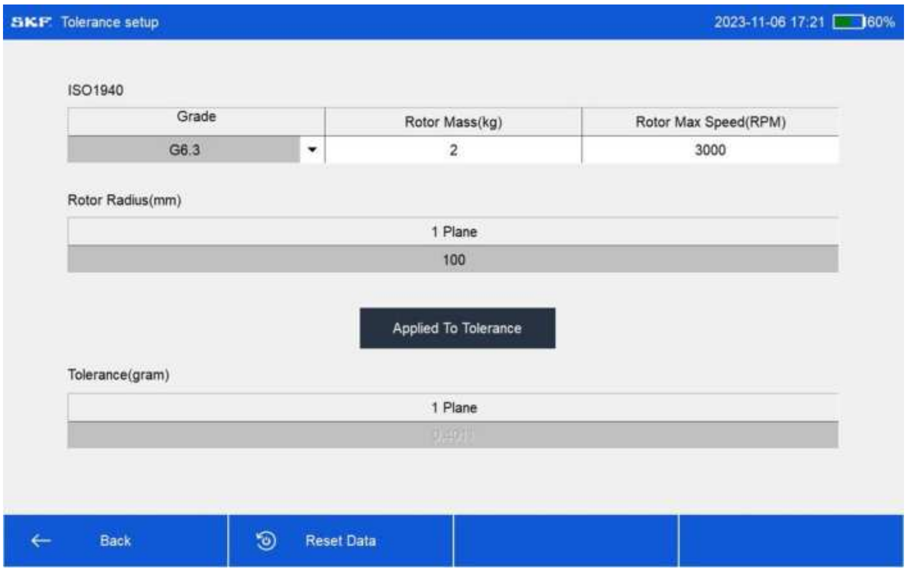


Figure 94 Tolerance setup

- **ISO 21940-11 Grade** – From the dropdown menu select one of the grades available. When you select **Applied To Tolerance**, it will apply values as shown in the table below. According to **ISO 21940-11**, the balance quality of a rotor is graded from G0.4 (best) to G4000 (poorest) or user defined.

Table 1 Balance grades available

G0.4	G100
G1	G250
G2.5	G630
G6.3	G1600
G16	G4000
G40	User Defined

The balance grade is a calculation of the rotor's maximum operational speed and the rotor's eccentricity, where eccentricity can be derived from the total rotor weight, the radius for balancing and the residual imbalance mass. When user defined is selected, you can only change the value for **Tolerance** for **P1**. The other fields are not selectable when user defined is selected.

Figure 95 Tolerance setup – Grade

- **Grade** – Select the balance level grade based on ISO 21940-11 standard.
- **Rotor Mass(kg)** – Enter the weight of the rotor.
- **Max Speed (RPM)** – Enter the maximum speed of the rotor. This value is used for calculating the maximum allowable unbalance according to **ISO 21940-11** standard.

  Residual Unbalance – allowable unbalance value remaining after the balancing job is completed as defined in the ISO 21940-11 standard.
- **Radius (mm)** – Enter the radius of Plane 1.
- **Tolerance** – Apply the ISO 21940-11 balance level to know whether the residual amount is within the standard.

After the setting is completed, tap **Applied To Tolerance.** Tap **Back** to return to the previous screen or tap **Reset Data** key without making any changes.

4.5 Phase correction

In this view the rotational direction and locations of the sensors are set.

4.5.1 When coast down 1X is disabled

When the coast down 1X measurement is disabled, the input of sensor locations is not needed and is disabled as shown below.

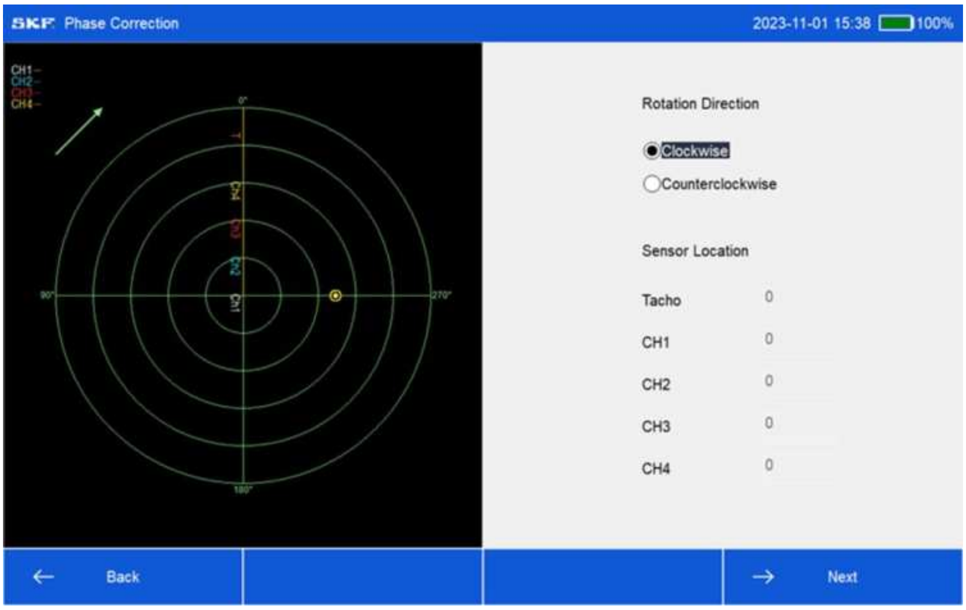


Figure 96 Phase correction view – coast down 1X disabled

In this scenario, you need to select the rotation direction to define the direction of the compensation angle. When the rotor rotates clockwise, the compensation angle is counterclockwise. When the rotor rotates counterclockwise, the compensation angle is clockwise.
The rotation direction and the sensing angle are related to the compensation angle calculated by the dynamic balance. Please select the correct rotation direction.

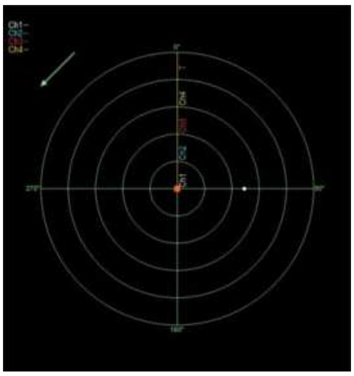


Figure 97 Clockwise (left), Counterclockwise (right)

4.5.2 When coast down 1X measurement is enabled

If you enabled 1X coast down measurement in the previous step, you'll need to input the sensor locations for measuring the "**absolute physical phase**" of the vibration signals correctly.
With the absolute physical phase, you can estimate the location of the heavy spot even before the trial weight measurement is taken.

The following plot shows an example of sensor locations:

- Tacho at 0 degrees,
- Ch1 at 45 degrees,
- Ch2 at 315 degrees with the rotation in a clockwise direction.

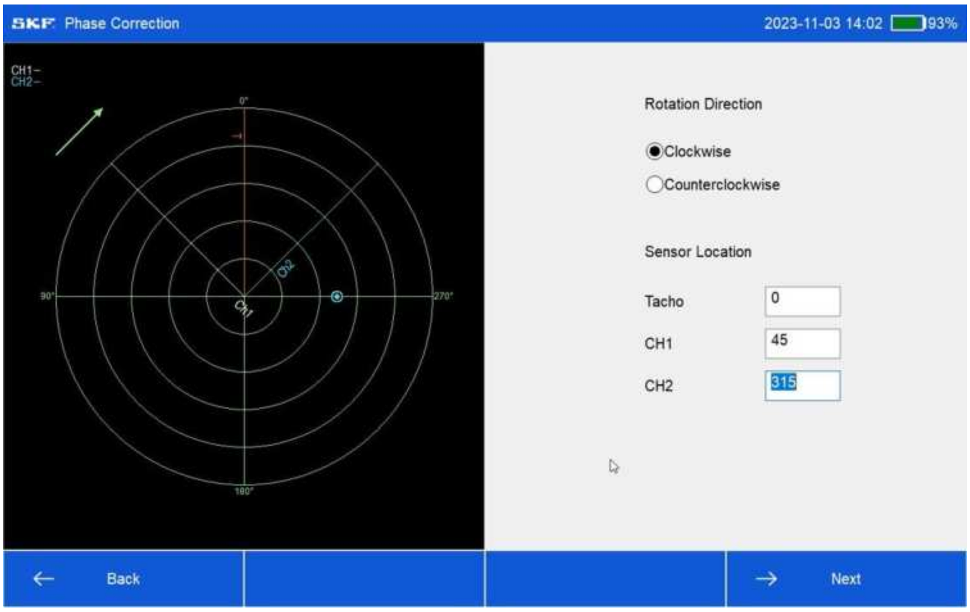


Figure 98 Phase correction view – coast down 1X enabled

4.6 Sensor setup

Use this view to configure the vibration sensor(s) being used for balancing.

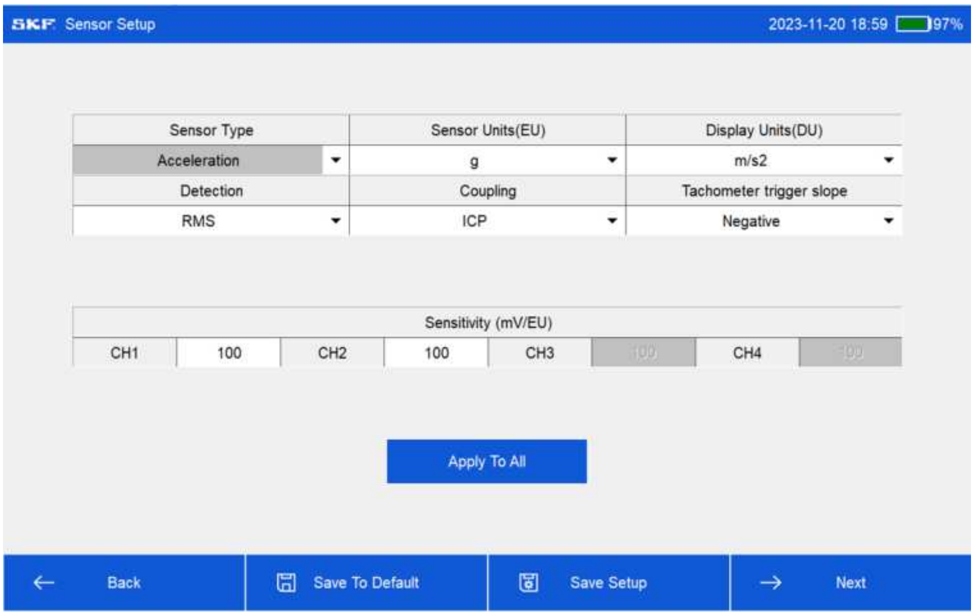


Figure 99 Sensor setup – example

- **Sensor Type** – From the dropdown, select **Acceleration**, **Velocity** or **Displacement**.

- **Sensor Unit (EU)** – From the dropdown, select **g**, **m/s2**, **in/s2** when Acceleration is selected, **m/s**, **in/s** when Velocity is selected and **m**, **um**, **mil** or **inch** when Displacement is selected.

  Note: Follow the sensor specification sheet for unit setting, do not choose the unit randomly. When mV/g is indicated on the specification sheet, select g.

- **Display Unit (DU)** – select **mg**, **mm/s2**, **mm/s**, **um**, **in/s2**, **in/s** or **mil**. The vibration levels will be displayed in the Display Unit on the vibration measurement page.

  Note: In this view, the display unit can be selected according to what the user wants to view. After the balancing process has started, the display unit cannot be changed again.

- **Detection** – Select **RMS**, **Peak,** or **Pk-Pk**.

- **Coupling** – Select sensor type **ICP**, **AC**, or **DC**.
  Note: Refers to the power supply method of the sensor. If using Microlog dBX power supply, select ICP. If using an external power supply, select AC or DC according to the sensor specification.

- **Sensitivity mV/ SU** – An input box for the user to enter the correct sensitivity for the sensor used. In the example above, it is 100 mV/ g.

- **Apply to all** – Use this function to apply the same sensor sensitivity values. For example: the sensitivity of Ch1 is 100. When you tap **Apply To All**, the sensitivity of Ch2 to Ch4 will be changed to 100.

- **Save to Default** – Save the settings as default values.

- **Save Setup** – Opens the Save Dialog view. Enter **Name** and tap **Save** to save the setup.

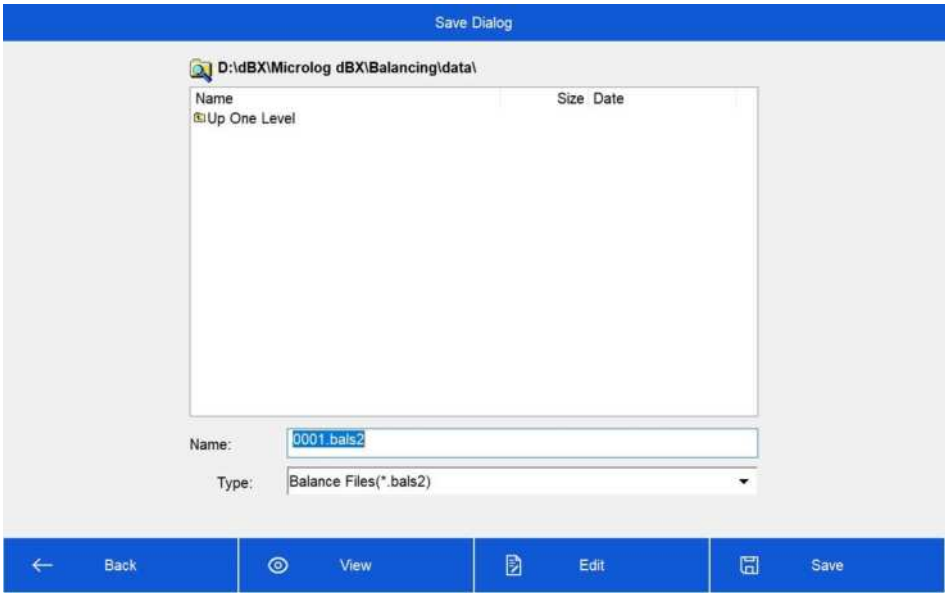


Figure 100 Save dialog – example

  - **Edit** – select a setup and tap the **Edit** menu button. From this menu you can **Cut**, **Copy**, **Paste**, **Delete**, or **Rename** a setup, or create a new folder.

Tab **Back** to return to the sensor setup view.

4.7 Vibration measurements

Select **Next** from the sensor setup page and the following view will appear. In the lower left part of the window, you will see the help text:

Start rotor. Press "Start" to read data

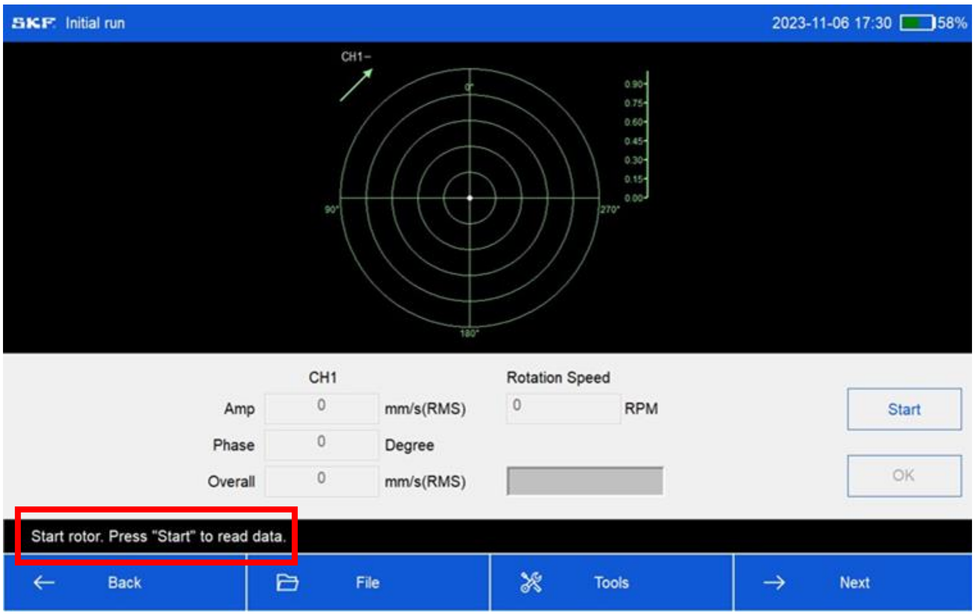


Figure 101 Initial run measurement – example

Tap **Start** to start the initial measurement.

Balancing a rotor is an interactive measurement process. Follow these steps and read the help text at the bottom of each screen to guide you through the process.

The rotation speed is now displayed. In the example below it is 3560 RPM.
At the bottom of the screen, the help text says:
Reading rotation speed, press "OK" to accept.

Navigate using the navigation keys and tap **OK**.

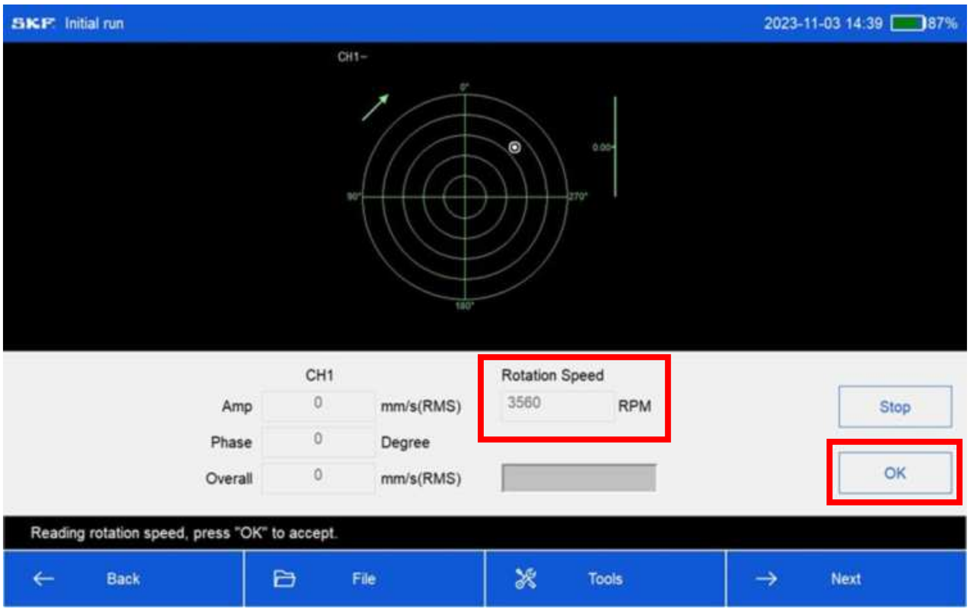


Figure 102 Rotation Speed – example

After you tap **OK**, notice that the amplitude, phase, and overall readings are displayed, and the 1X vibration data is shown in the polar plots.

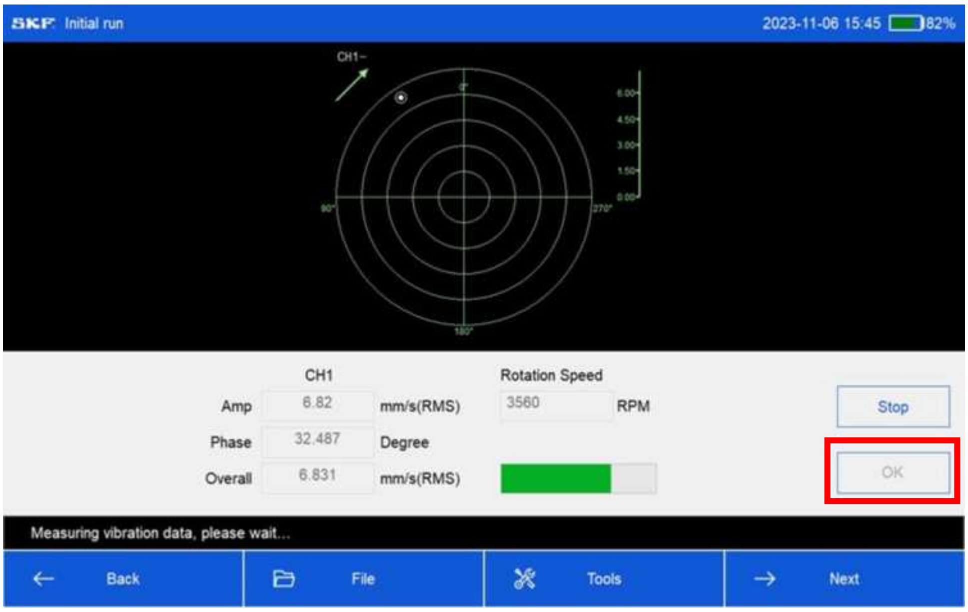


Figure 103 Initial Run measurement in progress - example

Tap **Next** to continue or **Redo** to repeat the test.

4.8 Manual entry

A **Manual Entry** option is available under the **Tools** menu for manual balancing data entry, allowing you to review and change balancing run data. Once selected, you can manually enter the rotation speed, vibration magnitude/overall, and phase angle for the balancing runs by selecting the entry.

Figure 104 Vibration measurement Manual Entry

4.9 Heavy spot estimation

If 1X coast down option is selected from the balancing setup menu, it will start to automatically make measurements of 1X and plot as the rotor will start decreasing its speed. The vibration level and phase of the 1X traces are shown under resonance, close to the 1st resonance and over resonance examples.

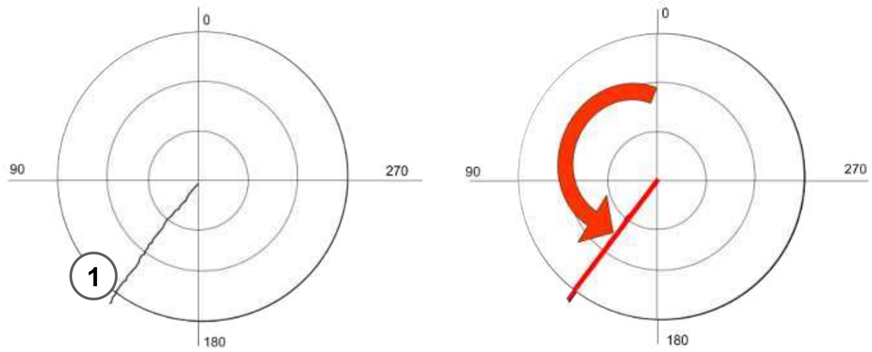
Identifying the heavy spot estimation after the 1X coast down is easy. Although you may derive the heavy spot after the trial weight measurements, the user can also use the 1X coast down to reduce the vibration levels with the trial weights. This technique prevents the user from potential danger by putting the trial weight in the wrong place, which could create excessive vibration, but the main point is to use the information to reduce the balancing time.

According to the physical rules, the 1X vibration **displacement** is in the same phase as the centrifugal force when the rotor is rotating at a speed which is lower than the 1st resonance of the structure, and the location of the centrifugal force is the same as the location of the heavy spot.
To show the location of the heavy spot, the coast down 1X measurement will always measure the vibration signal in the **displacement unit**.

4.9.1 Balancing with resonance

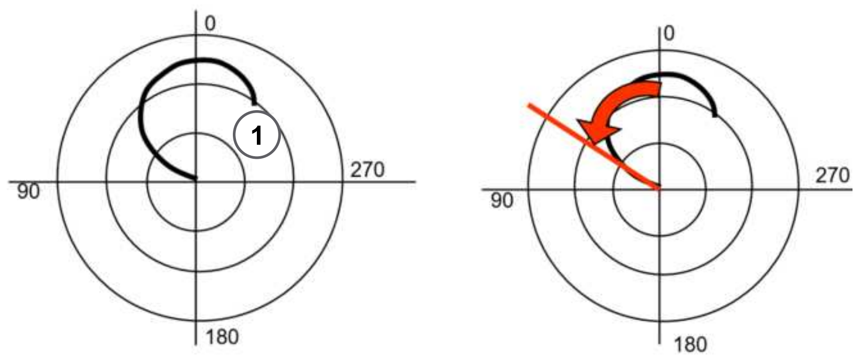
Sometimes a balancing job is performed at speed close to the resonance frequency. In such cases, the balancing coefficients may lead to poor balancing results because the vibration phase becomes unstable at or close to the resonance frequency.
The 1X polar plot is useful to identify resonance at or near the balancing RPM. It is recommended in this situation to use the 1X heavy spot from the polar plot as it would be more reliable in this situation.

Under resonance example
For a machine whose rotational speed is below the resonance, the 1X coast down trace would look like a straight line like in the example below. That means it has a constant phase during the coast down process. In this case, the constant phase is exactly the location of the heavy spot.



1. Operation speed

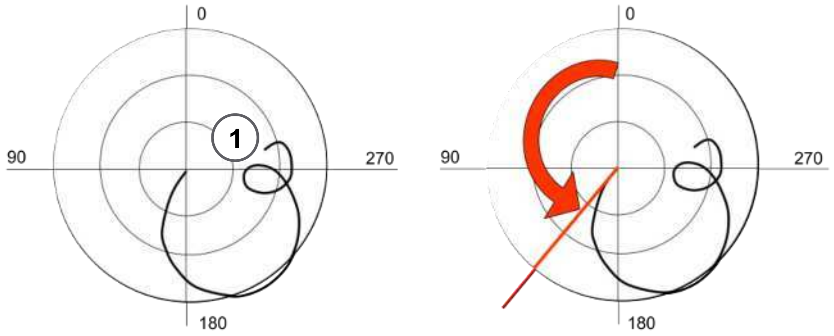
Close to the 1st resonance example
When rotational speed is close to the 1st resonance of the structure, the 1X coast down trace will look like a curve, as illustrated in the figure below. As the machine coasts down to a lower speed, the phase becomes constant again. This constant phase at low speed indicates the location of the heavy spot.



1. Operation speed

Above the 1st resonance example
When rotational speed is higher than the 1st resonance of the structure, the 1X coast down trace would look like a curve with some circles, as illustrated in the figure

below. As the machine coasts down to a lower speed, the phase becomes constant again. This constant phase at low speed indicates the location of the heavy spot.



1. Operational speed

4.9.2 Trial weight

Trial weights are necessary for the balancing process to balance your rotor. You can choose to add or subtract (cut) weight on the plane. If you are adding weight, you are physically attaching a weight to the rotor.

If you are removing weight, you are physically removing the weight (or material) from a rotor. This could be removing a set screw or by drilling a hole.

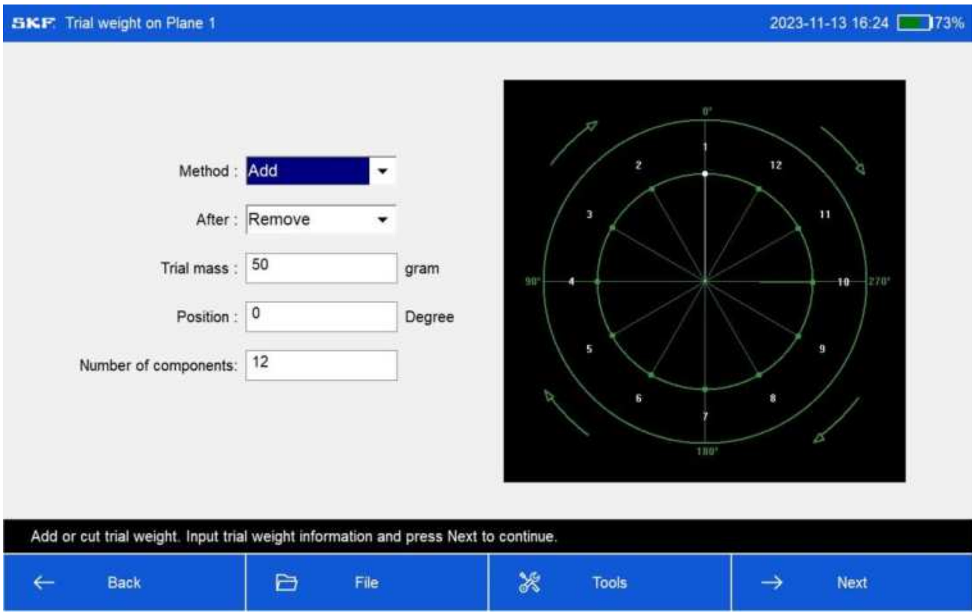
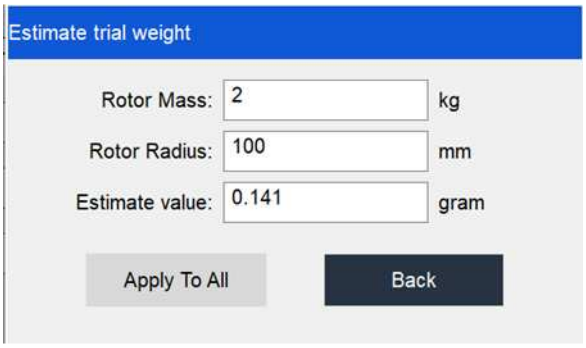


Figure 105 Trial weight on Plane 1

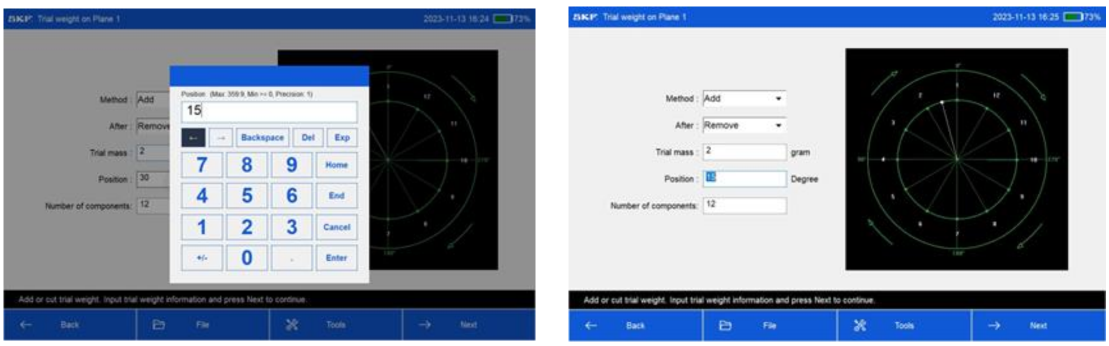
- **Method** – Select **Add** or **Cut**.
- **After** – Select **Remove** or **Remain** trial weight.
- **Trial Mass** – Tap the field and enter the trial weight.

To derive accurate balancing coefficients, select a trial weight which can make the vibration levels or phase(s) change effectively.

If you don't know how much trial mass needs to be added, select **Tools** > **Estimate trial weight** to make an estimation.



- **Position** – Select or enter the trial weight degree. There are two options to enter the position value, manually and by selecting a component.
  - Manual input – Tab the position field and enter the value. In the example below, the trial weight is set to 15 degrees, and the result is shown in the polar plot to right.



- *Figure 106 Trial weight – manual input*
  - **Selecting a component** - When the balancing weights are placed in fixed components, move the polar plot cursor to set the trial weight position. In the following example, component 2 of the 12 components selected, and the trial weight position is set as 30 degrees automatically.

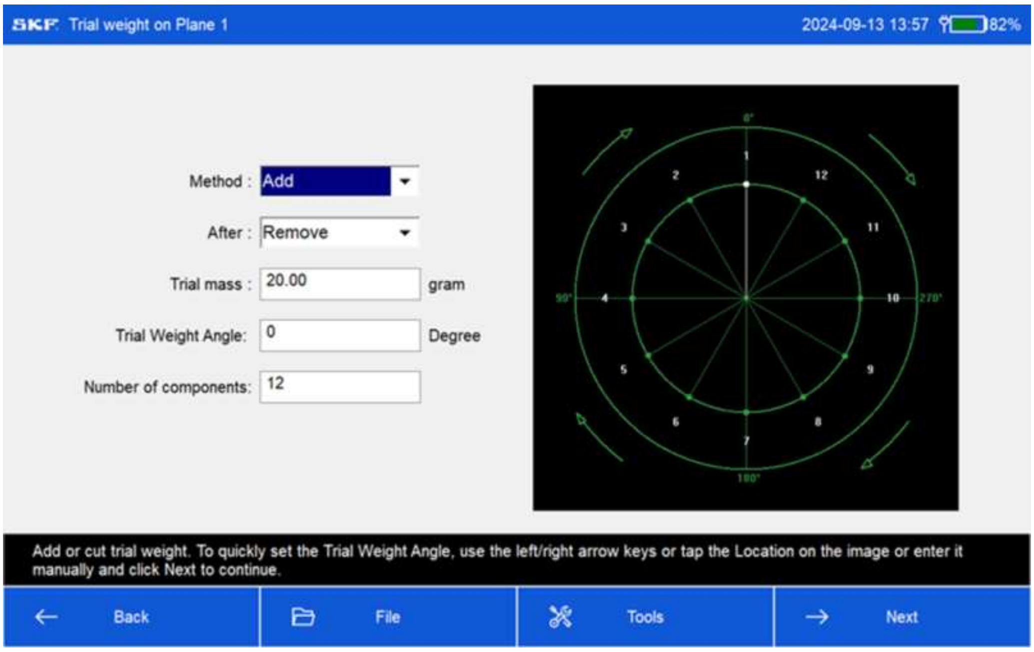


Figure 107 Trial weight with component 1 of the 12 components selected

- **Number of components** - Tap the field and enter the number of components.

4.9.3 Multiple Point balancing and projected vibration

When the number of measurement Points is larger than the number of balancing planes, multiple Point balancing is enabled. It will derive more than one set of balancing coefficients and may lead to different balancing mass instructions. However, when multiple Point balancing is enabled, the software will conduct a “least square” optimisation and minimise the sum of the square of vibration levels at different Points or directions. With this technique, it can ensure that all the measured vibration levels are reduced to the best possible conditions.

Example:
Suppose we select two Points for a single plane balancing job. The 1st vibration sensor is located at the vertical direction of bearing one (defined as Point one (CH1), and the 2nd sensor is located at the horizontal direction of the same bearing (defined as Point 2 (CH2).

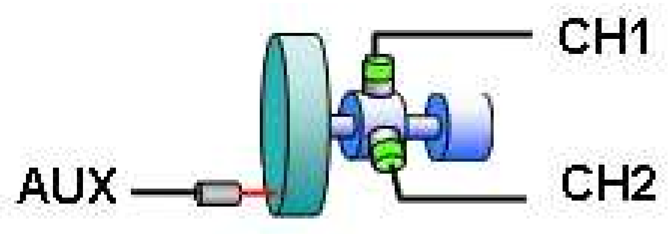


Figure 108 Multiple Point balancing – example

After the initial vibration and trial weight vibration measurements, the Balancing module then derives two balancing coefficients. One balancing coefficient from Point one and the other balancing coefficient from Point two.

Case A – **Balancing with coefficient from Point one**
If we balance the rotor by using coefficient from Point one, we expect to reduce the vibration of Point one from A to A1 (as shown in the plot below), which indicates the perfectly balanced condition.
In this case, the vibration of Point two based on the balance coefficient **one** is projected to change from B to B1, which shows an unsatisfactory improvement. These expected changes of vibration vectors are called **projected vibration**, and they are displayed in the following plots.

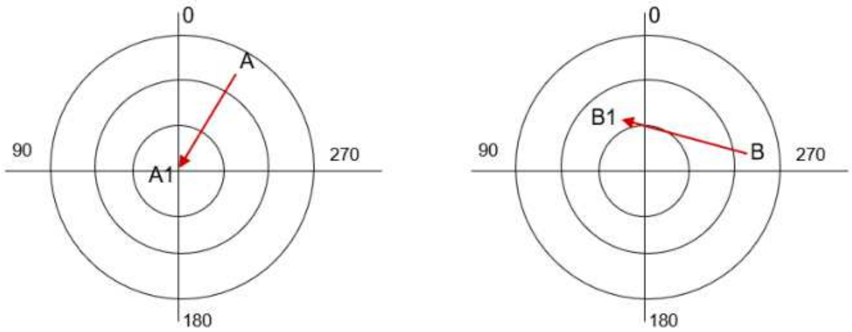


Figure 109 *Case A – Project vibration for Point one (left)*
Case A – Project vibration for Point two (right)

Case B – **Balancing with coefficient from Point two**
If we balance the rotor using coefficient from Point two (still using one coefficient), (using the standard balancing algorithms), we expect to reduce the vibration of Point two from B to B2, which indicates perfectly balanced. But in this case, the vibration at Point one changes from Point A to A2, which is not satisfactory. The projected vibration for this case is shown in the Case B Plot.

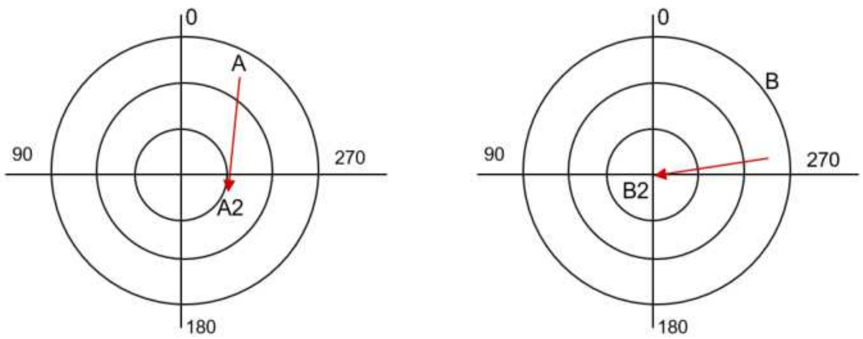


Figure 110 *Case B – Balancing with coefficient from Point two*

Case C – **Balancing with least squares calculation**
The method of least squares means that the overall solution minimises the sum of the squares of the errors made in the results of every single equation. Essentially the Least squares calculation is a method in balancing **to reduce the larger vibration** more quickly than the smaller vibration signals.

By conducting least squares calculations with both the balancing coefficients from Point one and Point two, we derive optimum balancing conditions as illustrated by the following projected vibration plots Case C and Case D. Note that both the vibration at Point one and Point two are reduced significantly **at the same time**.
In our Balancing module (multiple Point), least squares calculation is always used. The display will show the balancing information from least squares calculation.

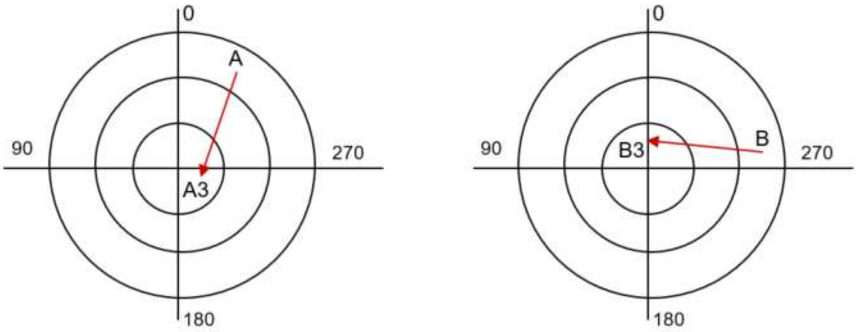


Figure 111 Case C – Projected vibration for Point one (left)
Case C – Projected vibration for Point two (right)

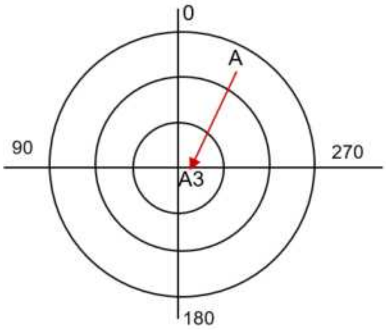
Case D – **Balancing with least squares calculation with weighting**
Adding a larger weighting factor to balancing reduces the vibration more at one Point. (Likewise, adding a smaller weighting will reduce the vibration less.)

What does weighting do?
Weighting involves emphasizing the contribution of some aspects of a phenomenon (or of a set of data) to a final effect or result, giving them more weight in the analysis. Rather than each variable contributing equally to the result some data are adjusted to contribute more than others.

Note: Weighting factors are numbered from 0 – 1.

For example, in Case C, suppose that you are not satisfied with the result and want to reduce the vibration more at Point one than at Point two.
With weighting you can apply a larger weighting factor to reduce the vibration more and achieve the desired effect.
In this example, factor 1 is applied to the vibration of Point one, and factor 0.5 is applied to the vibration of Point two.
Compared with Case C, the projected vibration of Point one is reduced more, as you can see from the following plot.

In practice, using the LSC calculation with weighting, adding a higher number will, in effect, reduce the vibration signal more.



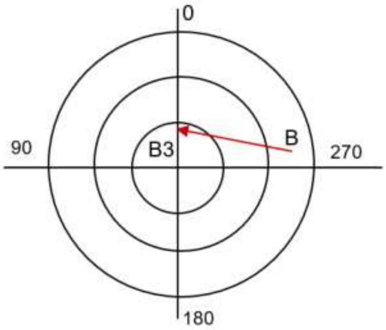


Figure 112 Case D – Projected vibration for Point one, with 1.0 weighting applied (left)
Case D – Projected vibration for Point two, with 0.5 weighting applied (right)

	Weighting applied to Point 1	**Weighting applied to Point 2**	**Least squares calculation**	**Multiple Point balancing**
Case A	Not applicable	Not applicable	No	No
Case B	Not applicable	Not applicable	No	No
Case C	1	1	Yes	Yes
Case D	1	0.5	Yes	Yes

4.9.4 Correction weight

After the vibration measurements with trial weight(s), the location and amount of the balancing mass required to balance the rotor are shown. This is called the correction weight. When the multiple Point balancing is not enabled, a single correction weight is calculated.

Balancing mass of single Point balancing
When the single Point balancing is selected, the balancing instructions are as below.

Figure 113 Correction weight – single Point

- **Mass** – The amount of balancing weight.
- **Phase** – The location of the balancing weight.
- **Residual imbalance** – The ratio of the corrected unbalanced mass to the first unbalanced mass. The first time Residual imbalance is 100%.

  For example: The first time, the unbalanced mass is calculated to be 100 grams. If the unbalanced amount is reduced to 50 grams after compensation, the Residual imbalance will display 50%. The smaller the percentage value is, the better the dynamic balance is. In other words, when the percentage rises, the compensation quality or direction is incorrect.

Balancing mass of multiple Point balancing

When the multiple Point balancing is enabled, the balancing instructions are as below.

Figure 114 Correction weight – multiple Point

Note that white circle in the polar plot shows the range of the balancing tolerance. For more information see section Tolerance setup.

- **Projected Vibration** – Click to show the expected 1X vibration of different measurement Points before and after the balancing mass is applied.

Figure 115 Correction weight – Projected Vibration – example

- **Edit** – Select to edit the balancing mass and phase.

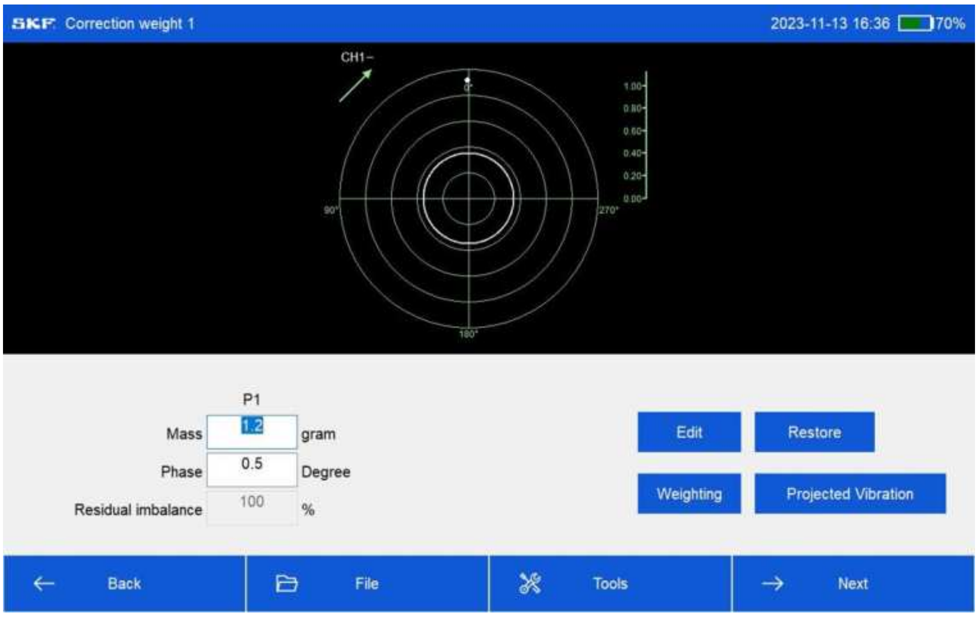
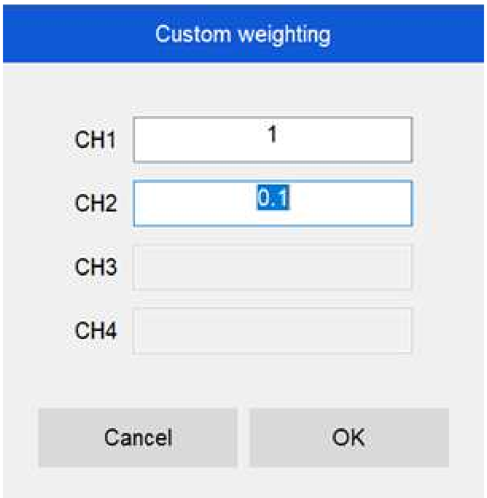


Figure 116 Correction weight 1 edit – example

After editing the balancing mass, from this view you can enter the **Projected Vibration** to see how the 1X vibration may change with the new balancing mass.

- **Restore** – Restore the balancing mass to the original values.

- **Weighting** – Add custom weighting number(s) to different measurements and recalculate the balancing results. For more information see section Multiple Point balancing and projected vibration.



- **Tools** – Tap to open the **Tools** menu.

Figure 117 Balancing Mass – Tools

  - **Add/Cut P1** – Changes the balance correction (plus/minus) method for plane 1.
  - **Tolerance** – Apply the ISO 21940 balance level. See section Home screen.
  - **Unequal radii** – Calculate the compensation mass when the test weight radius is different from the correction radius.
  - **Drill depth** – Calculate corrected (minus mass) drilling depth.
  - **Component** – Can calculate the mass required to increase or decrease in balance holes at different positions.
  - **Show text only** – Toggles between polar and displaying mass and phase values, in large text format.
  - **Vibration History** – View vibration measurements throughout the balancing process.
  - **Balance History** – View corrected mass values throughout the balancing process.

Description of timing and operating procedures of each tool

- **Add/Cut P1**– According to the rotor balance correction method, change the balance correction method at any time from adding mass to minus mass;

from minus mass to adding mass, the balance angle also changes accordingly.

Figure 118 Correction weight – example

- **Tolerance** – For more information, see section Tolerance setup. If the balancing tolerance is set, the range of tolerance will be shown with a white circle in the middle of the radar plot, with which you can tell whether the residual imbalance meets the selected ISO standard or not. Use **Zoom in** or **Zoom out** keys to adjust the radar display when needed.

- **Unequal radii** – When the trial weight radius is different from the correction radius, this function must be used to correct the correctness of the unbalanced mass.

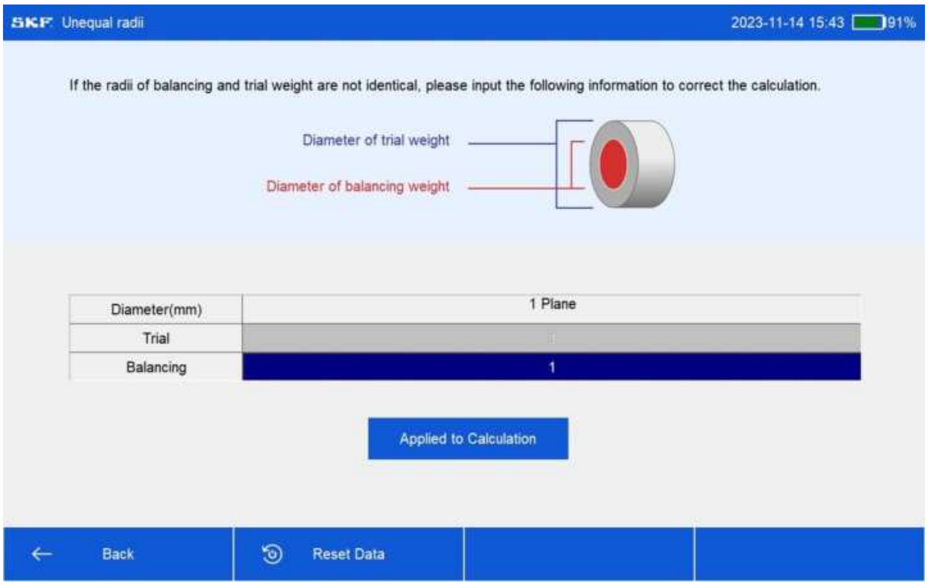


Figure 119 Unequal radii view – example

Tap the respective column in the table, and using lowercase on your keyboard, input the correct diameter of trial weight (**Diameter(mm)**) and Diameter of balancing weight (**Balancing**).

Tap **Applied To Calculation**, and the residual amount will be recalculated according to the changed size.

- **Drill depth** – When the balance correction method is to remove the mass, use this function to calculate the drilling depth according to the material and the diameter of the drill.

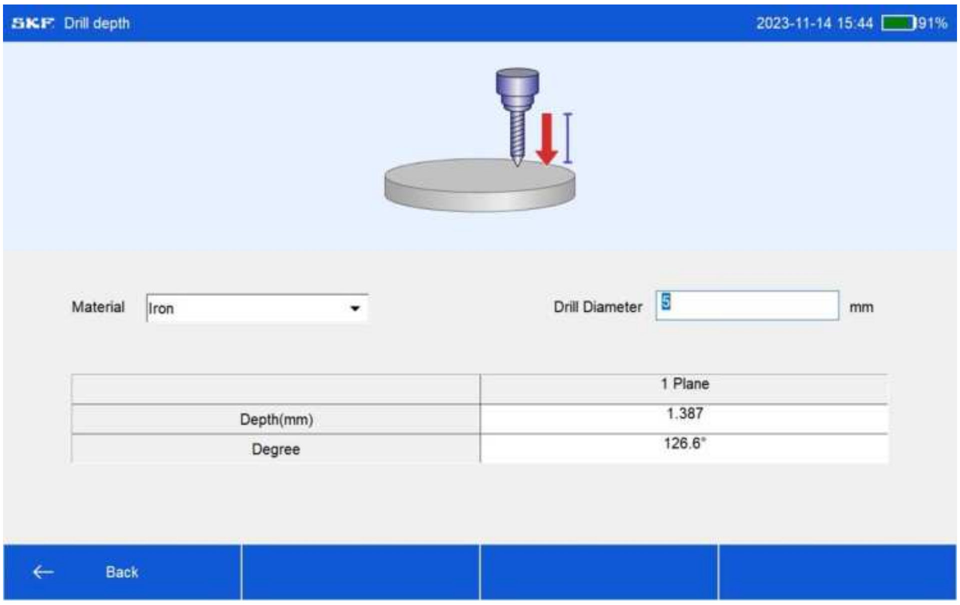


Figure 120 Drill depth view – example

After selecting the material and entering the drill diameter, the system will calculate the drilling depth and the drilling phase.

Note: There are currently three materials: iron, copper, and aluminium.

Tap **Back** to return to the previous screen.

- **Component** – If a counterweight hole has been reserved on the calibration plane, use this function to calculate the balance mass required to increase or decrease at the position of the counterweight hole.

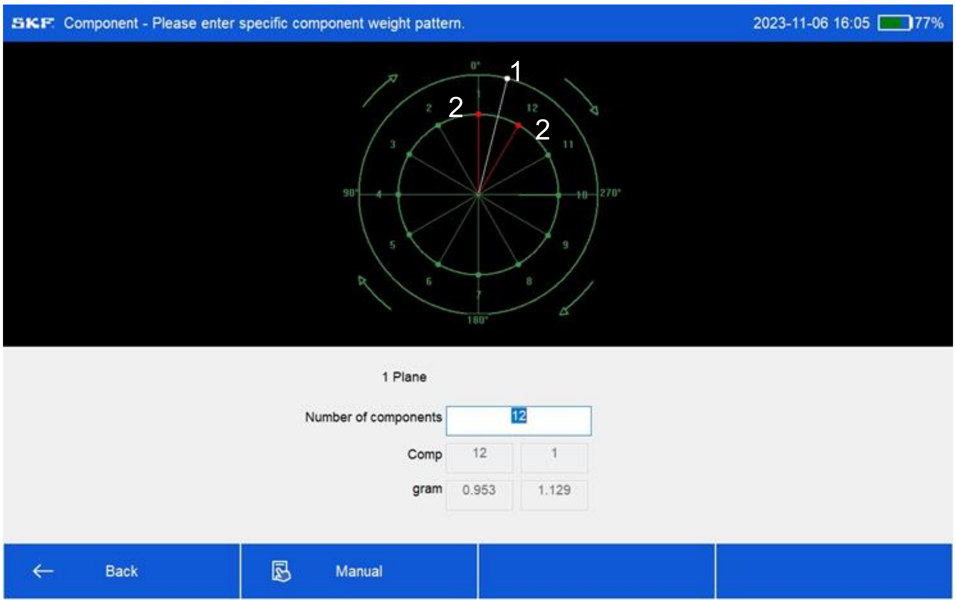


Figure 121 Component weight – example

1. Location of the balancing weight
2. Locations of the selected balancing components

Input the total number of counterweight holes and the software will automatically calculate the position and mass of the counterweight holes. If the position of the counterweight hole cannot add or subtract the unbalance mass, you can use the manual mode to modify the position of the counterweight holes.

Tap **Manual** and tap the **Component** table box. Input the hole position and tap **Enter**. The mass of the hole is now updated.

Note: When selecting the balancing components manually, always make the balancing weight (white line on the radar plot) between the positions of the components.

Tap **Back** to return to the previous screen.

- **Show text only** – The radar chart can be turned off and corrected mass and angle displayed in a larger font.

Figure 122 Correction weight – Large font example

- **Vibration History and Balancing History** – During the correction process, all vibration and balance measurements are recorded. Therefore, the entire balance and vibration history can be viewed at any time.
  Switch between vibration and balancing history view using the ◄ and ► buttons on the left and right below the radar graph.

  Note: When the Data ID begins with **V**, it represents vibration data, and when it begins with **M**, it represents balancing data.

Figure 123 Vibration History view –example

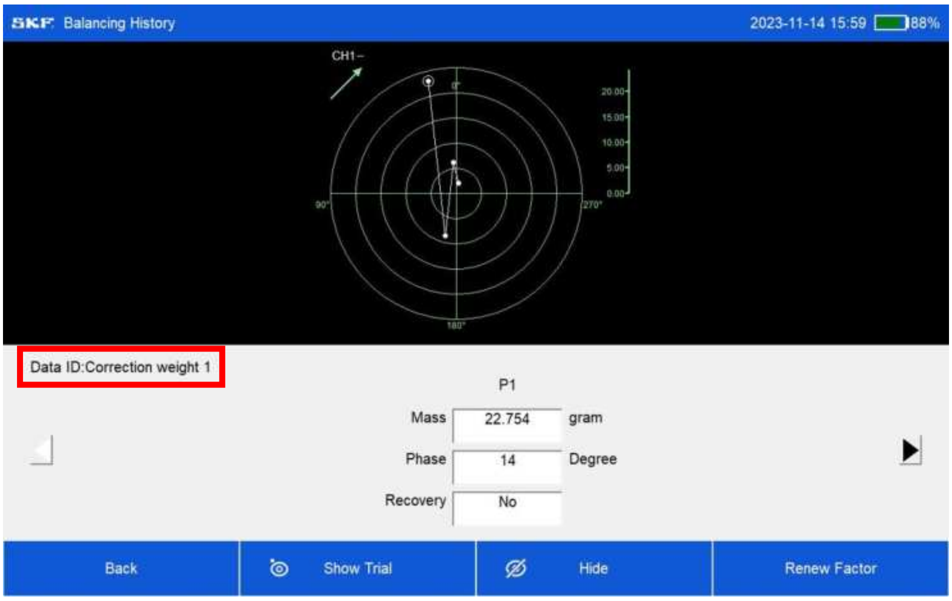
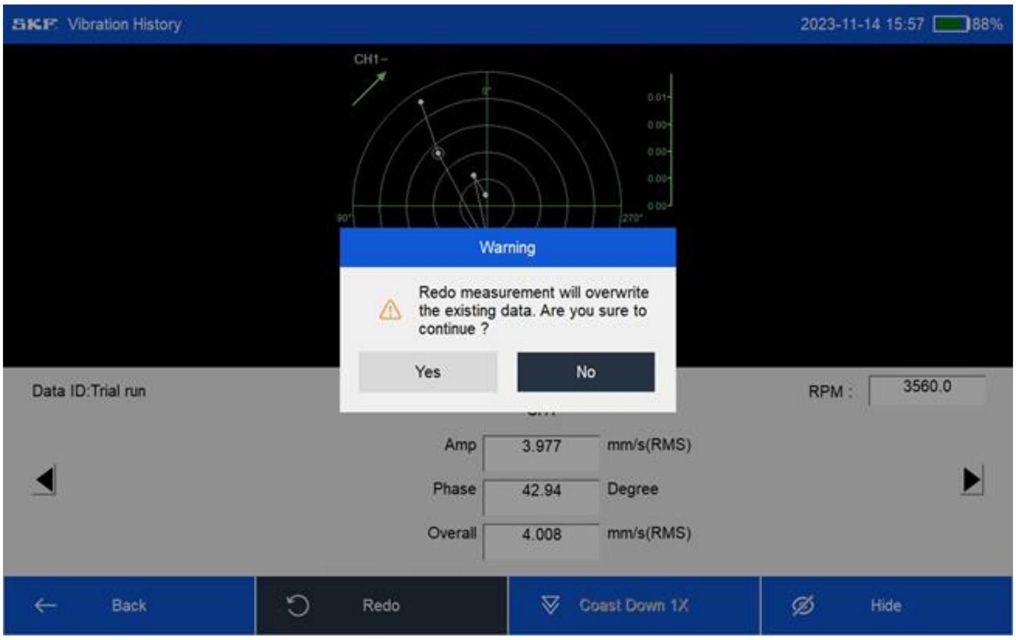


Figure 124 Balancing History view – example

- **Back** – Return to the original polar plot screen displaying residual quantity.
- **Redo** – Return to previous vibration (balance) history.
- **Coast Down 1X** – Switch the speed down state 1X polar plot. If the data is not available, it will be displayed in white**.**
- **Hide/Show** – Toggle to show or hide single-stroke vibration (balance) history.
- **Show Trial** – Display the trial run for comparison.
- **Redo** – Remeasure the vibration of a specific amount of data from the Vibration history. Only at V0, Vt can be used. If it is selected, a window will pop-up to confirm whether to redo the data. If it is confirmed to be redone, the measurement process will be started from the data.



Sometimes, the balancing coefficients are derived from very high initial vibration and are very nonlinear, which would cause inaccurate balancing results. Selecting a correction weight data with a smaller vibration level to renew the factors can improve the accuracy of balancing results.

- **Renew Factor** – Select from Balancing History to recalculate the balancing coefficients. Sometimes, the balancing coefficients are derived from very high initial vibration and are very nonlinear, which would cause the balancing results not accurate. Selecting a correction weight data with a smaller vibration level to renew the factors can improve the accuracy of balancing results.

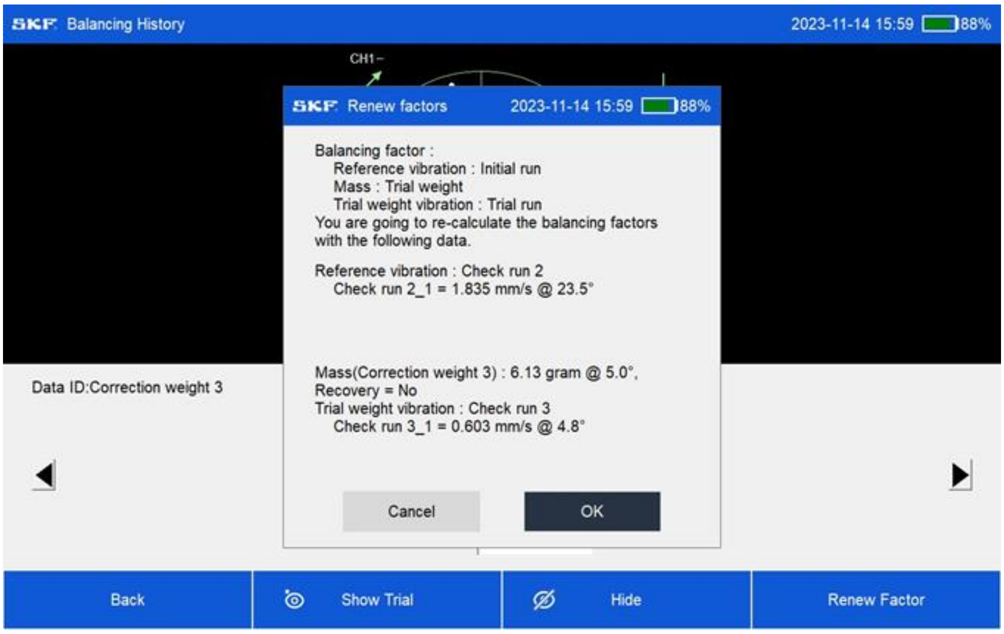


Figure 125 Renew factors – example

- **Generate report** – Tap to enter the Generate report view where you can enter the general information for the report.

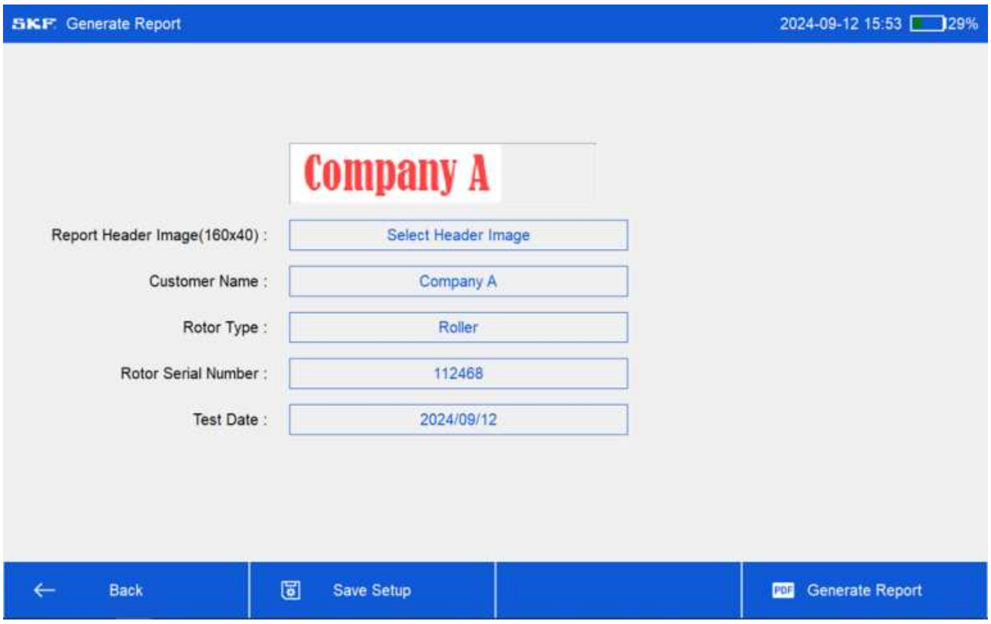


Figure 126 Generate report – example

  - **Report Header Image (160×40)** – Tap **Select Header Image** and select an image. The image will be shown in the top left corner of the report as a company logo. Recommended image size is 160×40px.

- **Customer Name** – Tap and enter the customer's name.
- **Rotor type** – Tap and enter the rotor type.
- **Rotor Serial Number** – Tap and enter the serial number of the rotor.
- **Test Date** – Tap to select the test date.
- **PDF Generate Report** – Tap to generate report as PDF.

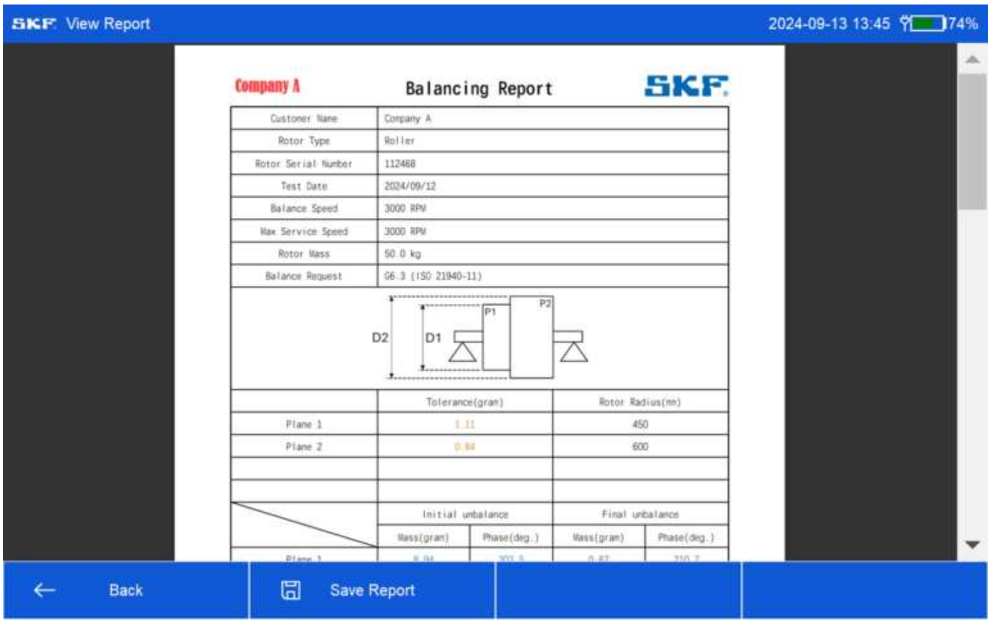


Figure 127 Generated PDF report – example

- **Save Report** – Review the report and tap **Save Report** to save the file. By default, the file is saved to D:\dBX\MicrologdBX\Launcher\. After saving, you can open the file from the dBX Documents application on the home screen.

  To save the file to an external USB disk, tap **Save report** and select the external disk to save the report.

4.9.5 Overhung balancing

Due to the high level of couple effects, balancing an overhung rotor with standard single-plane or two-plane techniques is often challenging. This module implements an alternative technique to enhance the balancing of overhung rotors. This balancing algorithm treats the imbalance weights as static imbalance and couple imbalance, respectively.
You'll find that balancing an overhung rotor is almost the same as balancing a standard two-plane rotor, with the main difference being the method of applying trial weights.

Plane names and sensor locations

After selecting overhung balancing, follow the illustration plot to set up your sensor connections. Note that Plane 1 is the inboard plane, close to the bearing, while Plane 2 is at the other end. You can also select 4-point balancing to reduce vibrations at other points simultaneously.

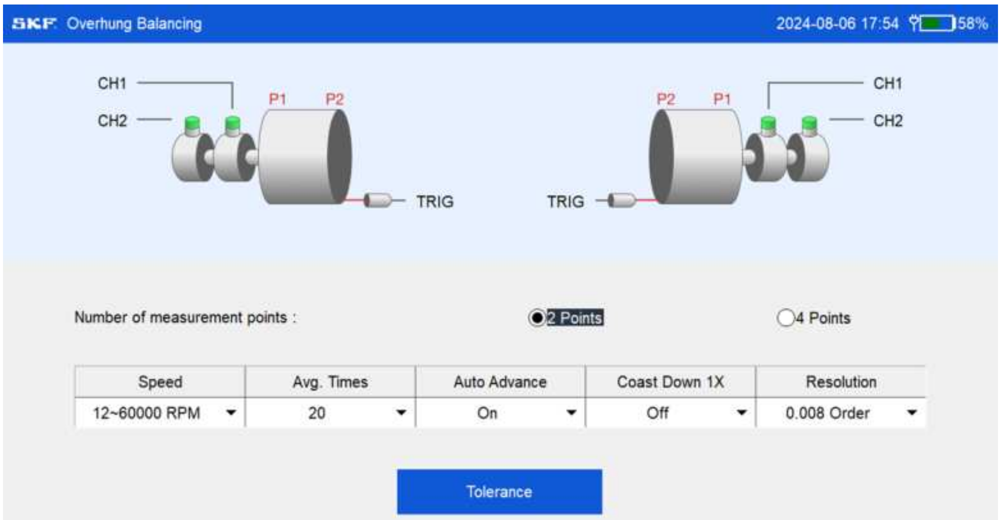


Figure 128 2-points overhung balancing – example

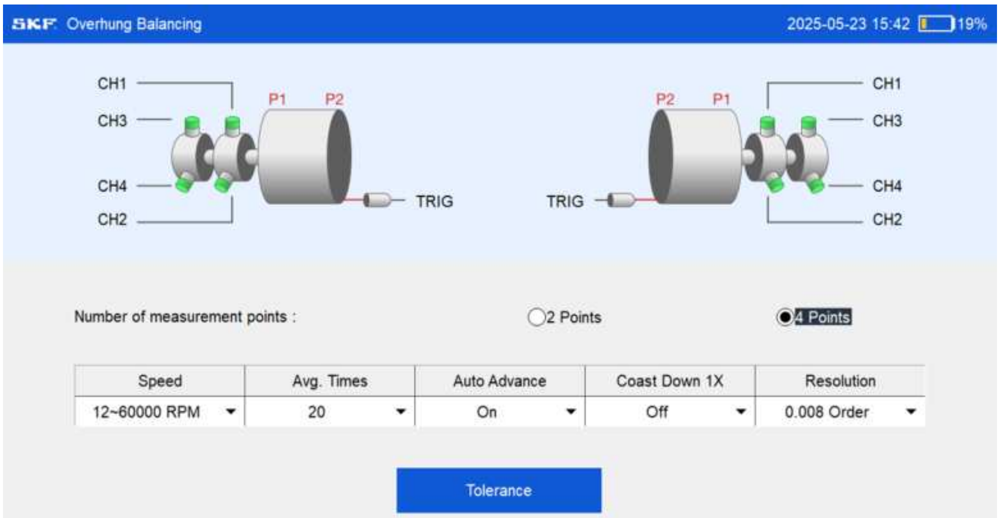


Figure 129 4-points overhung balancing – example

Adding trial weight to Plane 1

After the initial vibration measurement, the program will prompt you to add a trial weight to Plane 1. In the illustration, a 1 gram trial mass is added at 0 degree and will be removed after the vibration measurement. Note that Plane 1 is the inboard plane, as previously described.

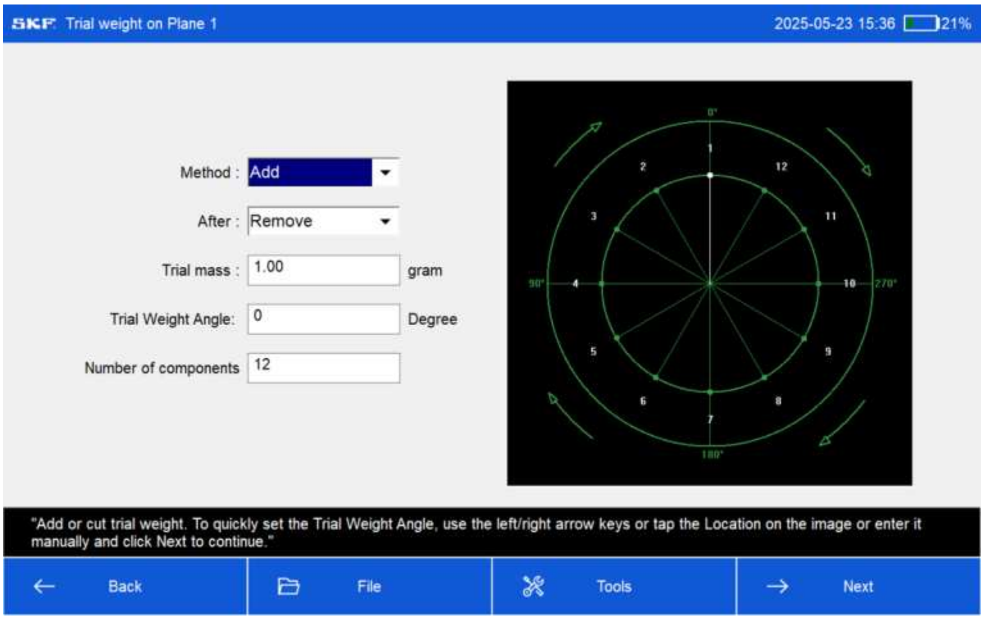


Figure 130   Trial weight on Plane 1 – example

Adding a couple trial weight to the rotor
After measuring the vibration with the trial weight on Plane 1, the software will guide you to add a couple trial weight to the rotor. A couple consists of a pair of equal-weight masses placed on Plane 1 and Plane 2, respectively, 180 degrees apart.

Correction for unequal diameters
Before setting up the couple trial weight, a window will prompt you to select **Equal** or **Unequal** for the diameters of Plane 1 and Plane 2. If **Equal** is selected, tap **Next** to proceed to the next view.

Figure 131   Overhung Trial Diameter Setup Equal – example

If **Unequal** is selected, enter the diameters of both planes before proceeding to the next view. This information will be used to accurately calculate the couple effect.

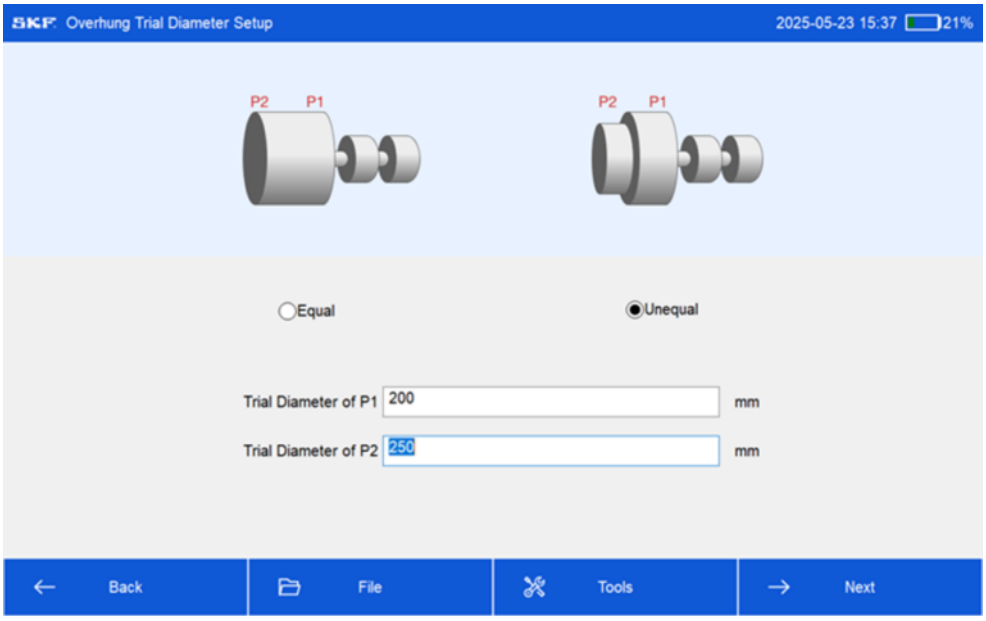


Figure 132 Overhung Trial Diameter Setup Unequal – example

Couple trial weight setup

The following view shows the proper way to add the trial weight to Plane 1 and Plane 2.

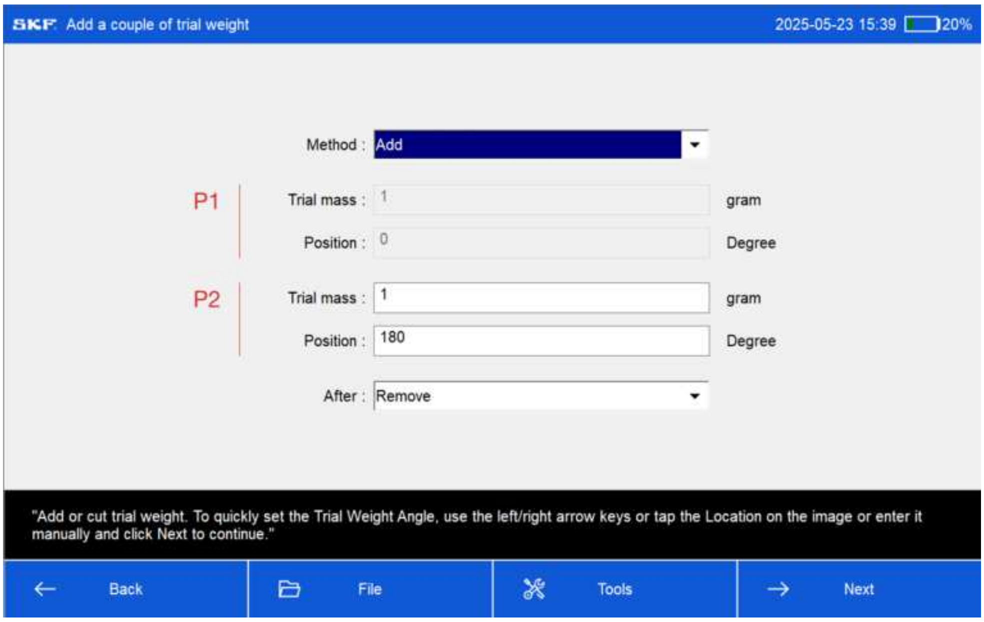


Figure 133 Add a couple of trial weight – example

In this example, the trial weight on Plane 1 was set as **After** -> **Remove**, hence it can remain on plane 1 since it is removed and then added back to Plane 1 as a portion of the couple. Therefore, only the trial weight on Plane 2 needs to be added to create a couple trial weight.

In the **P2** section of the view, the trial weight on Plane 2 is shown. It has the same mass as the trial weight on Plane 1 and is positioned 180 degrees apart.

You can manually adjust the position and amount of the trial weight on Plane 2. This will automatically update the settings for Plane 1 as well.

After adding the couple trial weight to the rotor and taking vibration measurements again, the software will guide you on how to add balancing weights to Plane 1 and Plane 2 to balance the rotor. Repeat this process until the desired balancing quality is achieved.

4.10 Three balancer balancing

From the home screen, select **3 balancers** to initiate the procedure for balancing a rotor with three permanently installed, equal-weight balancers. This method is typically used to balance the grinding wheel of a grinding machine.

The procedure for balancing with 3 balancers is similar to a regular balancing task. It includes the initial run, trial weight application, and applying correction weights to the rotor.

However, this method differs as it requires adjusting the angular positions of the 3 balancers to achieve the desired balancing effect on the rotor.

Reset the balancers
Before starting the balancing process, set the positions of the 3 balancers to **0**, **120**, and **240** degrees. This resets their combined effects to zero, allowing measurements of imbalance to be taken exclusively from the rotor itself.

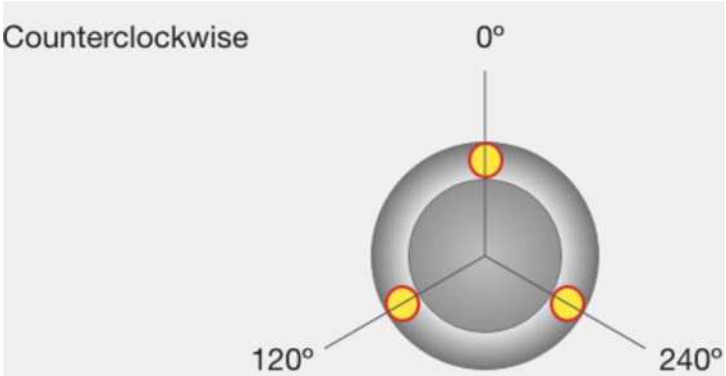


Figure 134 3 balancer positions – example

Apply a trial weight to the rotor

After the initial run, the software will prompt you to apply a trial weight to the rotor by adjusting the balancer from **0** degrees to **30** degrees. You can also enter a different angle to customize the trial weight setting.

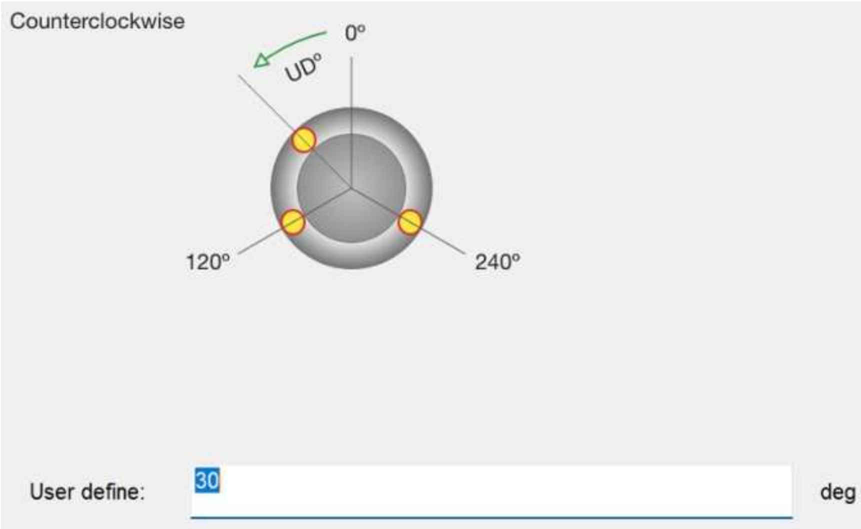


Figure 135 3 balancer positions – rotor degrees example

Apply correction weight

After measuring the vibration with the trial weights, the correction weight information and the angular positions of the 3 balancers will be calculated and displayed. Adjust the three balancers to the specified angular locations and restart the rotor to evaluate the balancing effect.

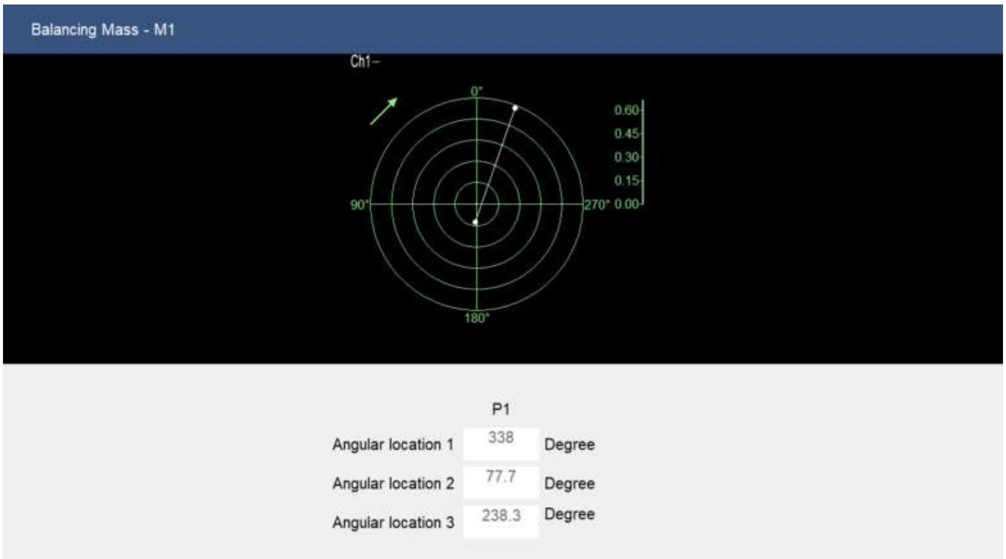


Figure 136 Angular location – example

Fine tune of the balancers

After applying the correction weight to the rotor, restart it and take vibration measurements to evaluate the balancing effect. The software will then prompt you to fine-tune the balancer locations to further improve the balancing quality. For example, based on the plot below, the software recommends adjusting the angular positions of the 3 balancers by **-2.7** degrees, **+0.6** degrees, and **-6** degrees. Repeat these adjustments until the balancing quality meets expectations.

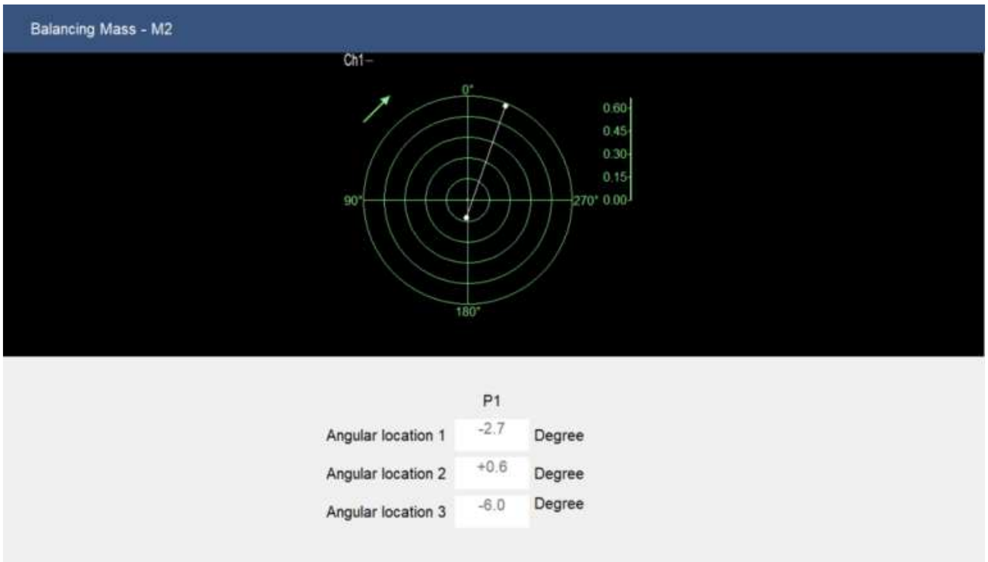


Figure 137 Angular location – fine tune example

5 Utility module

5.1 Introduction

Before using the software, please read and understand the separate Microlog Analyzer dBX Quick Start Guide.

The **Utility** module allows you to review the system information. The displayed information includes **Product information**, **Software information**, **Windows information**, etc.

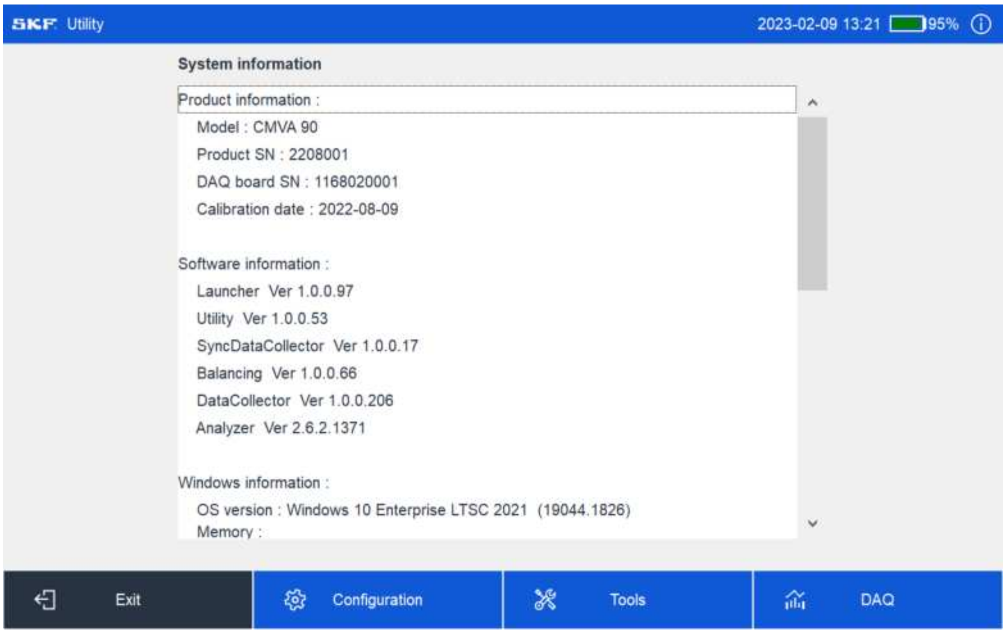
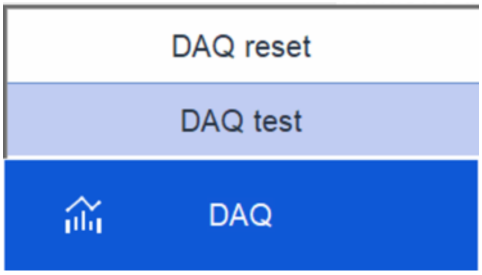


Figure 138 Utility module view - example

5.2 DAQ (Data Acquisition)

Tap to enter data acquisition hardware testing mode.



- **DAQ reset** – The reset function is to disconnect and reconnect the DAQ, through GPIO (general-purpose input/output). You can perform the DAQ reset to restore hardware connection when the Microlog dBX loses connection to the DAQ hardware.

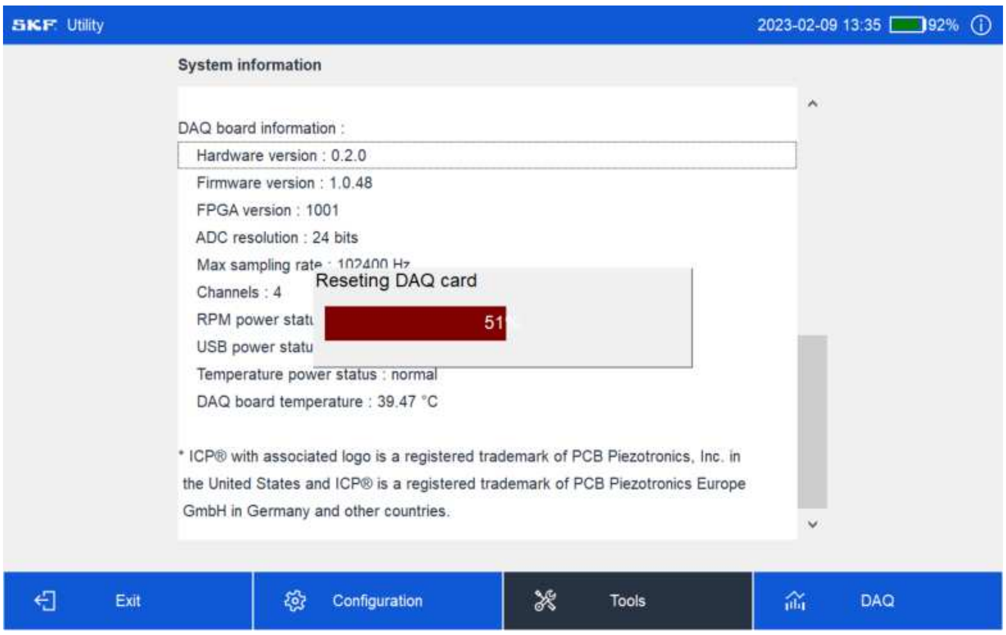


Figure 139 DAQ reset – example

- **DAQ test** – Switch to the test page to perform a simple function test. On the Test page, you can set the bandwidth, the number of analysis channels, voltage coupling and input range. This page provides viewing time waveforms.
  The test function is a useful tool to verify whether your hardware is working properly.
  You can set the bandwidth, select measurement channels, coupling mode and input range to conduct measurements.

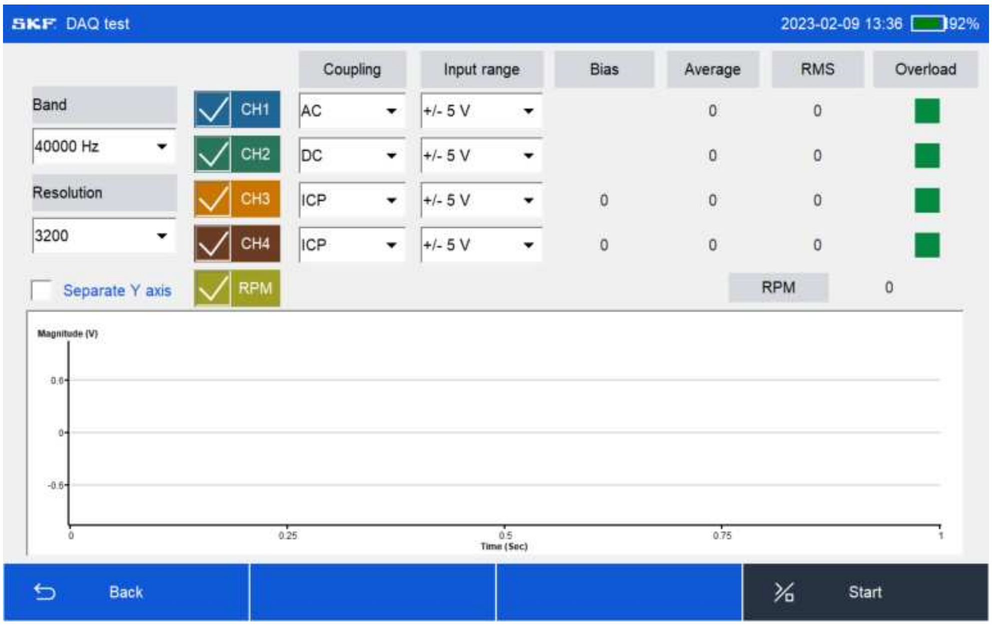


Figure 140 DAQ test – example

Tap **Start** to proceed with data acquisition.

Figure 141 Acquisition proceeds – example

Tap **Stop** to stop the data acquisition.

5.3 Configuration menu

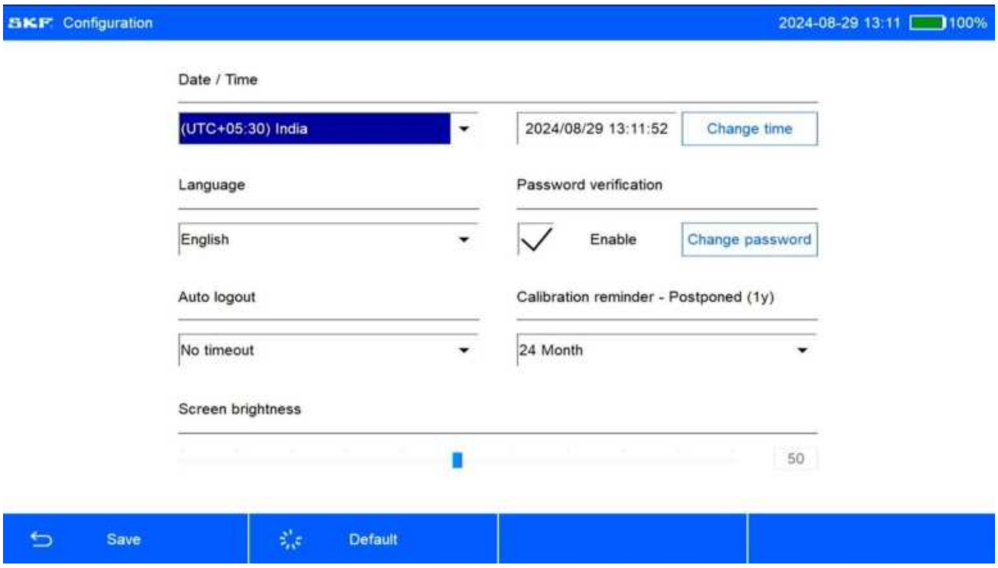


Figure 142 Configuration view – example

- **Date / Time** – Select time zone from the dropdown menu. Tap **Change time** and in the pop-up, window set the date and time. Tap **OK** to save the change.

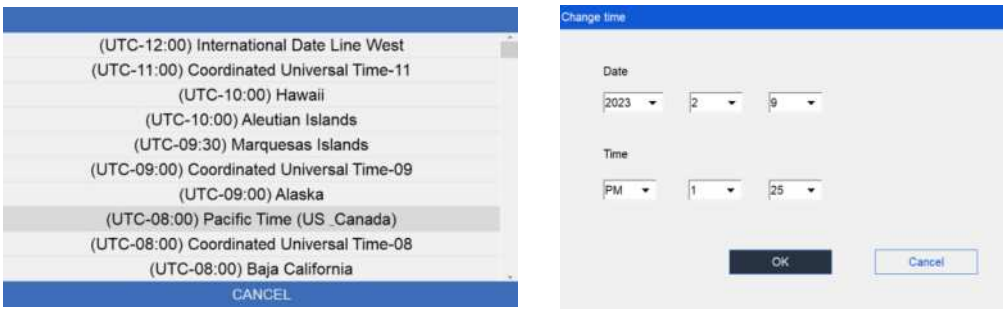


Figure 143 Change date and time – example

Tap **Default** to restore to the default settings.

- **Language** – Select a language from the dropdown menu. The selected language will be applied to all modules.

- **Password verification** – Manage password settings.
    - **Enable** / **Disable** – If you uncheck the **Enable** option, a pop-up window will ask you to confirm this action. Select **Yes** to disable the startup password or **Cancel** to exit.

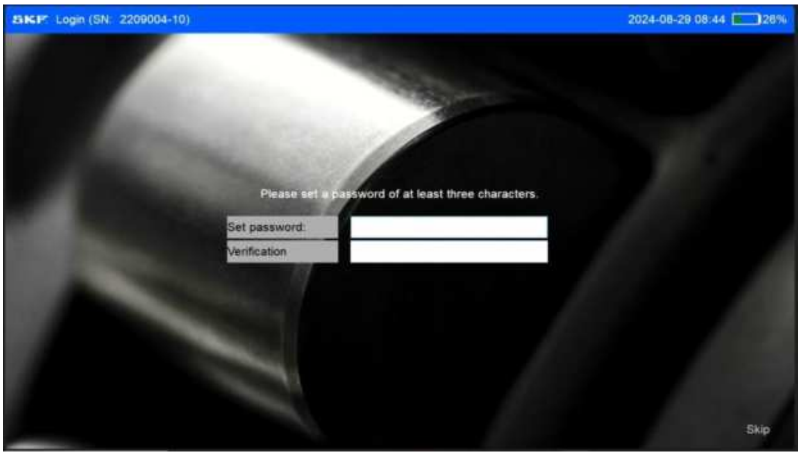
- **Change password** – Select to change the saved password. A pop-up window will ask you to restart the device and set a new password. To change the password:

1. Tap **OK**.

2. Tap the **Password** field, enter the old password and press **OK**. The device will restart.

3. Tap the **Set password** field to display the keypad and enter the password.

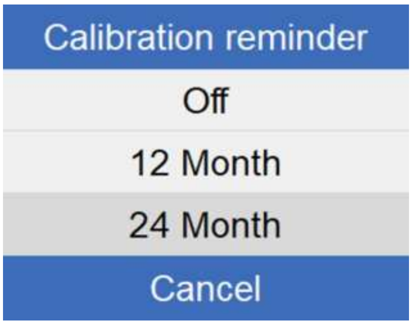
4. Re-enter the password in the **Verification** field.



- **Auto logout** – Select an auto-logout time**.**
  To disable the auto logout function, select **No timeout**.
  To set an auto logout time, select **5 minutes** or **15 minutes**.

- **Screen brightness** – Use the slider to adjust the screen brightness.

- **Calibration reminder** – Set the time for calibration reminder. To ensure your device functions properly and performs accurate measurements, we recommend calibrating it at least once every two years.
  Select **12** or **24** months calibration period reminder. Select **Off** to turn off the reminder.



When there are 30 days remaining until the set calibration date, the system will start displaying reminders about the time left until the calibration due date in the bottom-right corner of the home screen.

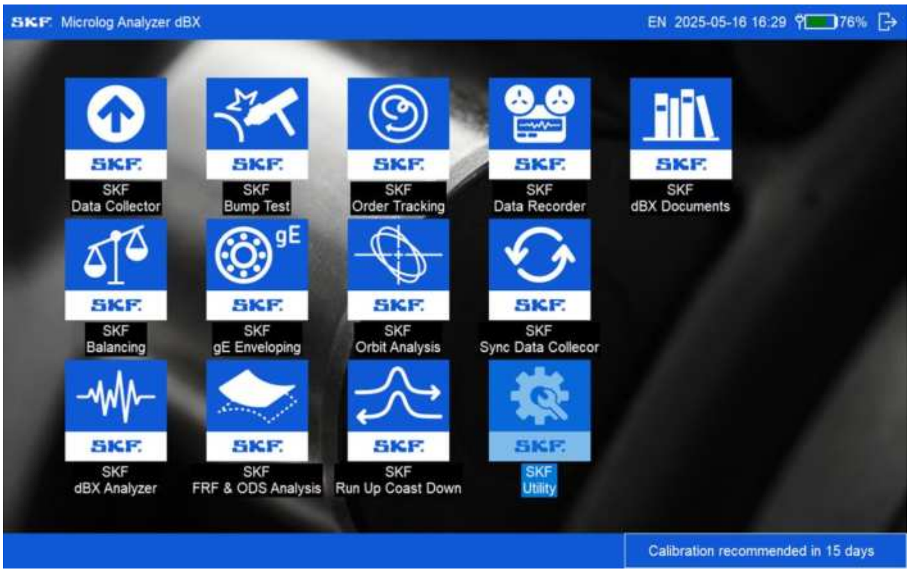


Figure 144 Calibration reminder message – example

To postpone the calibration date, tap on the reminder message and from the pop-up window select **OK.** Tap **Cancel** to exit the dialog without postponing the date.

5.4 Tools menu

- **Program list** – View the software modules, their released version and authorization status.

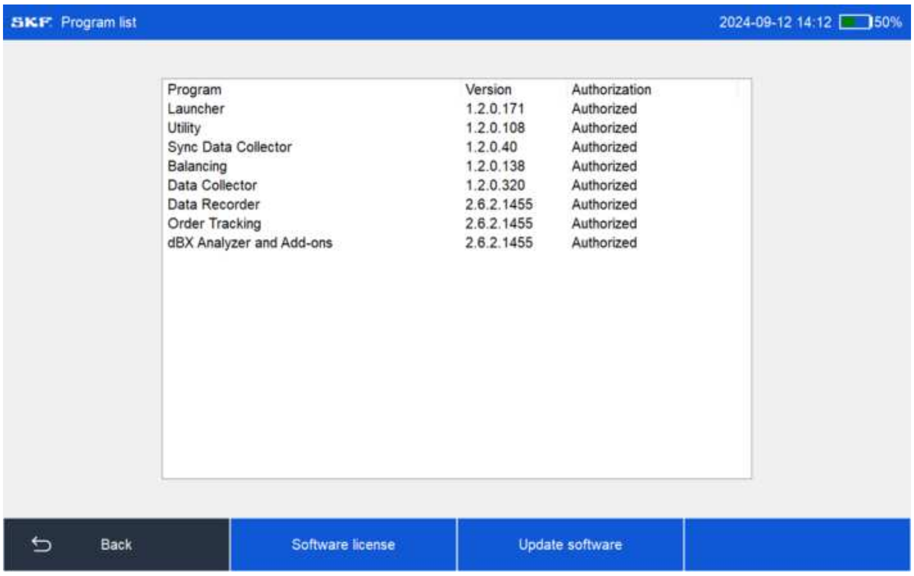
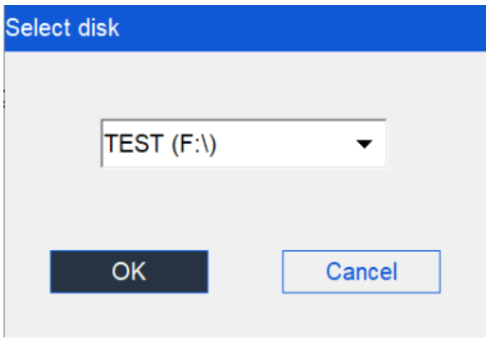


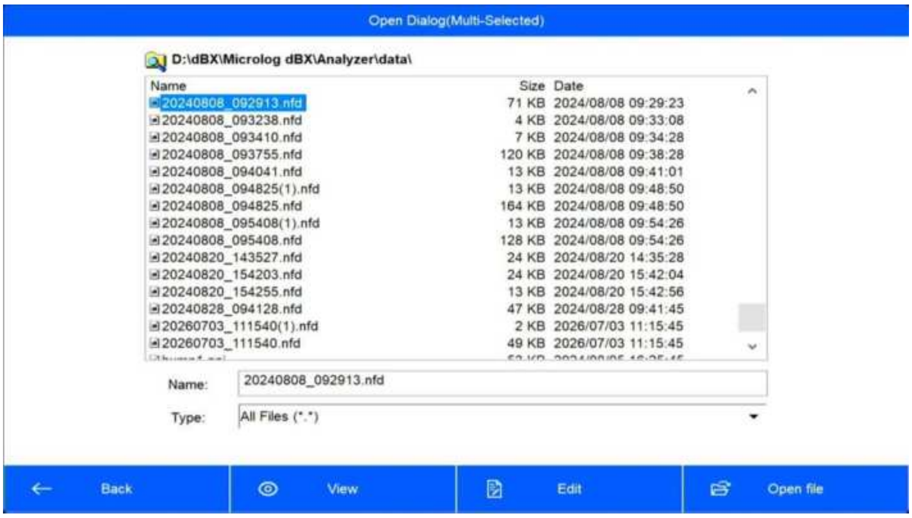
Figure 145 Program list view – example

  - **Update software** – To perform a software update:

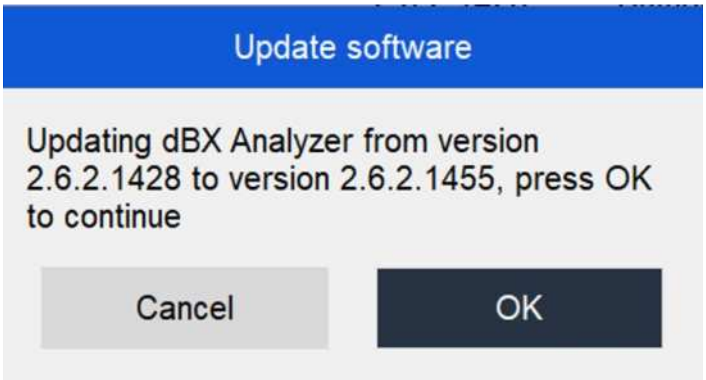
1. Tap **Update software** and from the dropdown select a disk (external USB device) containing the update file.



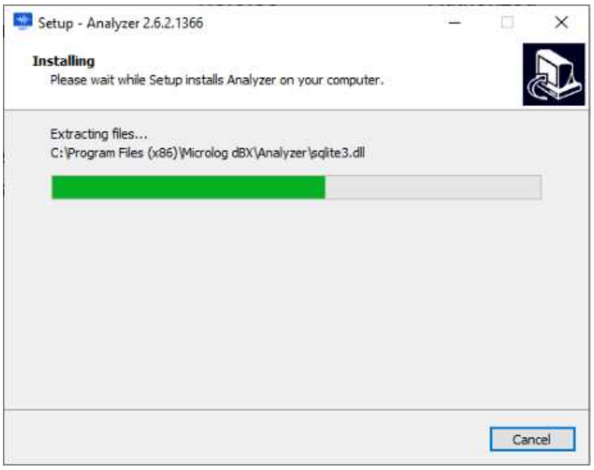
2. Select an installation file. Note that the installation file has *.exe* extension.



3. Tap **OK** to run the installation file.

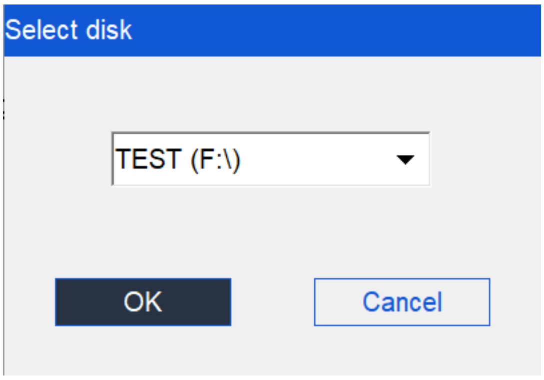


4. In the pop-up window you can follow the installation process. Tap **Cancel** to cancel the installation.

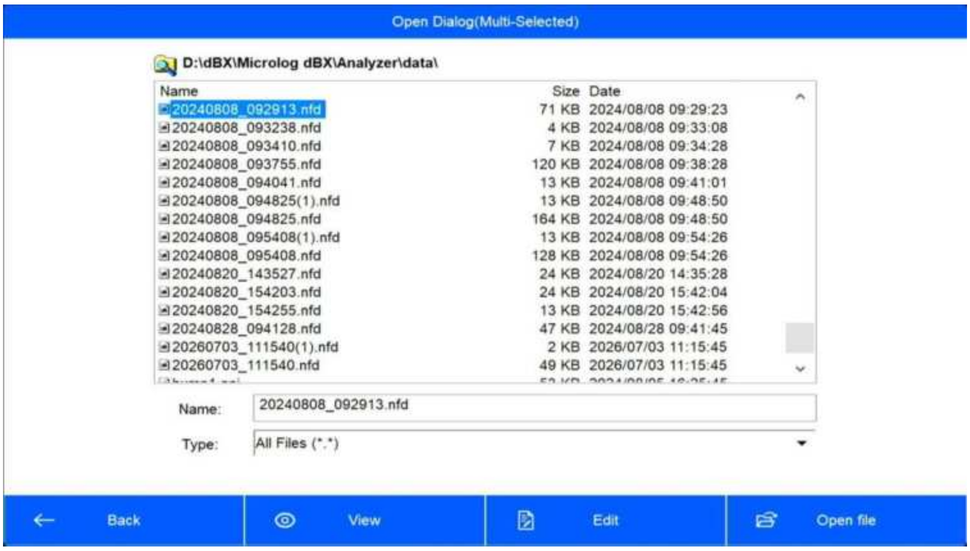


- **Software licence** – To install software module authorisation: This is mainly for upgrading from the M-Pack to the F-Pack features. Contact SKF's Technical Support Group TSG for upgrade purchase.

1. Tap **Software license** and from the dropdown select a disk (external USB device) containing the update file.



2. Select the license file from an external USB device.
   Note that a licence file has *.auh* extension.

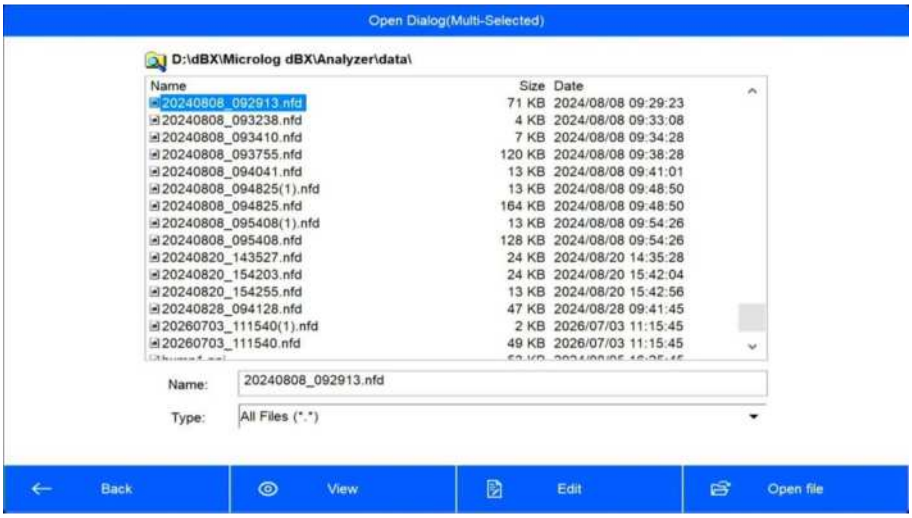


3. Tap **OK** to run the installation file.

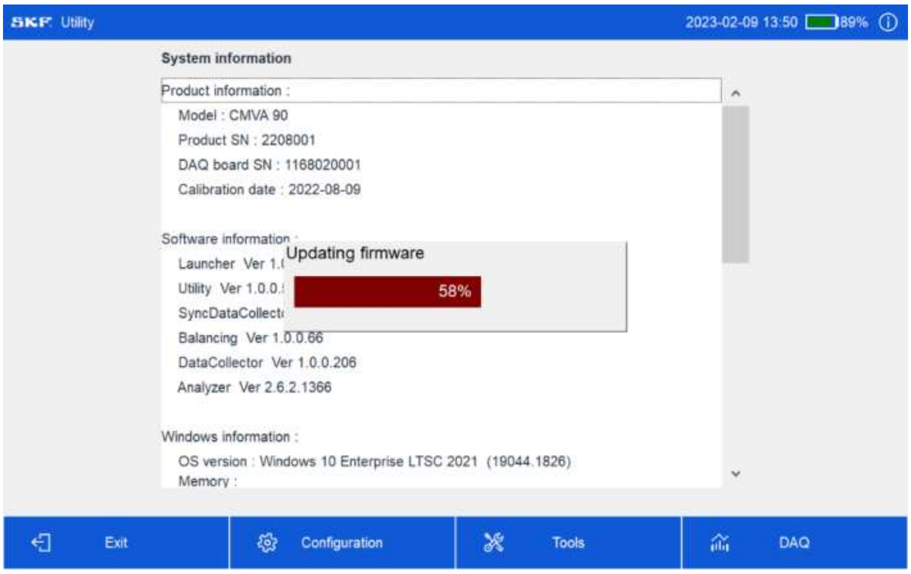
4. In the pop-up window you can follow the installation process. Tap **Cancel** to cancel the installation.

- **Update firmware**

1. Tap **Update firmware** and from the dropdown select a disk (external USB device) containing the update file.



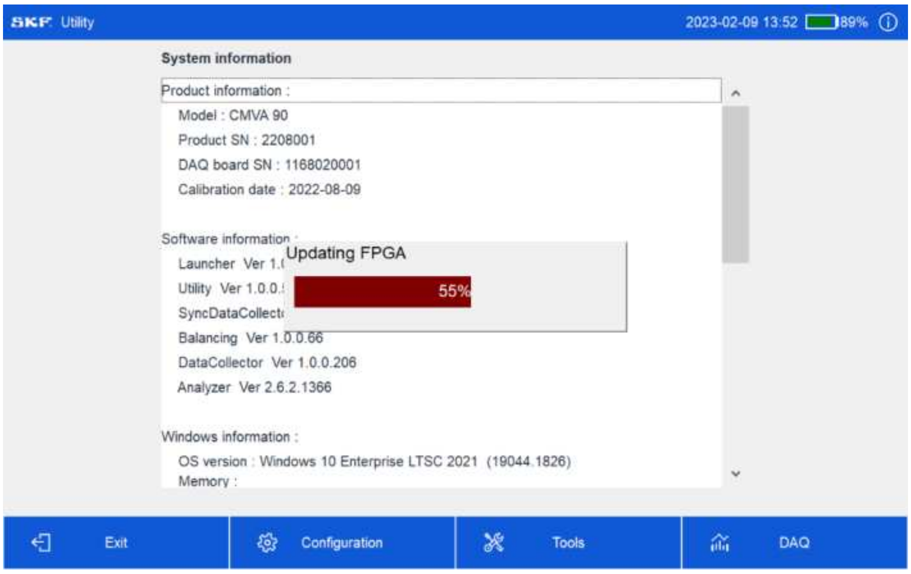
2. In the pop-up window you can follow the update process.



- **Update FPGA (Field-programmable gate array)**

1. Tap **Update FPGA** and from the dropdown select a disk (external USB device) containing the update file.

2. In the pop-up window you can follow the update process.



Tap **Exit** to exit the Utility software.

6 dBX Analyzer module

6.1 Introduction

> *Before using the software, please read and understand the separate Microlog Analyzer dBX Quick Start Guide.*

The **Analyzer** module is used for performing real-time measurements with FFT and Order Tracking, save the raw data as recorder files and playback them as real-time analysis.
It also supports multiple analysis modes, allowing you to run different analysis modules simultaneously, such as FFT, Order Tracking and Recorder.

Optional modules: Separate licenses are available for **Raw data recorder** and **Order Tracking** for running advanced real-time measurements and raw data recording.

To start the Analyzer module, using the navigation keys, navigate to the SKF Analyzer icon and press **OK** or double tap the icon.

Figure 146 Home screen – dBX Analyzer icon

6.2 Analyzer instrument modes

Analyzer can work in selected instrument mode:

- **Real time analysis** – Measure signal and perform real-time analysis with FFT or Order Tracking features.

- **Raw Data Recording** – Save the raw signal to the hard drive with a simple spectrum and waveform display.

- **Playback Analysis** – Analyzer supports raw data playback of a saved file. Once your raw data file is saved, Analyzer can **replay** the data file as a live measurement. An event can be recreated as if you were on-site and perform a playback analysis of the data file.

- **Real-time Analysis with raw data recording** – Combines Real-time Analysis and Raw Data Recording modes. While you perform Real-Time analysis, Analyzer allows you to throughput the raw data into the hard disk for a backup of important data.

- **Data review** – With the Data review mode you can:

    - Add data files for data viewing, analysis, and playback.
    - Work with multiple channels on a single graph, or separate if needed.
    - Perform post-analysis by applying mathematical processing. FFT, IFT, integration, differentiation, weighting functions, and filters.
    - Work with multiple cursor types: Single, Dual, Sideband, Harmonic, Move peak and multiple single cursors.
    - Export data files to XML, UFF or Wav file formats.

6.3 Analysis modules

6.3.1 FFT module

- **Functions for FFT analysis**: Time waveform, Auto spectrum, Complex spectrum, Cross spectrum, FRF, Coherence, Power cepstrum, Power spectral density, Envelope spectrum, Envelope waveform, Overall level, Mean trend, Rate trend and Rotation speed.

- **Trigger for FFT analysis** – Trigger source, trigger slope, trigger level, trigger delay and peak trigger.

- **Band & resolution for FFT analysis** – Select from 50 Hz to 40 kHz frequency band and 50 to 51 200 lines of resolution with real-time zoom FFT.

- **Average for FFT analysis** – Spectral or time domain average; linear, exponential average or peak hold; overlap percentage: 0%, 25%, 50%, 75% or max.; average number: from 1 to 5 000 times selectable; overload rejection and a preview of a waveform.

- **Signal map for FFT analysis** – Map functions: Time waveform, Auto power spectrum, Complex spectrum, and Cross spectrum. Measurement controls: Free run, armed by time step or armed by RPM step.
When the signal map is enabled, you can display it as a waterfall or intensity plot from the **Live data** panel.

6.3.2 Order tracking module

- **Functions for Order Tracking analysis** – Order trace, Order spectrum, Spectral map, Orbit, Filtered orbit, Orbit and waveform, Shaft centerline, DC gap and RPM profile.

- **Order resolution** – 1/2 order, 1/4 order, 1/8 order or 1/16 order.
- **Max order** – 5, 10, 20, 50, 100, 200, 400, or 800.
- **Rotation speed** – 30 to 240 000 rpm.
- **Types of average** – Off, linear, exponential, and peak hold.
- **Overlap level** – 0%, 25%, 50%, 75% and max.
- **Trigger method for tachometer channel** – Time waveform or spectrum.
- **Measurement control** – Time step, RPM step (run up, coast down or up and down) or both.

6.3.3 Raw data recorder module

- **Display** – Display Block waveform, raw waveform, and power spectrum during data recording in Raw data recorder mode. Display real-time analysis functions and raw waveform in Real-Time + Recorder mode.

- **Recording modes** – Record raw data in **Raw data recorder** mode or perform real-time measurement and record raw data at the same time in **Real Time + Recorder** mode.

- **Playback analysis** – Replay the raw data and conduct FFT or Order Tracking analysis.

6.4 Navigation and functionality

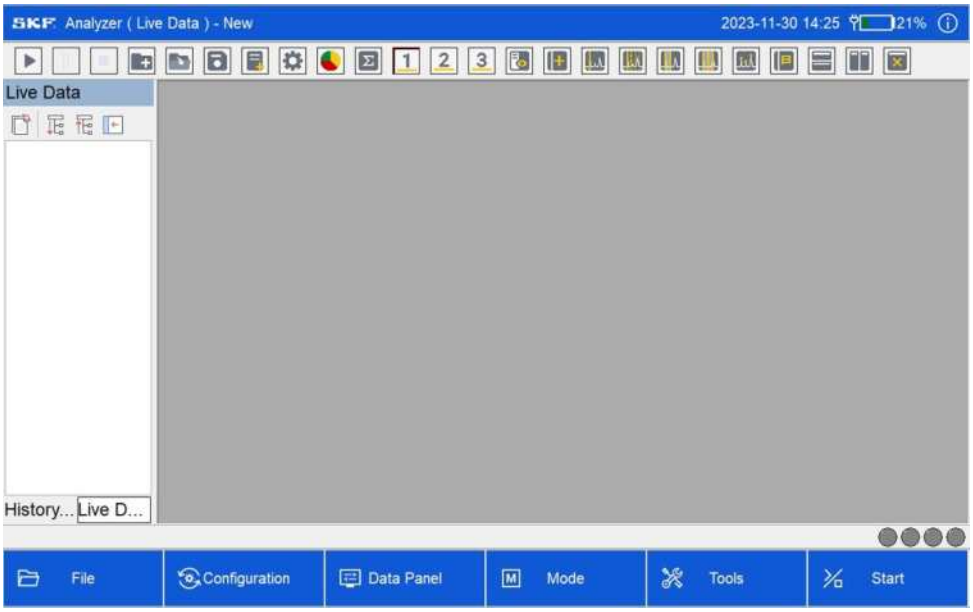


Figure 147 Analyzer Home screen with first use of the module

You can customize the display to show various toolbars, projects, files, charts, and status.

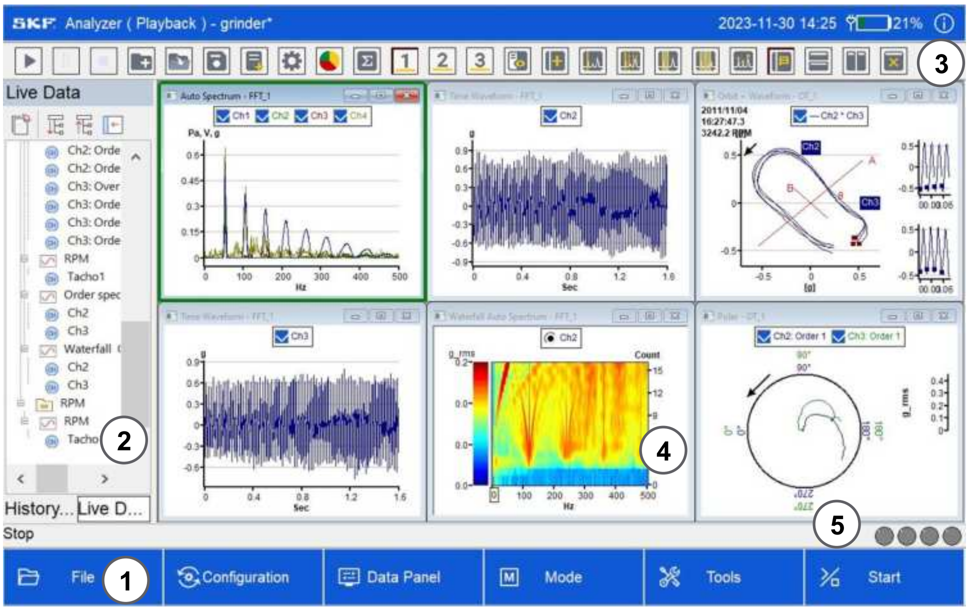


Figure 148 Analyzer Home screen – general layout

1. Menu buttons
2. Data panel
3. Toolbar
4. Plot area
5. Status bar

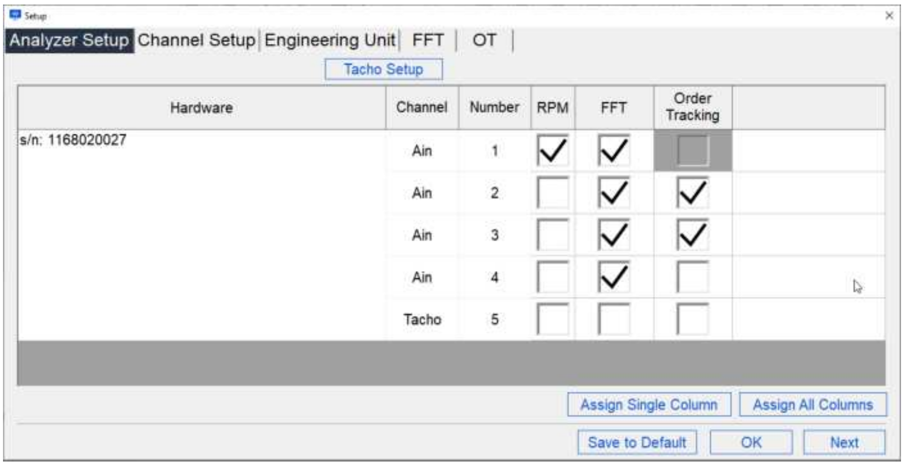
- **Menu buttons** – Tap on each menu or use the menu key to open the menu options. Each menu function is described in detail in this manual.

- **Toolbars** are a convenient way to create a shortcut for your file management, measurement, or display setup.

6.5 How to perform a measurement

1. Select a mode from the **Mode** menu. For example, if you are conducting a real-time analysis, select **Real Time Analysis**.

2. From the **Configuration** menu, select **Analyzer Setup** set up **Analyzer**, **Channel**, **Engineering Unit**, **FFT**, or **Order Tracking** measurement parameters.



3. To arrange the display layout, select a function from the **Live Data** panel and tap **OK** to create a data plot in the plot area.

4. To start / stop a measurement, press the **Start** / **Stop** key or the **Start** / **Stop** buttons on the screen.

5. After the measurement is completed, select **File** > **Save project** to save the measurement data and the setup parameters to a project file.

6.6 Perform test from a pre-set template

From the home screen, you can select a pre-set template to perform a specific test with the dBX Analyzer. These pre-set templates are built with the typical parameters required to perform specific types of testing. You can also change the settings of a template for future applications.

Figure 149 Microlog Analyzer Home screen

6.6.1 Bump test

Double tap the **Bump test** icon on the home screen to activate dBX Analyzer and load the pre-set parameters bump for a single channel bump test. With this template, you can perform a Bump test to investigate the resonance of a structure or collect necessary test data for a complete modal analysis with a 3rd party software.

Connect an ICP accelerometer to channel 1 and press the **Start** / **Stop** key to start the measurement. Use a hammer to impact the structure to initiate the measurement and display the result.

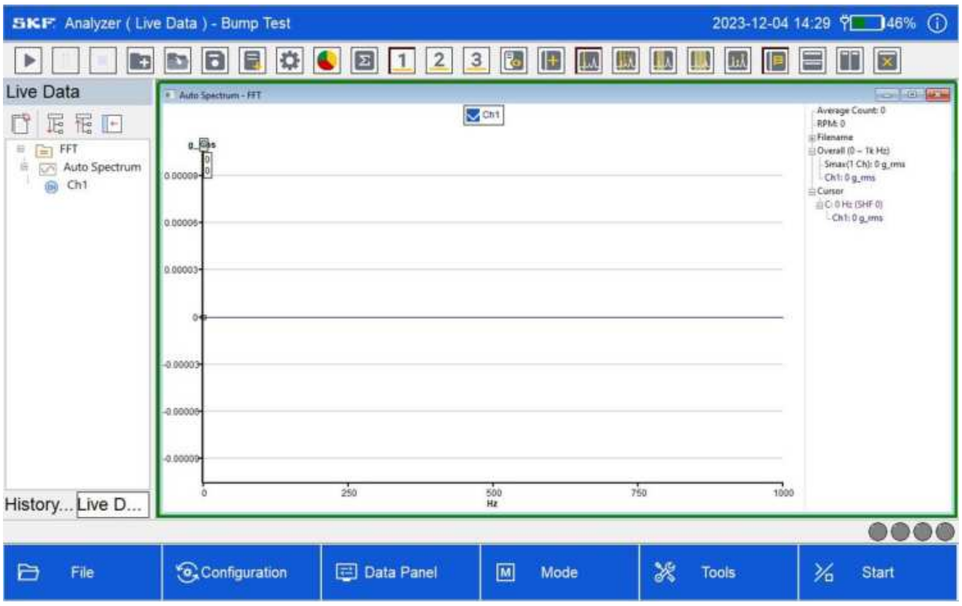


Figure 150 Bump test view – example

The pre-set parameters for this template are shown in the table below. To change the values of the parameters, enter the **Setup** menu.

Table 2 Pre-set parameters

	Ch 1	Ch 2	Ch 3	Ch 4
Coupling	ICP	Off	Off	Off
Input range	5 V	-	-	-
Window	Uniform	-	-	-
Node	1	-	-	-
Direction	+X	-	-	-
Advance number	0	-	-	-
Sensor type	Acceleration	-	-	-
Sensor unit	g	-	-	-
Sensitivity	100 mV/g	-	-	-
Display unit	g	-	-	-
Filter	Off	-	-	-

- **Measured functions** – Auto Spectrum
- **Frequency** – 1000 Hz
- **Resolution** – 800 lines
- **Average** – Peak hold average, 1 time
- **Trigger** – Channel 1, 100 mV trigger level, -3% trigger delay

6.6.2 gE Enveloping

Double tap the **gE Enveloping** icon on the home screen to activate dBX Analyzer and load the pre-set parameters for a gE Enveloping test. With this template, you can measure the time waveforms and envelope spectrum to investigate the condition of bearings. The envelope filter is set as ENV3: 500~10 kHz.

Select **Setup** to change it when necessary. Connect a uniaxial ICP accelerometer to channel 1 and press the **Start** / **Stop** key to start the measurement.

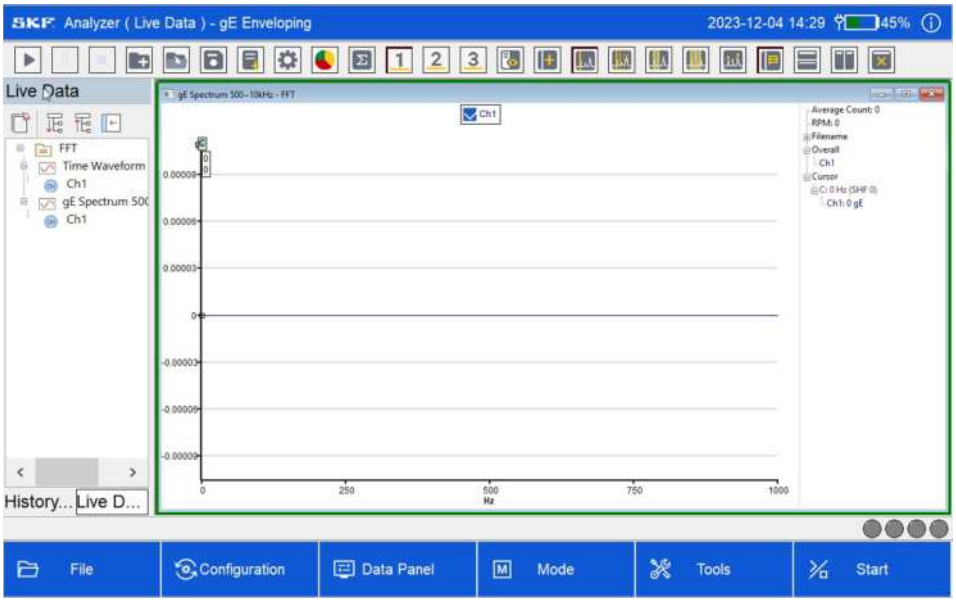


Figure 151 gE Envelope view

The pre-set parameters for this template are shown in the table below. To change the values of the parameters, enter the **Setup** menu.

Table 3 Pre-set parameters

	Ch 1	Ch 2	Ch 3	Ch 4
Coupling	ICP	-	-	-
Input range	5 V	-	-	-
Window	Hanning	-	-	-
Sensor type	Acceleration	-	-	-
Sensor unit	g	-	-	-
Filter	Off	-	-	-

- **Measured functions** – Time waveform and Envelope Spectrum (ENV3 500–10 kHz)
- **Frequency** – 1000 Hz
- **Resolution** – 1600 lines

- **Average** – Spectral average, 1 time
- **Delay** – 1 second
- **Trigger** – Off

6.6.3 FRF & ODS Analysis

Double tap the **FRF & ODS analysis** (Frequency Response Function and Operational Deflection Shape Analysis) icon on the home screen to activate dBX Analyzer and load the pre-set parameters for an FRF and ODS test. Connect a triaxial ICP accelerometer to channels 1, 2 and 3 and connect the reference accelerometer to channel 4.

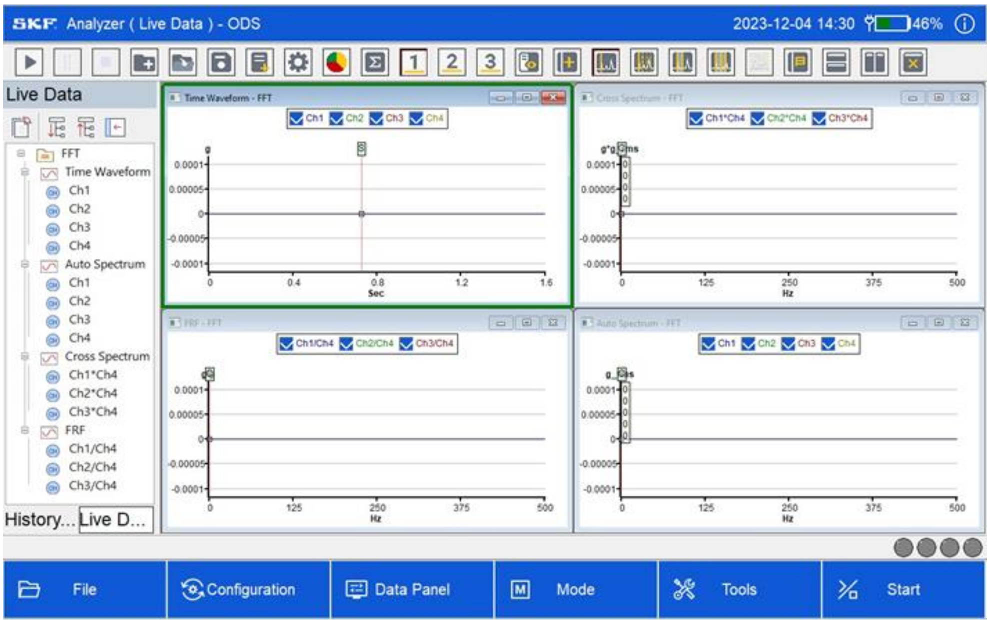


Figure 152 FRF & ODS Analysis view

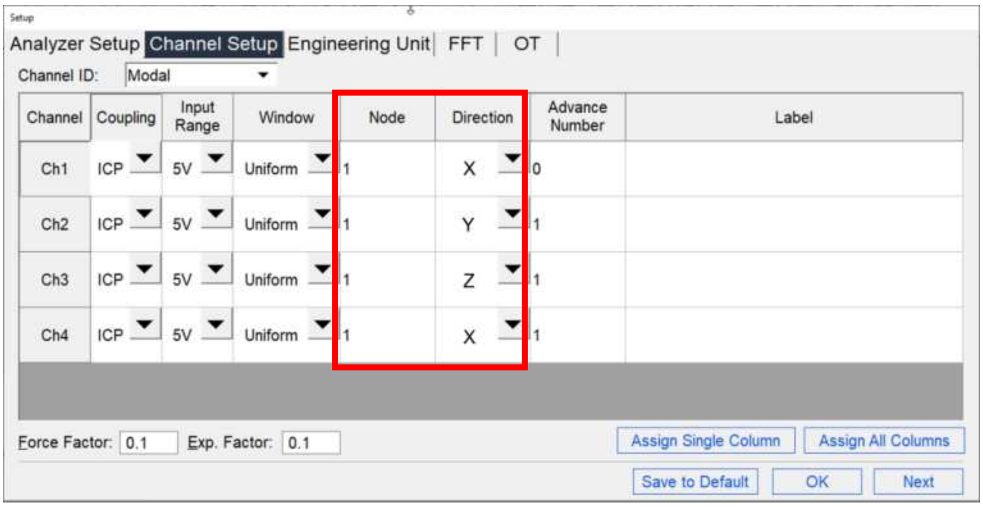


Figure 153 Channel Setup view: Initial node numbers and directions

In the default setting, the channel IDs are set as **+1X**, **+1Y** and **+1Z** and **1X**. You can edit it when you need to. The pre-set parameters for this template are shown in the table below. To change the values of the parameters, enter the **Setup** menu.

Table 4 Pre-set parameters

	Ch 1	Ch 2	Ch 3	Ch 4
Coupling	ICP	ICP	ICP	ICP
Input range	5 V	5 V	5 V	5 V
Window	Hanning	Hanning	Hanning	Hanning
Node	1	1	1	1
Direction	+X	+Y	+Z	+X
Advance number	0	1	1	1
Sensor type	Acceleration	Acceleration	Acceleration	Acceleration
Sensor unit	g	g	g	G
Sensitivity	100 mV/g	100 mV/g	100 mV/g	100 mV/g
Display unit	g	g	g	g
Filter	Off	Off	Off	Off

- **Measured functions** – Time Waveform, Auto Spectrum, Cross spectrum, FRF
- **Frequency** – 500 Hz
- **Resolution** – 800 lines
- **Average** – Spectral average, 10 times, overlap 50%
- **Trigger** – Off

When the sensors are installed to the correct positions and directions, press the **Start**/**Stop** button to take the first measurement. After the measurement is completed, the software will prompt you to save the data with a file name which consists of the point IDs and the date. In this example, the prompt file name is set as *+1X+1Y+1Z+1X_20231031*. Press **OK** to save the data file or select to save it with another file name or to another folder when you need to.

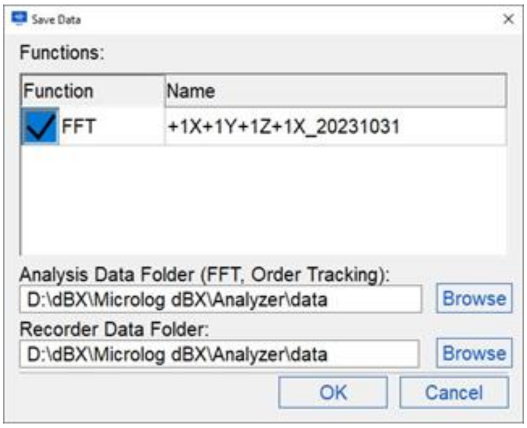


Figure 154 Save data – example

Now, you can move the triaxial accelerometer to the second node. Because the advance number of channels 1, 2 and 3 are set to "**1**", their node number will advance to "**2**" after the data saving.
It is recommended to enter the Channel Setup page to check the node number and the sensor directions before each measurement.

When necessary, change the node number or direction to match the location and orientation of the triaxial accelerometer.

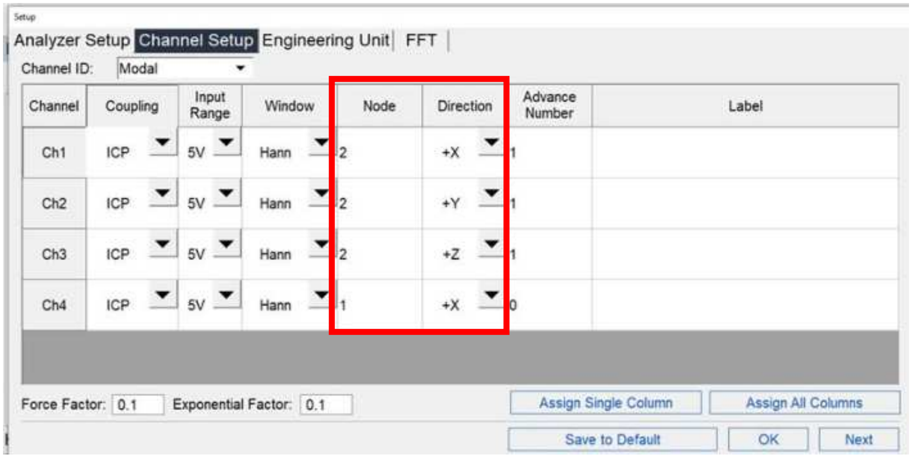


Figure 155 Channel Setup view – Initial node numbers and directions

Press the **Start**/**Stop** button to take the second measurement. Then the software will prompt you to save the data with a new file name *+2X+2Y+2Z+1X_20231031*

After the measurements of all Points are completed, exit the Analyzer module, and use the Sync Data Collector module to transfer the data files to your computer for post-analysis. Note that when a data file is saved in Analyzer format, another file in UFF (Universal File Format) is saved simultaneously. The UFF format is supported by most 3rd party software for Modal or ODS analysis, and you can use them for the ODS analysis.

Figure 156 UFF saved files – example

How to save the UFF files to a specified holder
By default, the UFF files are saved to the same folder where the test data files are saved. However, you can specify the destination folder for saving the UFF files. Select the **File** > **Save Option** from the main menu and select the **Auto Export** page. Tap **Open Folder** to select a destination folder.

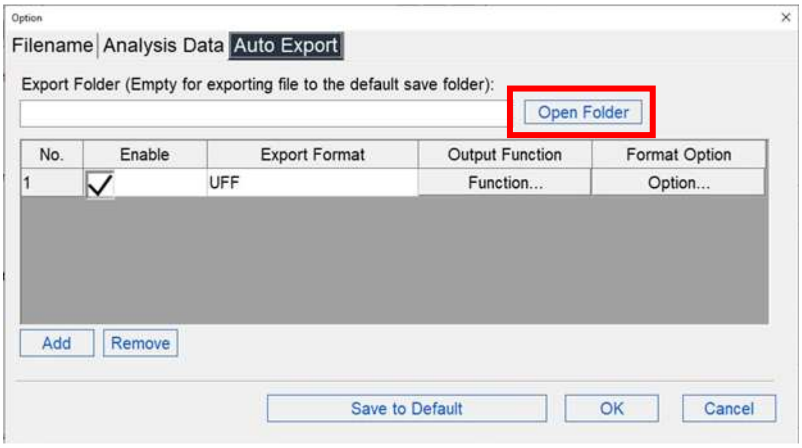


Figure 157 Auto Export

For example, after tapping the **Open Folder** button, you can select an external USB disk from the dropdown as a save destination for the UFF files and make it convenient to use them for ODS analysis.

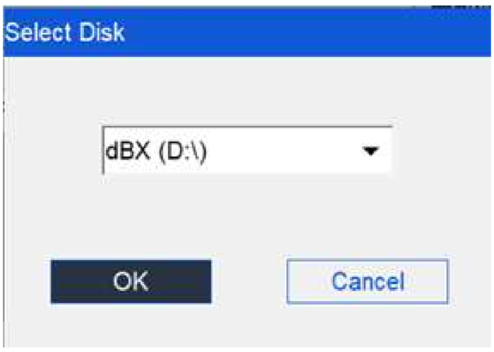


Figure 158 Select disk

6.6.4 Order tracking

Double tap the **Order Tracking** icon on the home screen to activate dBX Analyzer and load the pre-set parameters for an Order Tracking test. With this template you can measure order traces, order spectrums, spectral maps, and orbits within the speed range of 400~3600 rpm.

Connect an ICP accelerometer to channel 1 and connect the tachometer to channel **PWR/TRIG**. Press the **Start** / **Stop** key to start the measurement.

Note: The order tracking will not work without the signal from the tachometer.

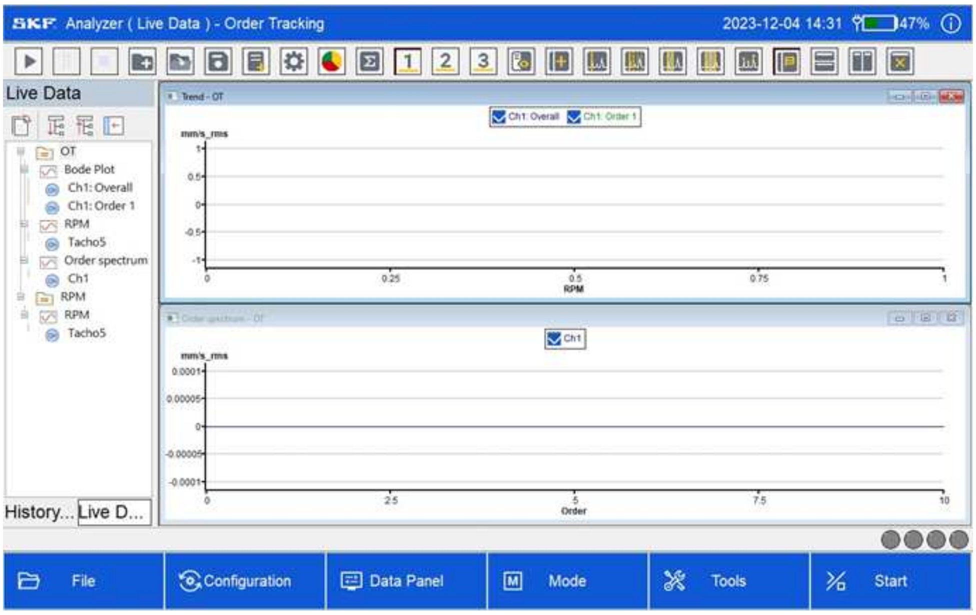


Figure 159 Order Tracking view

The pre-set parameters for this template are shown in the table below. To change the values of the parameters, enter the **Setup** menu.

Table 5 Pre-set parameters

	Ch 1	Ch 2	Ch 3	Ch 4
Coupling	ICP	-	-	-
Input range	5 V	-	-	-
Window	Hanning	-	-	-
Sensor type	Acceleration	-	-	-
Sensor unit	g	-	-	-
Sensitivity	100 mV/g	-	-	-
Display unit	mm/s	-	-	-
Filter	5 Hz HP	-	-	-

- **Measured functions** – Bode plot, Spectrum
- **Frequency** – 10 orders
- **Resolution** – 1/8 order
- **Speed range** – 400 to 3600 rpm
- **RPM step** – Event: Up and Down, Delta RPM: 10, Delta time: 5 seconds
- **Average** – Linear
- **Overlap Level** – Max%

6.6.5 Orbit analysis

Double tap the **Orbit analysis** icon on the home screen to activate dBX Analyzer and load the pre-set parameters for Orbit measurements. With this template, you can measure an orbit plot with channels 1 and 2 as the orbit pair.

Connect Eddy Current sensors to channels 1 and 2 and connect the tachometer to channel **PWR/TRIG**. Press the Start / Stop key to start the measurement.
Note that the Orbit measurement will not work without the signal from the tachometer.

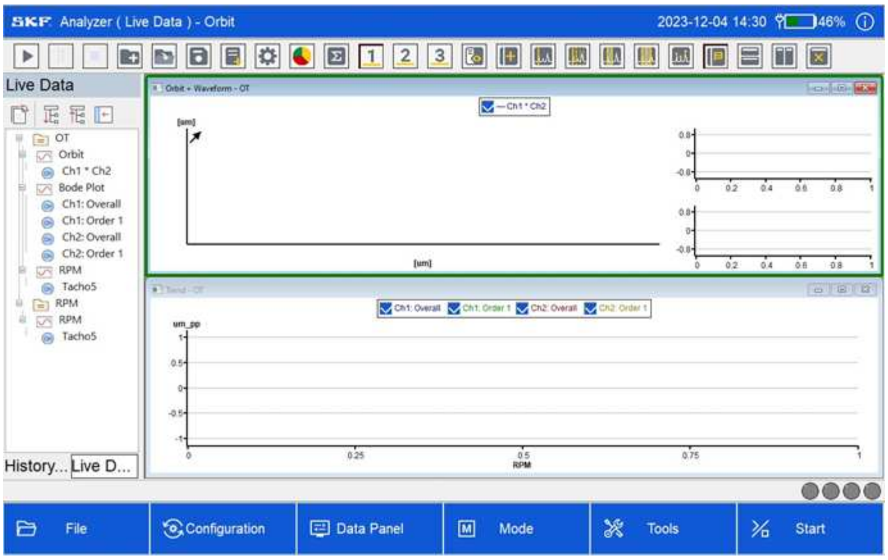


Figure 160 Orbit Analysis view

The pre-set parameters for this template are shown in the table below. To change the values of the parameters, enter the **Setup** menu.

	Ch 1	Ch 2	Ch 3	Ch 4
Coupling	DC	DC	-	-
Input range	20 V	20 V	-	-
Window	Hanning	Hanning	-	-
Sensor type	Displacement	Displacement	-	-

	Ch 1	Ch 2	Ch 3	Ch 4
Sensor unit	um	um	-	-
Sensitivity	7.874 mV/um	7.874 mV/um	-	-
Display unit	um	um	-	-
Filter	Off	Off	-	-

- **Measured functions** – Bode plot, Orbit
- **Frequency** – 10 orders
- **Resolution** – 1/8 order
- **Speed range** – 400 to 3600 RPM
- **Event** – Up and Down
- **RPM** – 10 RPM
- **Time step** – 5 seconds
- **Measurement Control** – Both RPM and Time step

6.6.6 Run up coast down

Double tap the **Run up coast down analysis** icon on the home screen to activate dBX Analyzer and load the pre-set parameters for run up coast down measurements. With this template, you can measure vibration signals during a run-up or coast-down process and derive order traces, order spectrum, and spectral maps within the speed range of 400 rpm to 3600 rpm.

Connect ICP accelerometer to channels 1, 2, 3 and 4 and connect the tachometer to channel **PWR/TRIG**. Press the **Start** / **Stop** key to start the measurement.

Note: The Run up coast down measurement will not work without the signal from the tachometer.

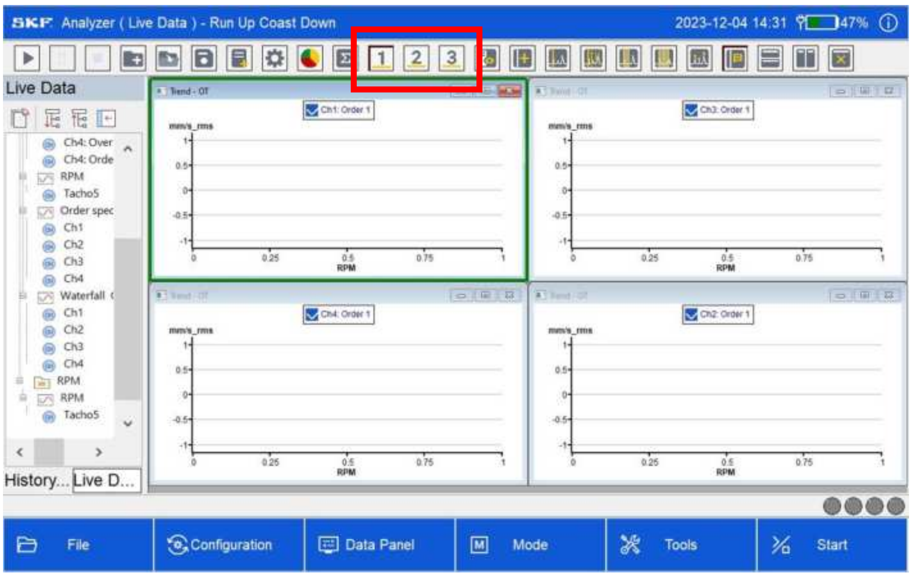


Figure 161 Run up coast down view

To display overall traces and waterfall plots, select the layout **2** and **3** icons from the **Toolbar**.

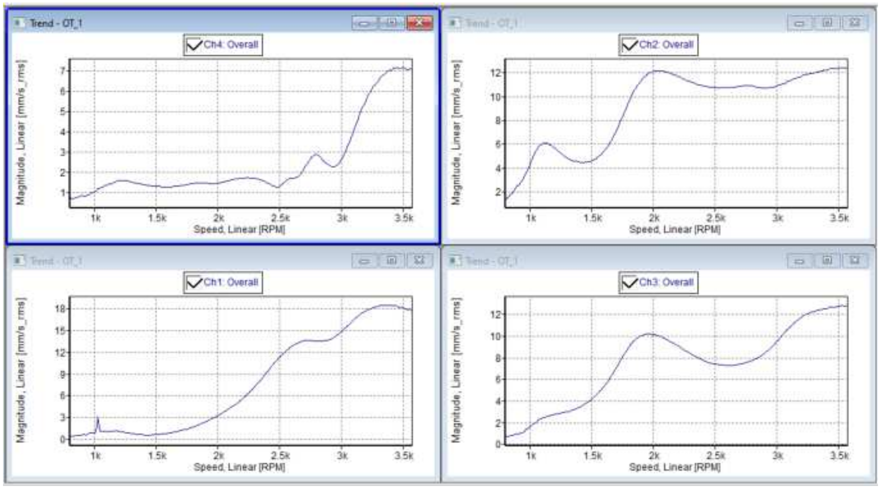


Figure 162 Layout 2: Overall traces plot – example data

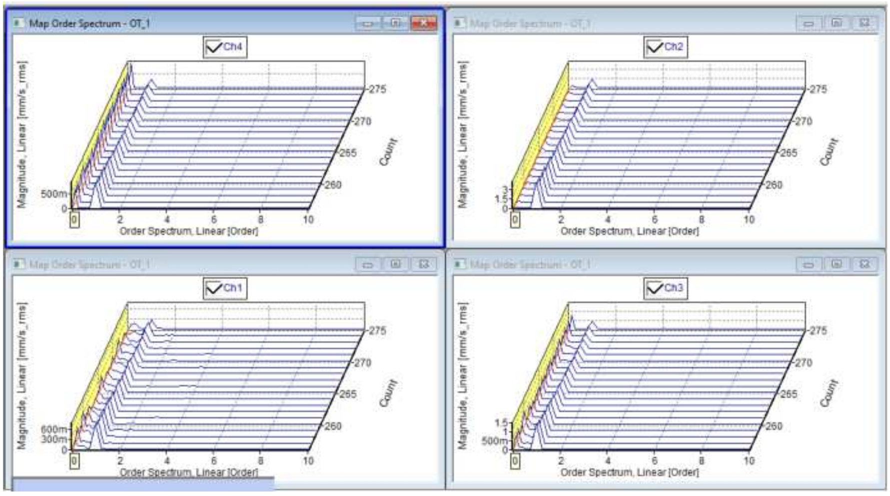


Figure 163 Layout 3: Waterfall plots – example data

6.6.7 Data recorder

Double tap the **Data recorder** icon on the home screen to activate dBX Analyzer and load the pre-set parameters for raw data recording. With this template, you can throughput the measured signals into the hard drive continuously and save the raw data into raw data files for post-analysis.

The measurement bandwidth of this template is set to 2 kHz. You can change the setting when necessary. Connect ICP sensors to channels 1, 2, 3 and 4 and connect the tachometer to channel **PWR**/**TRIG**. Press the **Start** / **Stop** key to start the measurement.

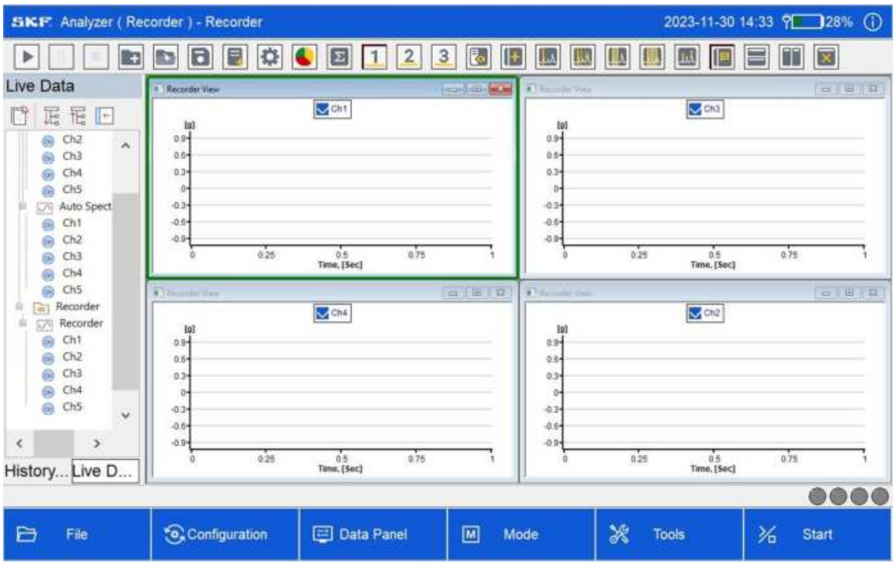


Figure 164 Data recorder view – example

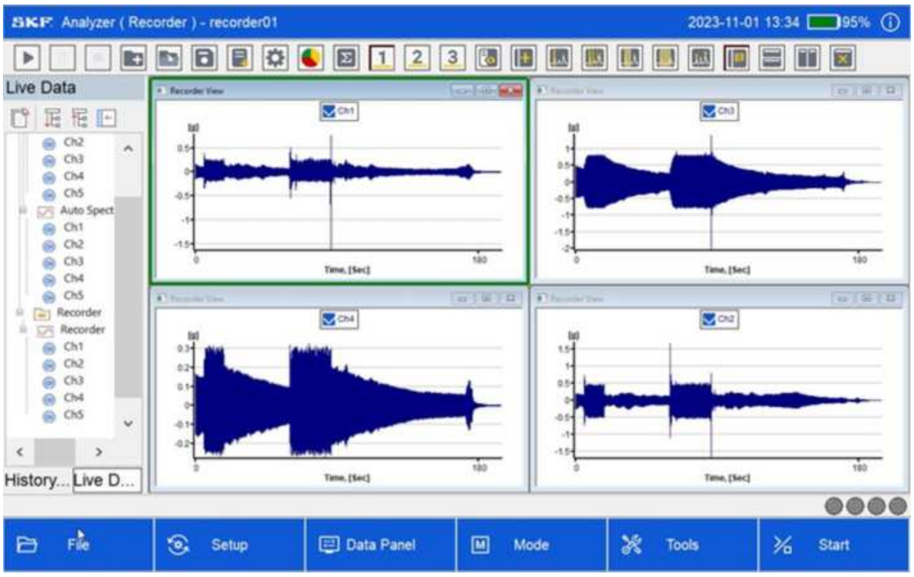


Figure 165 Data recorder measurement – example

Press the **Start** / **Stop** key to stop the recording and save the data to a project or data file.

6.7 Working with Plots

6.7.1 History and Live Data panels

The **History data** view contains the data you add to the project, while the **Live data** view contains the data from real-time measurement or playback analysis.

You can find History Data and Live Data labels on the left-hand side of the display. Tap on a label or select the **Data panel** menu to open it.

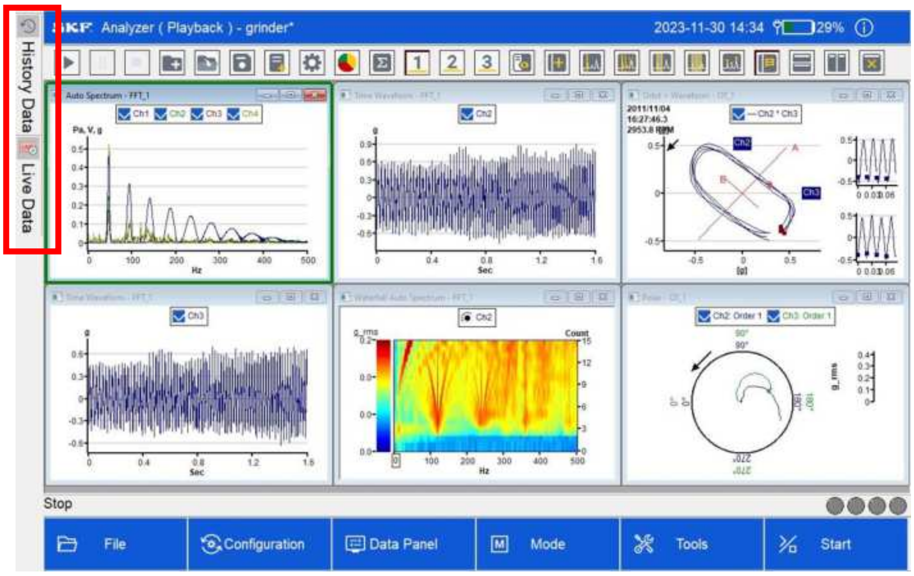
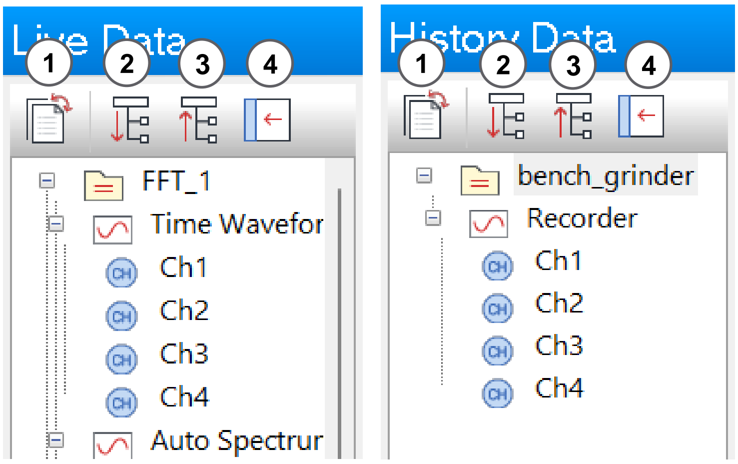


Figure 166 History and Live data

When the **History Da**ta or **Live Data** views are not in use, they can be hidden.



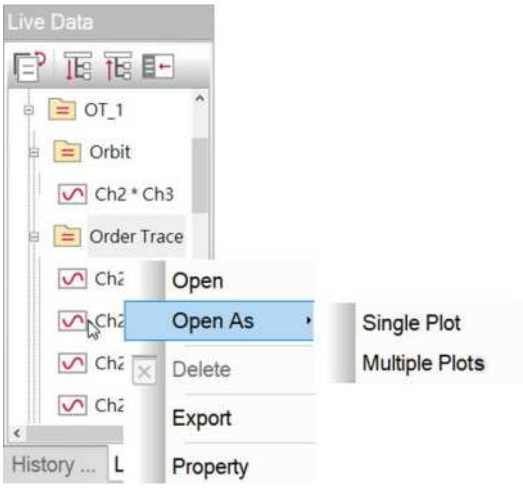
1. Arrange the hierarchic structure
2. Expand all nodes
3. Collapse all nodes
4. Hide this view

6.7.2 Create a plot

To create a chart, first select a function from the History Data or Live Data view to show it in the display area. Use one of the following methods to create plot(s):

1. Press the right-click key and select **Open As** > **Single Plot** or **Multiple Plots**.

Open as a single plot will create a plot which contains all the channels under this node. Open as multiple plots will create an independent plot for each channel.



2. Select a function or a channel and press the **OK** key to create a plot.

6.7.3 Working with single-plot and multiple plot view

You can display multiple plots in one display. Press the **C** key to enable active window selection.

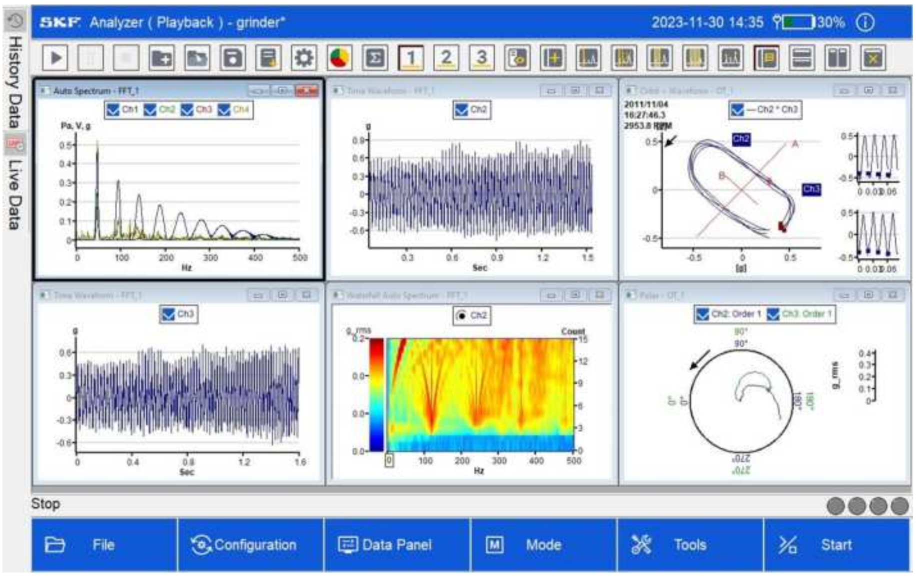


Figure 167 Multiple plot

It will highlight the selected plot with a black frame. Use the navigation keys to move the red frame to a selected plot and press **OK** to maximize it.

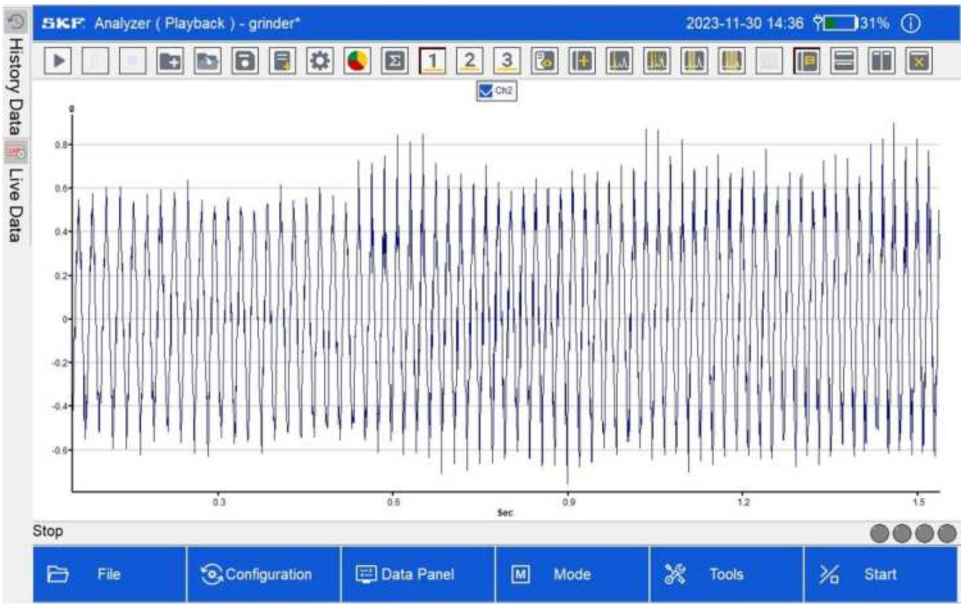


Figure 168 Single plot

Press **OK** again to restore it. The selected active window displays its title bar with blue colour, while the unselected window with grey. You can also tap on a plot to make it active.

6.7.4 Displaying the plot menu

Press the right-click key or tap for 2 seconds on the plot to display the plot menu on the active plot. With the plot menu, you can set up **X**, **Y**, and **Z** axes, select plot format, etc. Note that the content of the plot menu depends on the plot type.

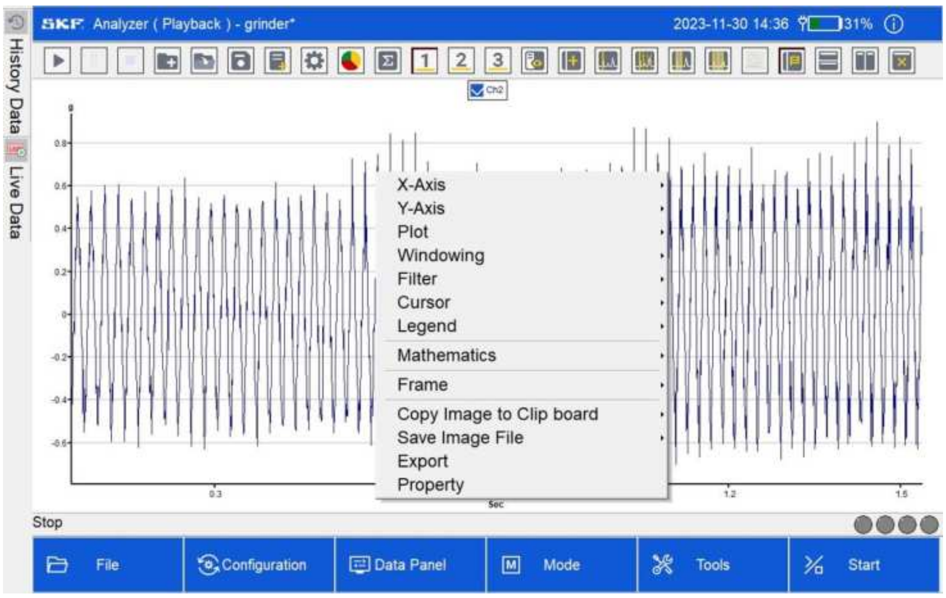
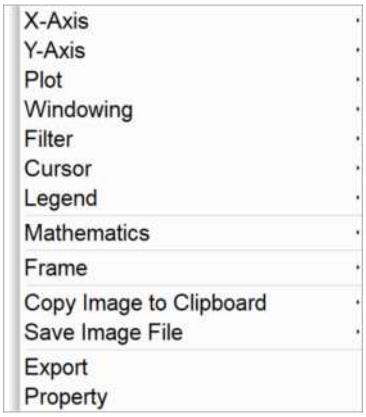
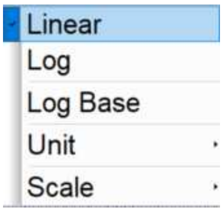


Figure 169 Plot menu

6.7.5 Plot menu of a time waveform plot



- X-axis



  - Select **Linear** or **Log** scale
  - **Log Base** – Specify the log base for log axis display
  - **Unit** – Select **Sec**, **Min**, or **Hour**
  - **Scale** – Select **auto** scale or **fixed** scale

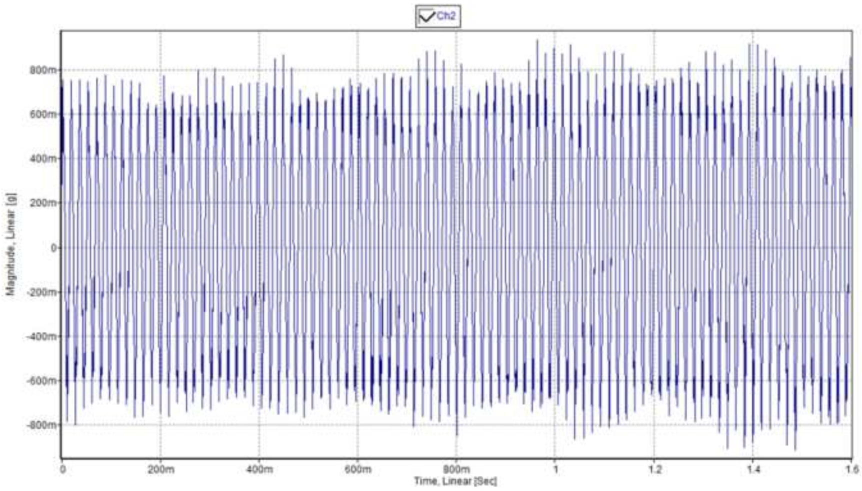
- **Y-axis**

  - **Scale** – Select auto scale, fixed scale, full scale, or a fraction of the full scale
  - **Overall Detection** – Select rms, peak or peak-to-peak as the detection of overall values (displayed in the side frame)

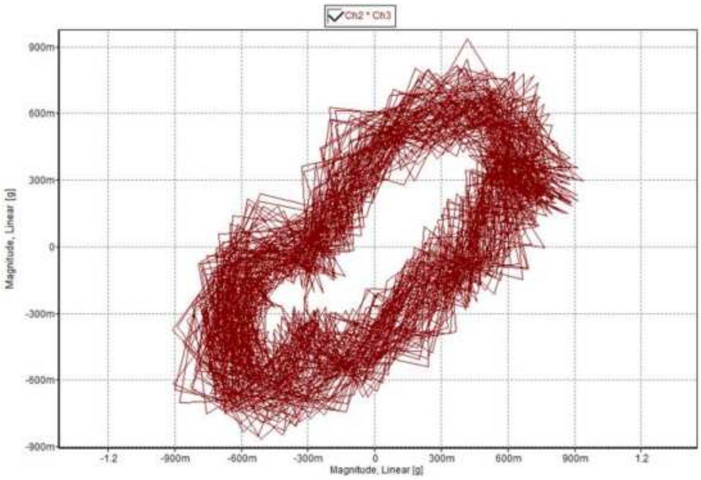
- **Plot**

Select the plot format as Linear, XY or Circular.

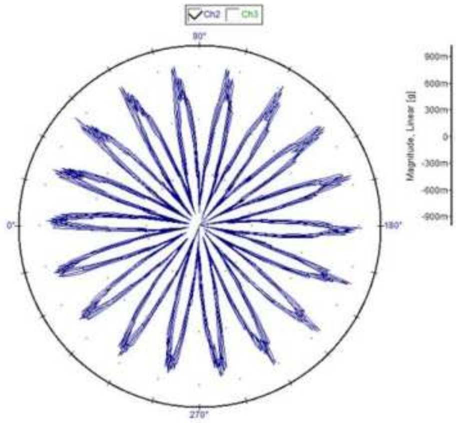
- **Linear plot**



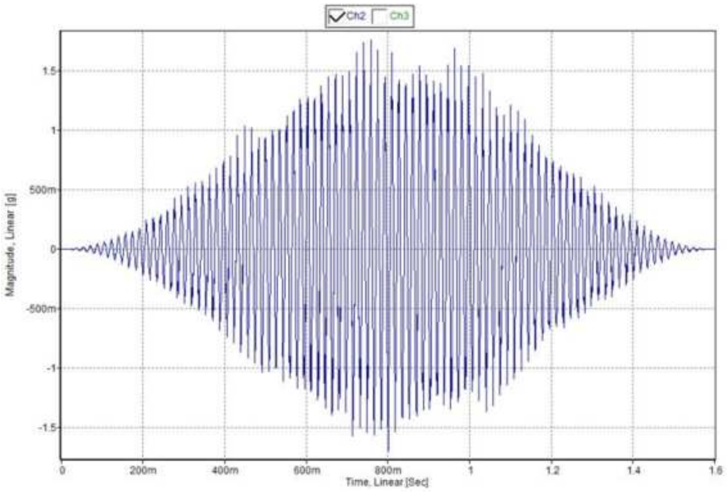
- **XY plot** – Available when two waveforms are shown in the same plot



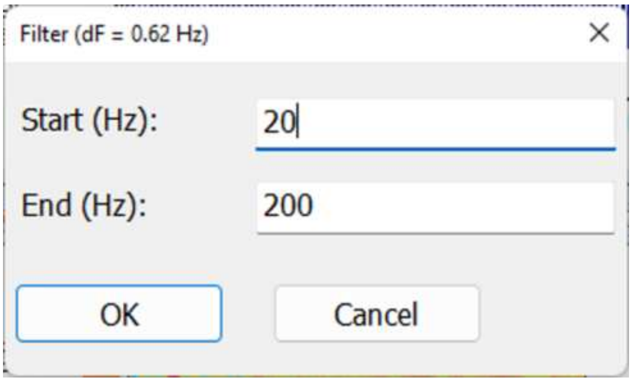
- **Circular plot** – Require you to input the number of gear teeth and the rotation speed to display the waveform in circular plot format



- **Windowing** – Select to display the waveform with or without the windowing function. The plot below shows an example of a time waveform with a Hanning window.



- **Filter** – Specify a band pass filter and apply it to the waveform. Select **Unfiltered** to undo it.

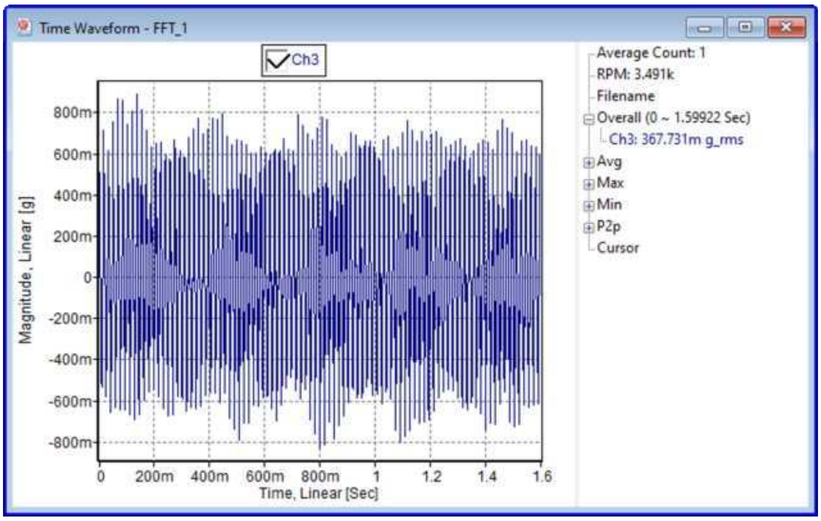


- **Cursor** – Press the cursors key to select a cursor type.

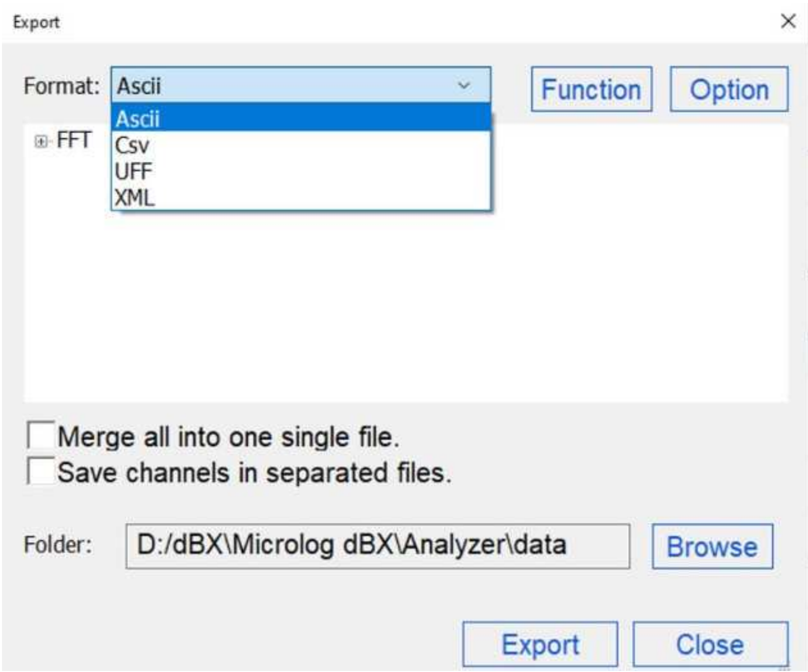
- **Legend** – Select to show or hide the channel legend.

- **Mathematics** – Tap the mathematics icon on the Tools bar to select mathematical operations to conduct post-analysis.

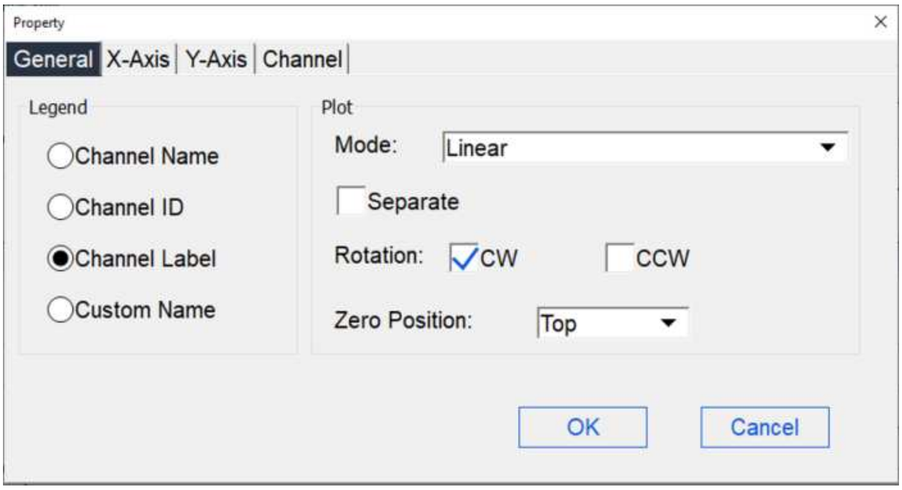
- **Frame** – Show or hide the side frame. The detailed information and cursor information is shown in the side frame. You can also tap the icon on the Tool bar to show or hide the side frame.
  Show or hide the Caption an



- **Copy image to clipboard** – Copy the plot to clipboard as an image.
- **Save image file** – Save the plot as an image file.
- **Export** – Export the data of the plot to a specified format. You can also export the data of the project from the **File** menu.

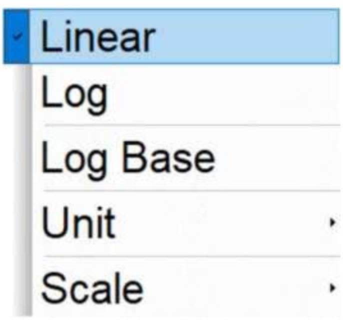


- **Property** – Edit plot properties.



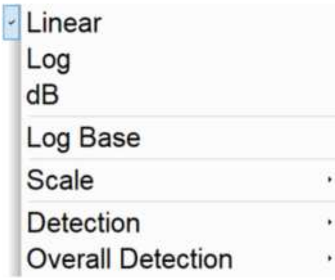
6.7.6 Plot menu of an auto spectrum plot

- **X-Axis**



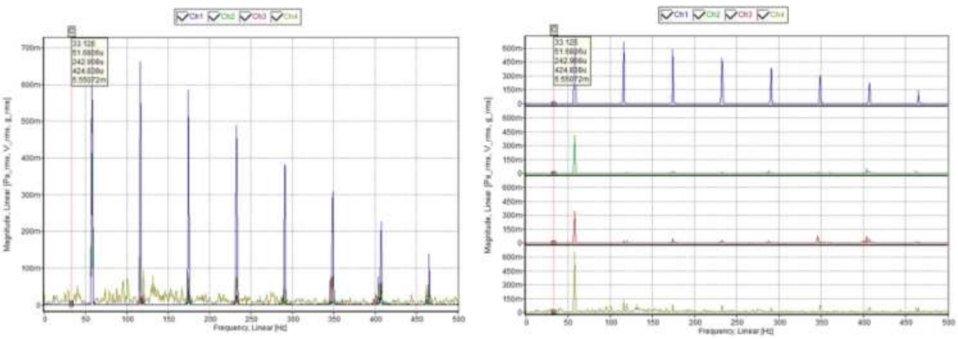
  - Select Linear or Log scale.
  - **Log Base** – Specify the log base for log axis display.
  - **Unit** – Select **Hz**, **CPM** or **Order.**
  - **Scale** – Select auto scale or fixed scale.

- **Y-Axis**



  - **Linear, log** or **dB** – Select the scale type of the Y axis.
  - **Log Base** – Specify the log base for log axis display.
  - **Scale** – Select **auto** scale, **fixed** scale, **full** scale, or a **fraction** of the full scale.
  - **Detection** – Select the detection type of the Y-axis.
  - **Overall Detection** – Select **rms**, **peak** or **peak-to-peak** as the detection of overall values displayed in the side frame.

- **Plot** – **Separate** or **overlap**. Used when there is more than one trace in the plot.



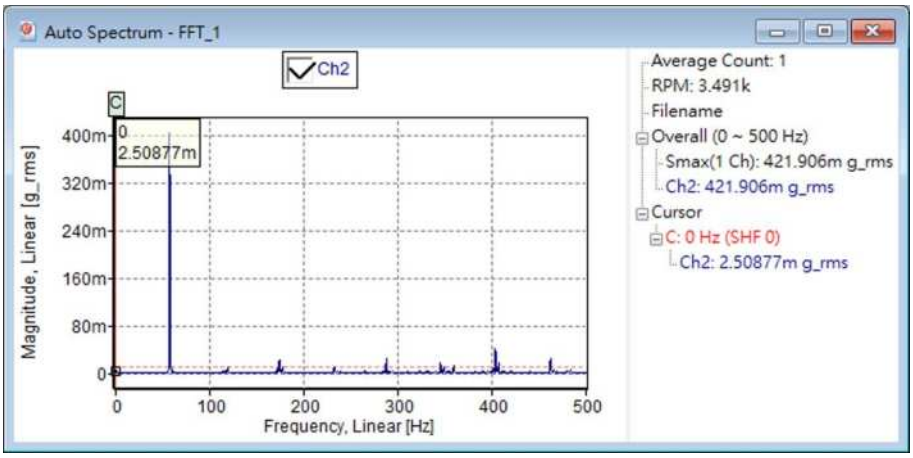
- **Cursor** – Press the cursor key to select a cursor type.

- **Legend** – Select to show or hide the channel legend.

- **Mathematics** – Tap the mathematics icon on the Tools bar to select mathematical operations to conduct post-analysis.

- **Frame** – Tap and select one of the options:

  - Show or hide the side frame. The detailed information and cursor information is shown in the side frame. You can also tap the icon on the Tool bar to show or hide the side frame.
  - Show or hide Caption.
  - Show or hide Border.

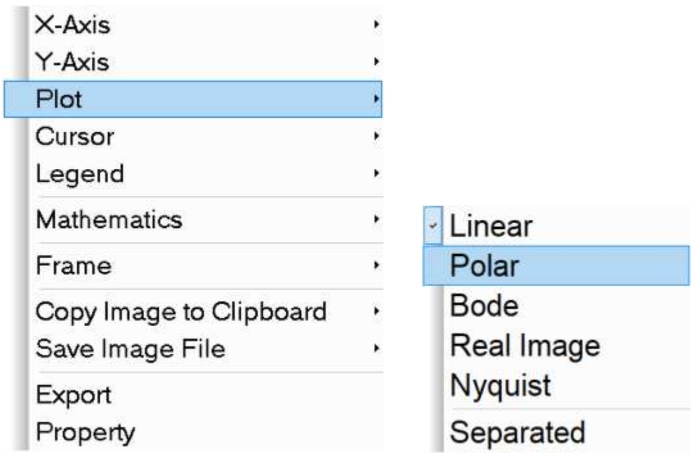


- **Copy image to clipboard** – Copy the plot to clipboard as an image.
- **Save image file** – Save the plot as an image file.
- **Export** – Export the data of the plot to a specified format. You can also export the data of the project from the **File** menu.
- **Property** – Edit the various properties of the plot.

6.7.7 Plot menu of a complex spectral plot

A complex function contains both amplitude and phase.
For example, the **FRF**, **Cross power spectrum**, and **Complex spectrum** are the typical complex functions which contain amplitude and phase data.

- **Plot** – Select plot format.



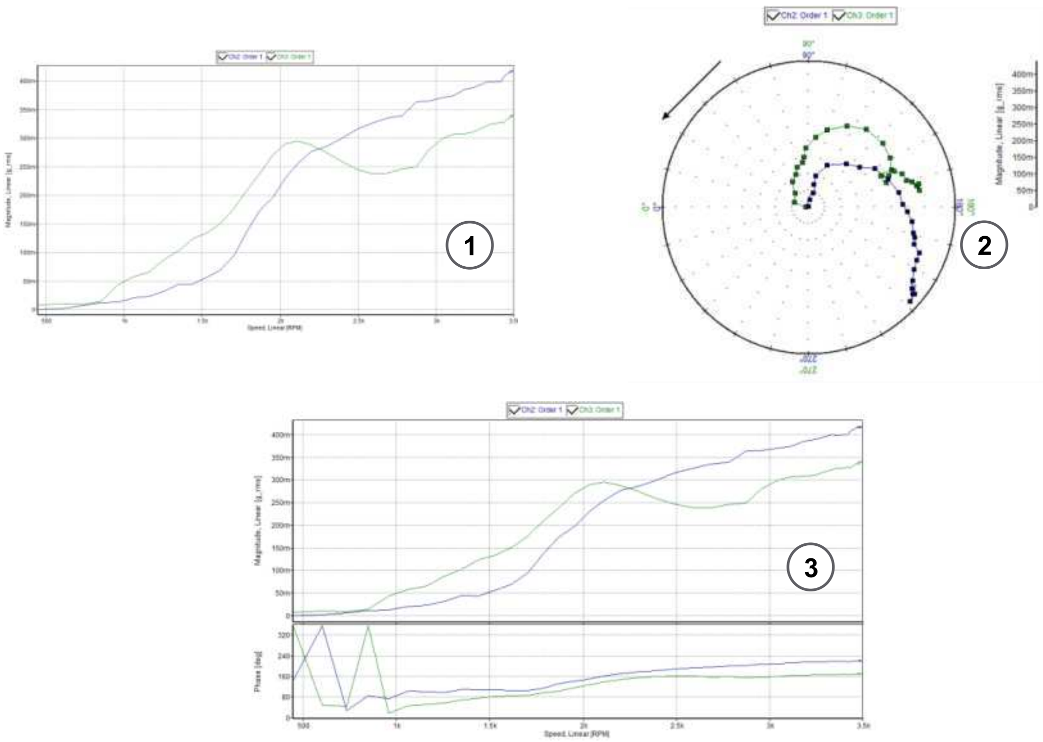


Figure 170 *Plot types – example*

1. Linear plot
2. Polar plot
3. Bode plot

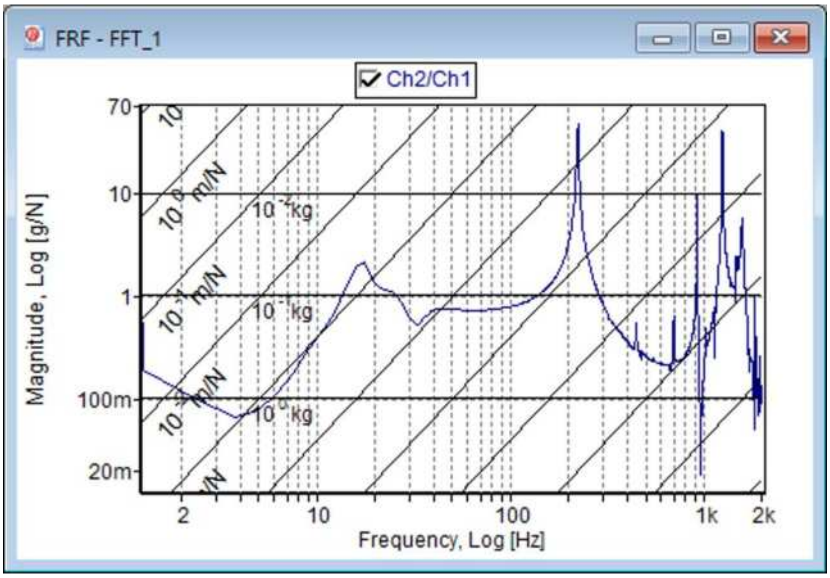
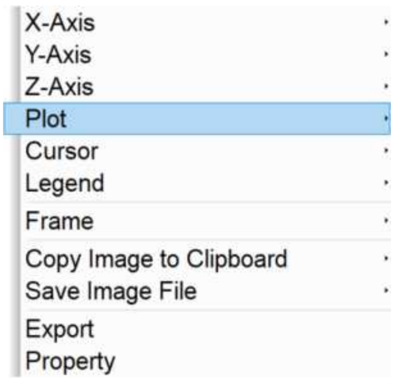


Figure 171 Accelerance / Mobility / Compliance plot of a FRF function

6.7.8 Plot menu of a 3D plot



- **Z-Axis** – Select Linear, Log, Count, Time, or RPM.
- **Plot** – Select plot format: **Waterfall**, **Intensity**, or **Overlap**.

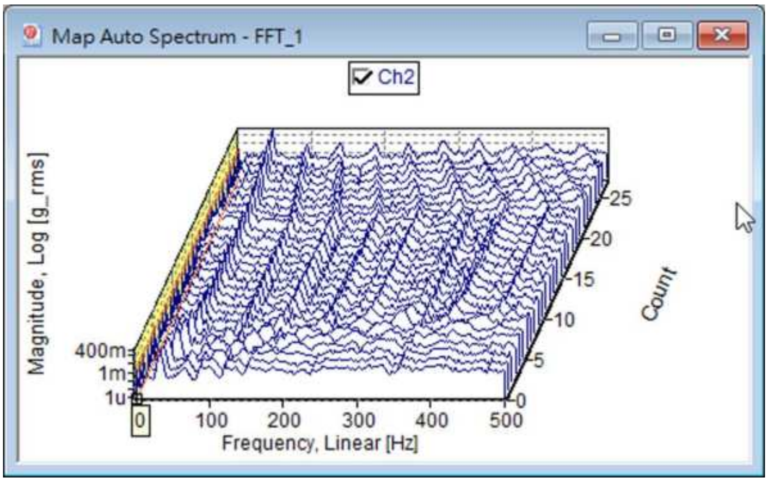


Figure 172 Waterfall plot

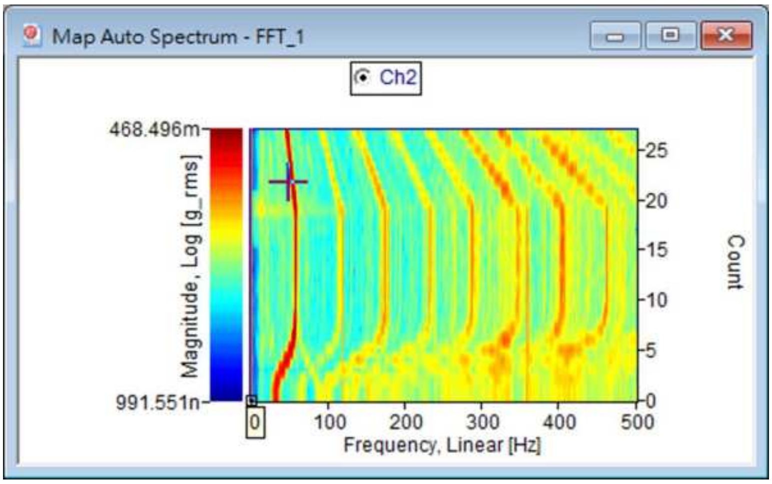


Figure 173 Intensity map

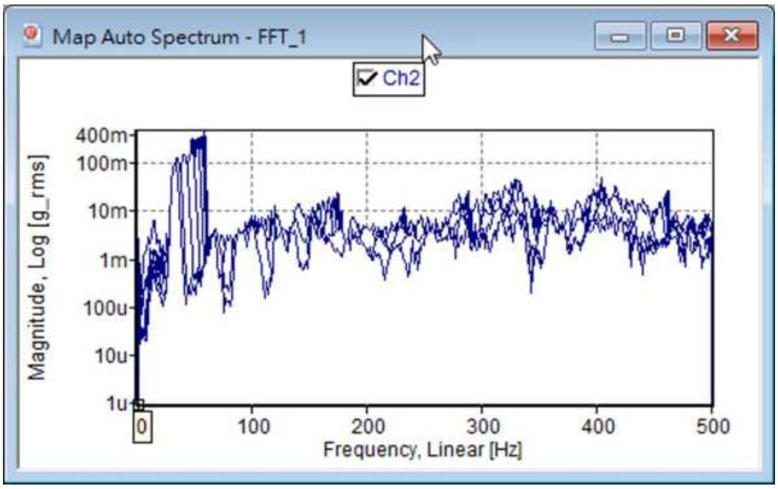
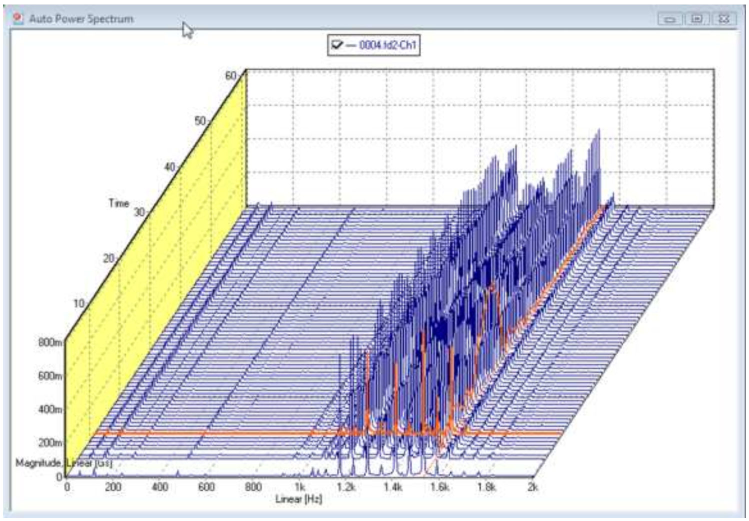
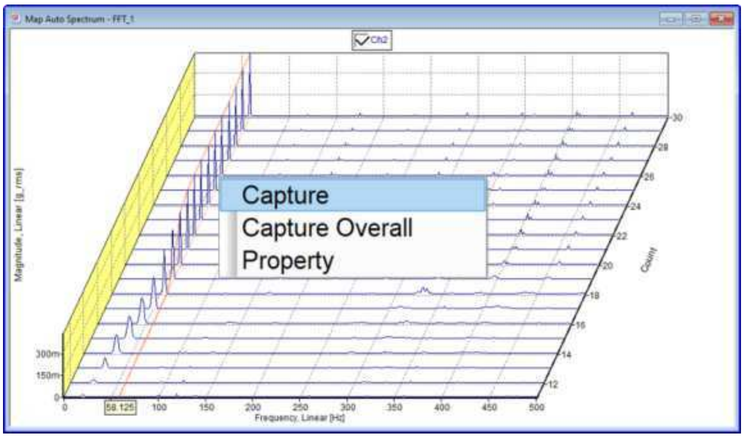


Figure 174 Overlap plot

- **Cursor** – Select the cursor type from **X**, **Y**, **Dual X**, **Dual Y** and **All X & Y** cursor shown on the graph with time selected in the Z-Axis.



  - **Cursor/Capture –** Select to Capture the trace where the cursor is on and display it from the Live data panel.



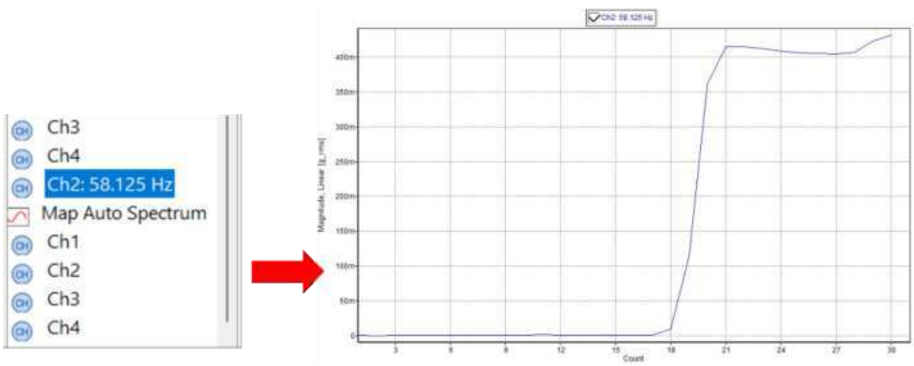
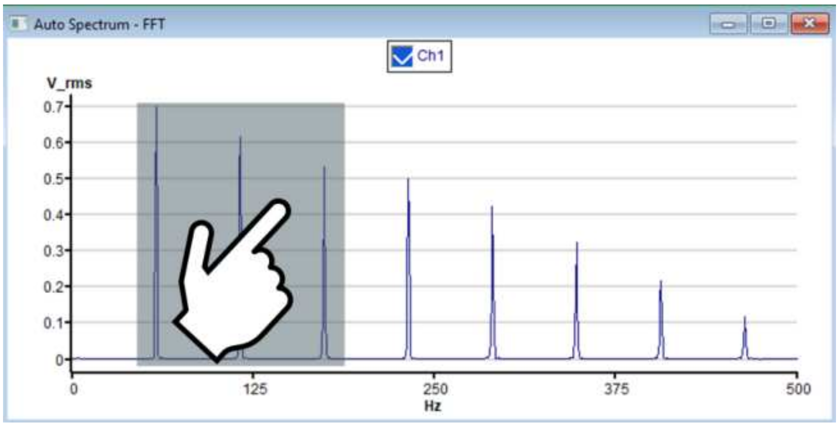


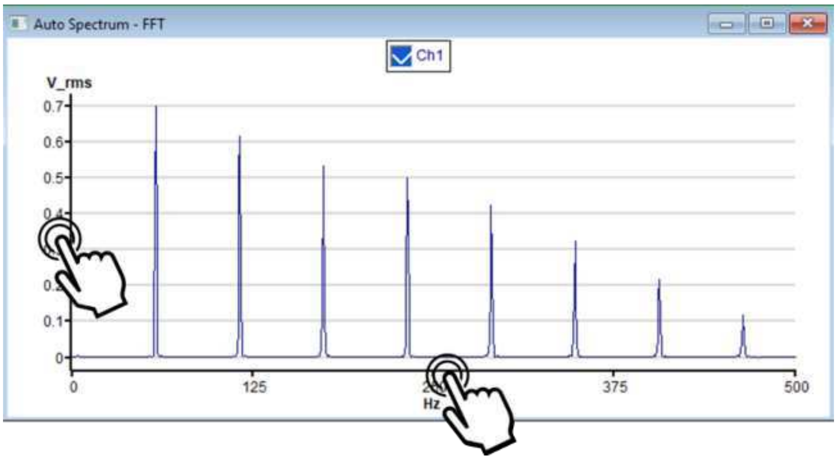
Figure 175 Cursor capture

6.7.9 Zoom in, Zoom out and Pan

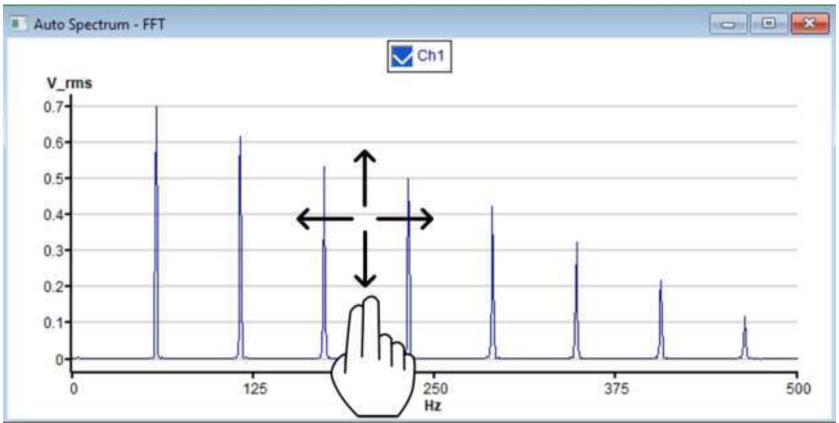
- **Zoom in and out** – Press the zoom-in and zoom-out keys to zoom in or zoom out a plot along the X-axis from the cursor position.
  Or use your index finger and thumb to zoom in.



- **Auto Scale** – To auto-scale an axis, double tap the axis legend area.



- **PAN** – Use two fingers to drag the plot.



6.7.10 Add an additional signal to the plot

To add additional signal to the plot, select a signal from the Live data panel or the History panel and drag and drop it into the plot.

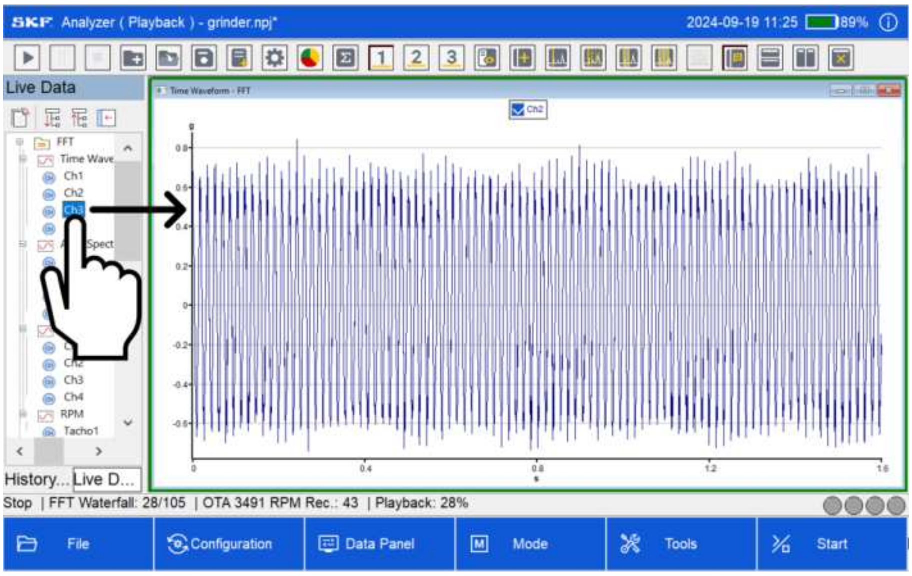
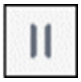


Figure 176 Add signal to the plot

6.7.11 Toolbar functions

The toolbar on the top of the screen provides a shortcut to the most commonly used functions of the dBX Analyzer. Use the touchscreen feature to work with the toolbar.

The toolbar functions are divided into several groups:

Measurement control	
	Start a measurement
	Pause a measurement
	Stop a measurement

File functions	
	New project, for more information refer to File menu

File functions

Open project, for more information refer to File menu

Save project, for more information refer to File menu

Add data, for more information refer to File menu

Setup

Open the Setup view

Plot format

Toggle different plot format for the active plot

Mathematical operation

Open the mathematical operations menu to perform post analysis on the active plot

Layout management

1 2 3 Select from the 3 available layouts to arrange the plots

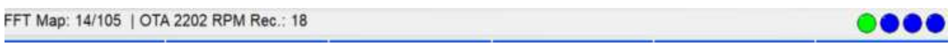
Side frame

Display or hide the side frame of the active plot

Cursor and cursor functions	
	When the Single cursor is active, tap this icon to add one more single cursor to the plot. You can add more single cursors to the plot when needed.
	Display or hide single cursor
	Display or hide harmonic cursor
	Display or hide dual cursor
	Display or hide sideband cursor.
	Enable or disable the Move peak cursor feature. When the Move peak is enabled, the cursor will stay at a peak.
	Display or hide the cursor marker, for more information refer to Show or hide cursor marker

File functions	
	Tile horizontal
	Tile vertical
	Close all

6.7.12 Status bar



The status bar shows the measurement information, which includes the average number, triggering status, RPM reading, etc.
The channel status with four colour icons is shown on the right side of the status bar.

- **Green** – normal
- **Blue** – ICP bias abnormal
- **Red** – overloaded

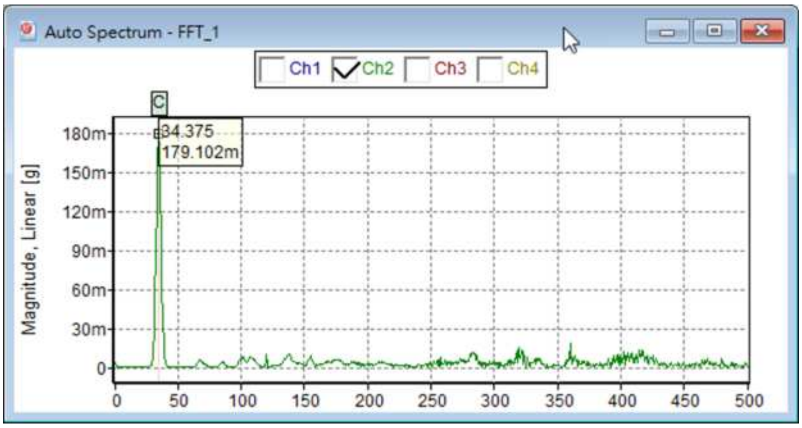
6.8 Working with cursors

Select a cursor using one of the following methods:

- Select a cursor from the Plot menu.
- Select a cursor from the Toolbar icons.
- Press the cursor taggling key repeatedly until the desired cursor is selected.

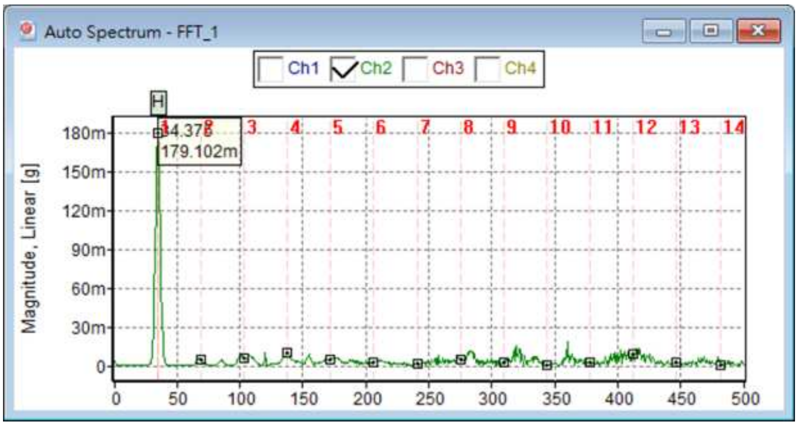
6.8.1 Cursor types

- **Single cursor** – The basic cursor type which shows only one cursor value at a time.

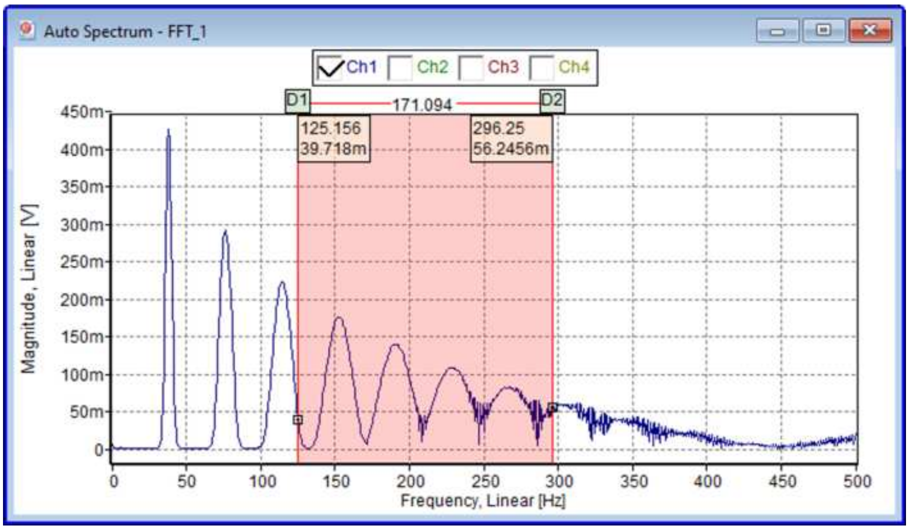


  -  **Add cursors** – Add a single cursor to the plot. It is possible to add multiple single cursors. To add single cursors to the plot, tap the **Add single** icon from the Toolbar.

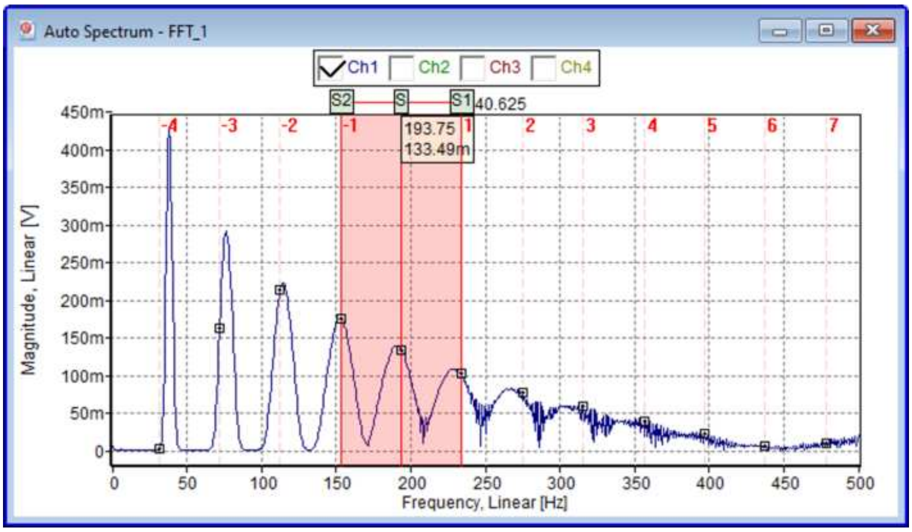
     **Harmonic Cursor** – Places a main cursor with its harmonic (orders) of the cursor position. This helps to understand the harmonic frequencies of the measurement.



-  **Dual cursor** – Places two cursors to shade an area D1 – D2.



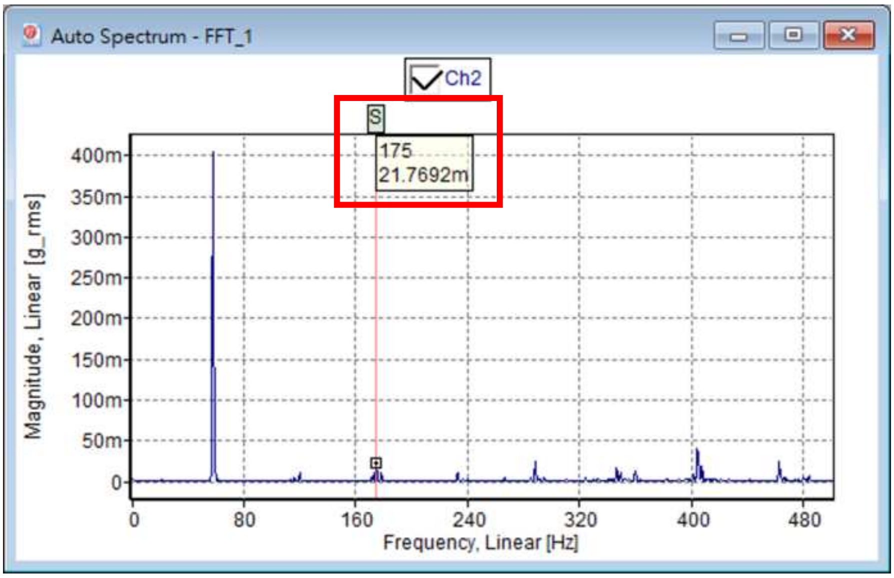
-  **Sideband Cursor** – Will display cursor labels **S**, **S1** and **S2** with harmonic frequencies.



-  **Jump peak** – When Peak Cursor is selected, and you move the highlighted cursor on the graph, it will always move from one peak to another.

6.8.2 Show or hide cursor marker

 This action will display cursor values on the graph, and it can be helpful to highlight a particular value on the graph.

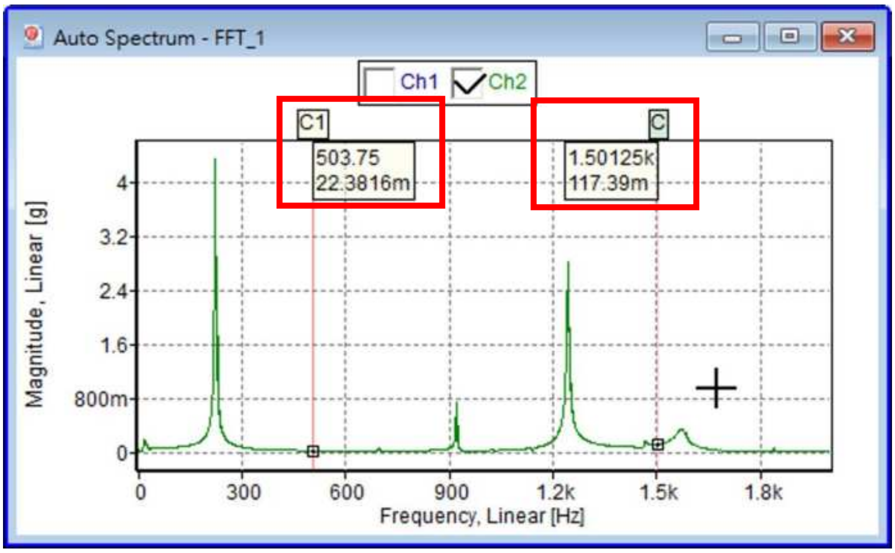


6.8.3 Move cursors on the plot

Select an active cursor

To move a cursor, tap the cursor to activate and highlight it.

Once activated, use the navigation keys, or drag and drop the cursor using the touchscreen to move it to a specific position on the plot.



6.8.4 Moving Dual Cursors

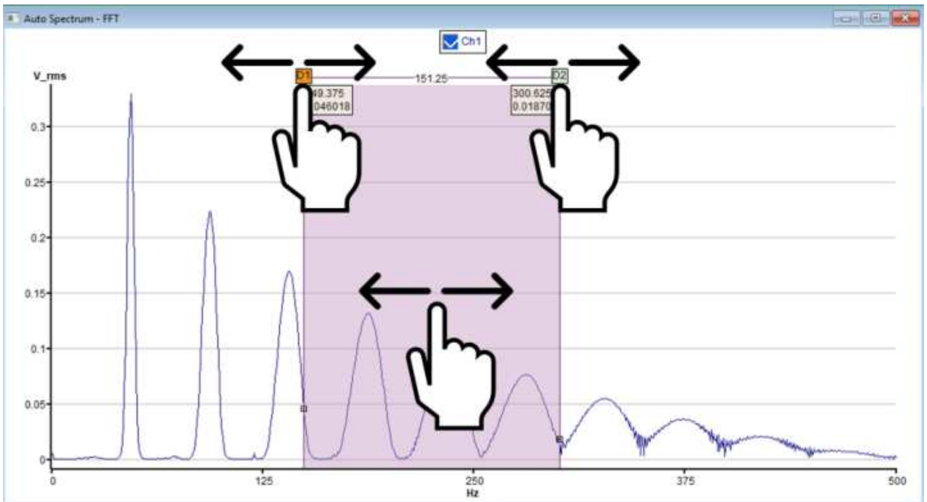
To move dual cursors together, first, highlight D1 and D2 by tapping somewhere between **D1** and **D2**. When the dual cursors are active, D1 and D2 are highlighted.

To move either **D1** or **D2** independently:

1. Select the edge of **D1** or **D2**
2. Drag and drop it to the desired location

To move **D1** and **D2** together:

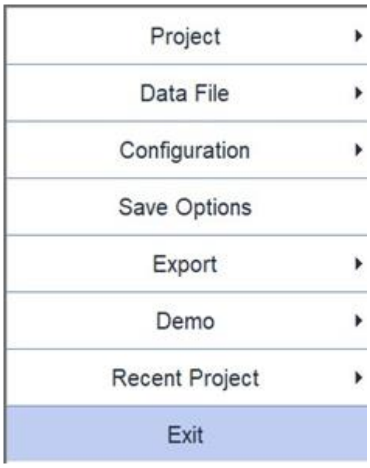
1. Select either the green arrows from the Chart toolbar to move left or right, or tap somewhere between **D1** and **D2** to move it.



Use the same procedure for sideband cursors.

6.9 File menu

Use the **File** menu to manage your data files in the dBX Analyzer. There are some basic principles that you can use throughout the software.



6.9.1 Project: New, Open, Save, and Save Project as

A project is an accumulation of setup parameters and measured data. When starting work with the dBX Analyzer, create a new project or open an existing project. Once a project is created, click **Save Project** or **Save Project As…** to save it to another project file.

Select **Project** from the main menu and press **OK** to display its sub-menu. With the sub-menu, you can create a new project, open an existing project, save a project, or save the project with a different project name.

- **New** – Select to create a new project.
- **Open** – Select to open an existing project.
- **Save** – Select to save the open project.
- **Save as** – Select to save the open project with a new project name.

Note that the Recorder raw data files are saved separately. To move a project with Recorder raw data files to a different folder, select **Export** > **Export project**.

6.9.2 Data File: Save or Add data

Select **Save** from the main menu and press **OK** to display its sub-menu. With the sub-menu, you can save the measured data to a file or add saved data to this project.

- **Save** – Select to save the test data to a file.
  A data file contains test data only, it does not contain display or measurement parameters as a project file does. When saving data, a pop-up window will show the files and the path where they will be saved on your Analyzer device. Saved data can be added to another project for further application.

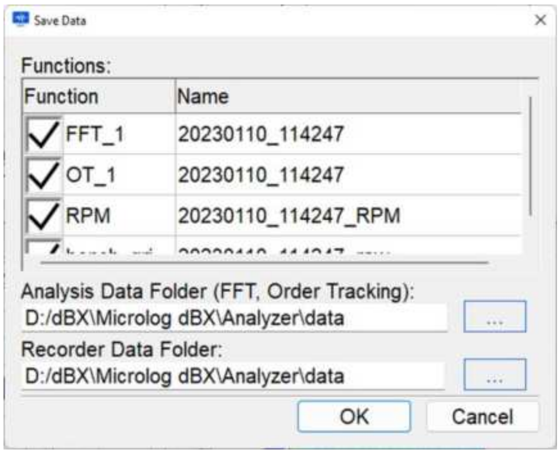


Figure 177 Save Data

- **Add** – Use this function to add dBX Analyzer format data to your project.
  When you select **Add data**, you can specify the data files you wish to add. For example, you can add a raw data file to this project for playback analysis. The added data file will be displayed in the History data panel.

6.9.3 Configuration: Save to default

Saves various setup parameters to a file called *default.nst.* Every time a new project is created, the default settings will be applied to this new project.

6.9.4 Save Options

Save Options allows the opportunity to specify more items on how the data can be saved with prompted file names, file path, analysis data or file exporting. This is useful when you have many files and want to specify a particular feature or feature subset in saving the files for easy recall.

- **File Name** – To define the default file name prompted by the system select **Save Option** > **File Name**

    - Edit **Prefix**, **Postfix** and **Note** used in the file name menu. The dBX Analyzer will use the setup to prompt a file name automatically while you are saving a project or data file. For example, you can select Channel ID to generate file names during a modal or ODS test. Select an item and use **Up** and **Down** buttons to change the order of the file name.

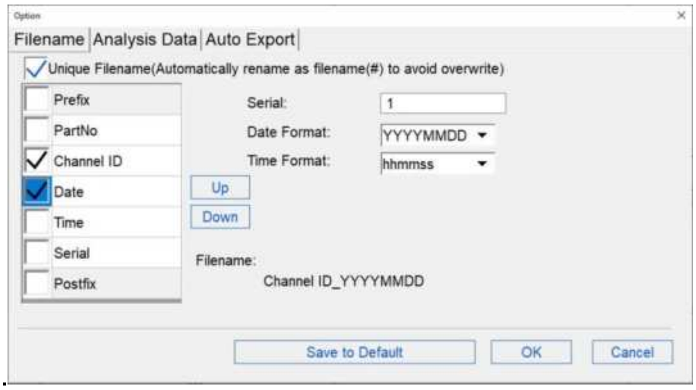


Figure 178 Save Option – File Name view

- **Analysis Data** – Enable or disable the auto save function after each measurement from this menu. Select the data types and destination folder(s) for the auto save function.

    - **Auto Save** – If selected, your selected analysis data will be automatically saved to the specified folder(s) each time the measurement is complete.
    - **Auto Prompt** – If selected, it will prompt you to save the data file(s) after each measurement.
    - **Open Folder** – Select destination folder(s) to save the data file(s).

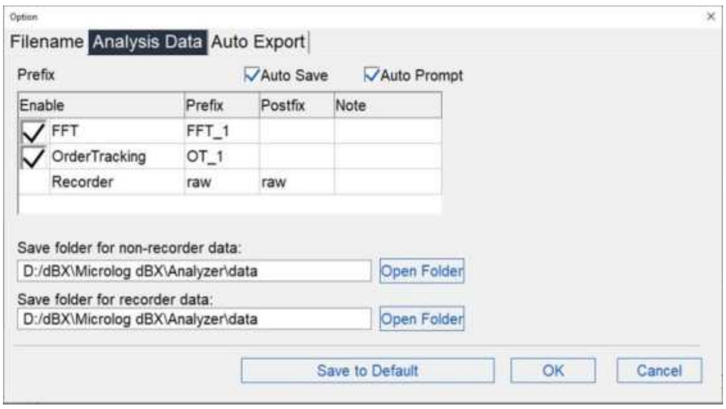


Figure 179 Save Option – Analysis Data view

- **Auto Export** – When Auto Export is enabled, dBX Analyzer will export measured data to the selected folder with a specified format whenever a data file is saved. Click **Add** to add new setup. You can enable multiple export setups at the same time.
  Select setup and click **Remove** to remove it from the list.
  For example, you can enable the UFF file auto export feature during a modal or ODS test procedure. These exported UFF files can be imported to a 3rd party Software for ODS or Modal analysis.

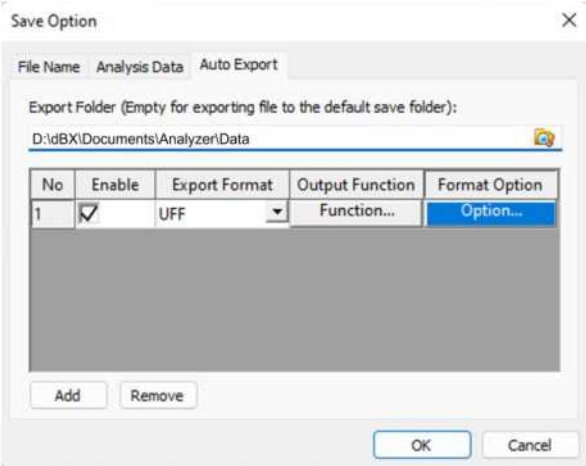
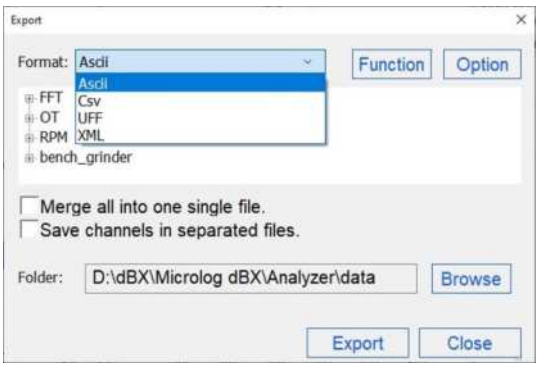


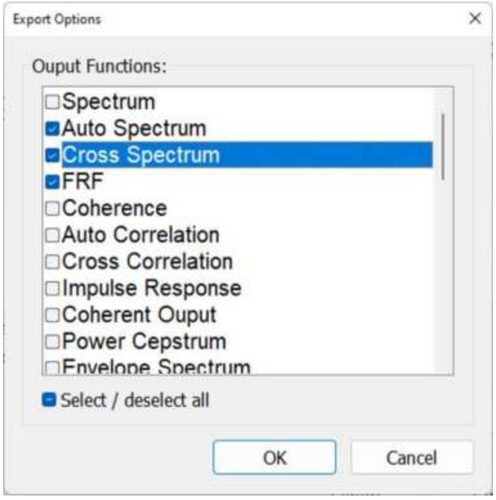
Figure 180 Save Option – Auto Export view

6.9.5 Export

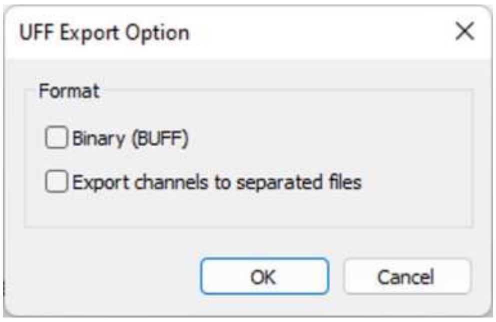
- **Export file**
  - **Format** – To export files in other file formats, from the **Format** dropdown, select one of the options:
    - **UFF** – Time Waveform format. File extension is *.uff* and it stands for universal file format
    - **ASCII** – Data record file format. File extension is *.TXT*
    - **CSV –** Data record file format. File extension is *.CSV*
    - **XML** – Extensible Markup Language. File extension is *.xml*



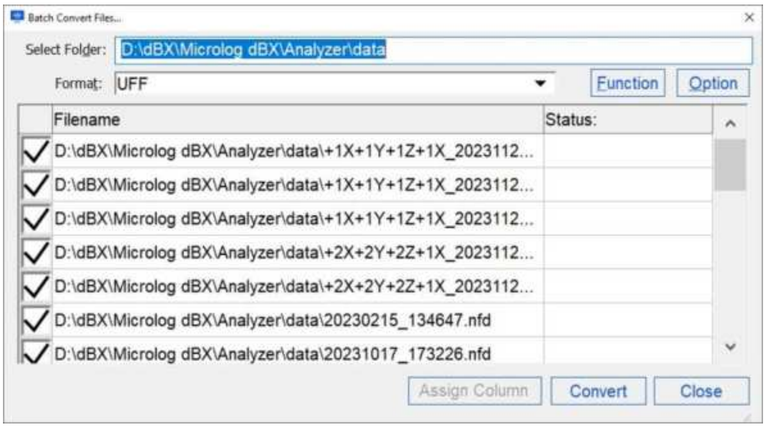
- **Function** – Select specific functions to be exported. Export options are **Time Waveform**, **Complex Spectrum**, **Auto Power Spectrum**, **Cross Power Spectrum**, **FRF**, **Coherence**, **Cepstrum**, **Demodulated Spectrum** and more.



- **Option** – Select export options



- **Batch export files** – Select to convert selected files in a selected folder into a specific format. This feature is useful for converting a large number of files at a time.



1. Tap **Select Folder** to select the folder where the data files are stored.

2. Check the **File Name** column to select the data files in the specific folder.

3. Tap the **Format dropdown** to select the file formats you wish to convert: UFF, ASCII, CSV, XML.

4. Tap **Function** to select the measured functions you wish to convert.

5. Tap **Option** to select the options for the data conversion.

6. Tap the **Assign Column** to assign the status (selected or unselected) to other columns.

7. Tap **Convert** to perform the file format conversion.

8. Tap **Close** to exit the view.

- **Export project** – If a project contains a Recorder raw data file(s), you'll need to export the project before you move it to another folder. That is because the Recorder files are saved separately and are **not** included in a project file.

6.9.6 Demo

Select **Demo**/**Grinder** to perform a playback analysis with the raw vibration data recorded from a bench grinder.

6.9.7 Recent projects

List the file paths of the most recently used projects. It is a helpful tool to help find the data files and most recent dBX Analyzer projects.

Exit – Tap to **Exit** the Analyzer module.

6.10 Mathematics menu

Once you have decided on the data to plot, you can now look at different aspects of the data on the plot with mathematics. An example is to convert the data from acceleration to displacement.

Tap the mathematics icon from the Toolbar to display a dropdown menu and select a mathematical operation for the active plot.

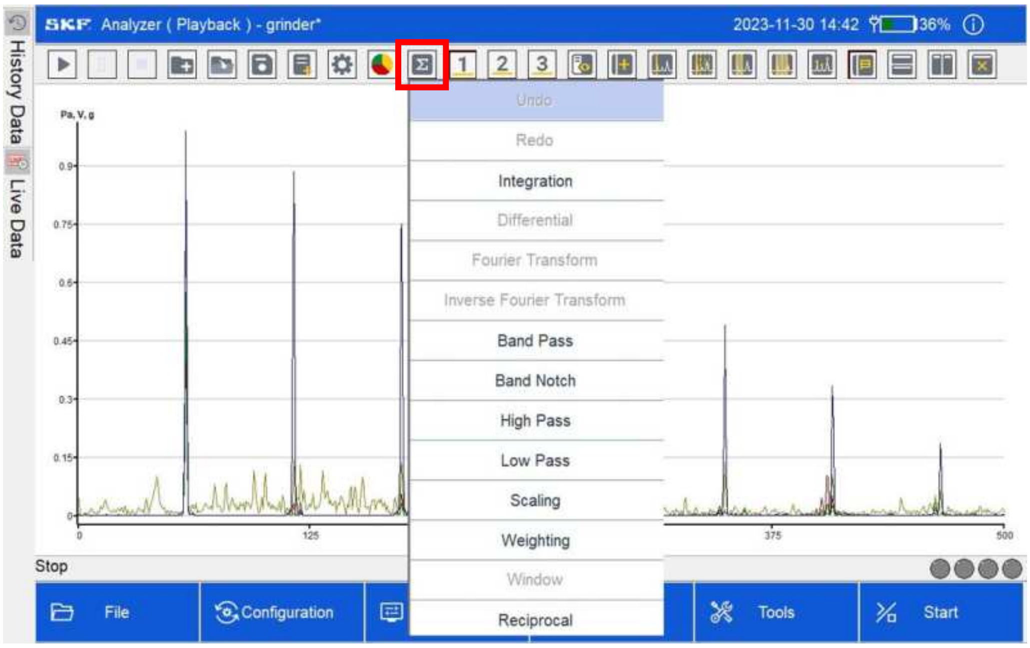


Figure 181 Mathematics menu

6.10.1 Integration / Differentiation

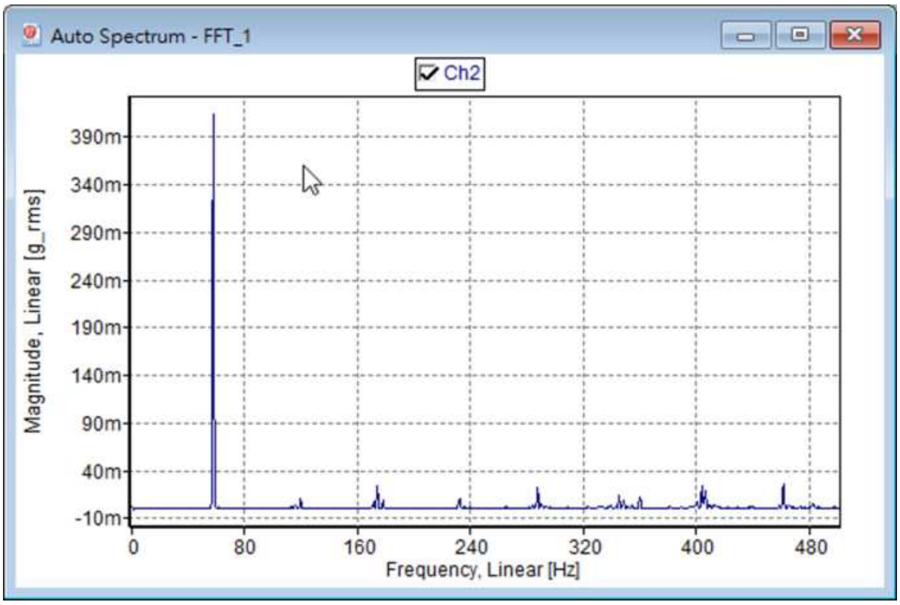


Figure 182 Auto Spectrum – FFT file example

In the plot below, we will convert **g's** (acceleration) to **um** (displacement) by selecting integration **two** times. This will convert this plot information to displacement.

To return to g's, select **Differentiation**.

Integration pressed once – The plot shows mm/s or Velocity.
The integration function is selected again, now the plot shows **um** or displacement of the signal.

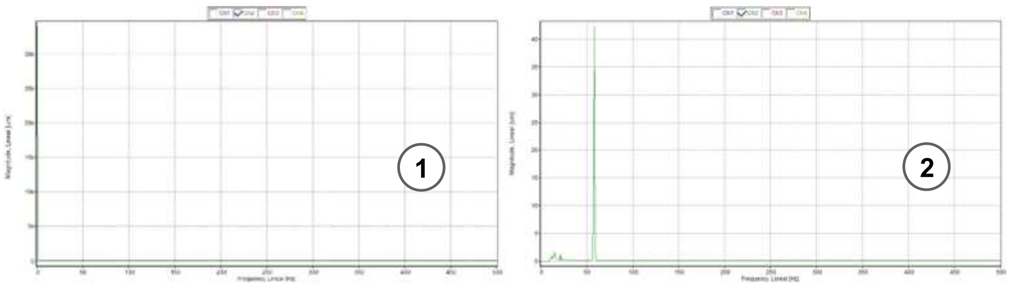


Figure 183 Integration function – example

1. After two times integration
2. Filter out low frequency noise

6.10.2 Fourier transform / Inverse Fourier transform

Inverse Fourier transform converts the plot from **Hz** to **Time**, as shown in the plot below. To go back, select **Fourier Transform**.

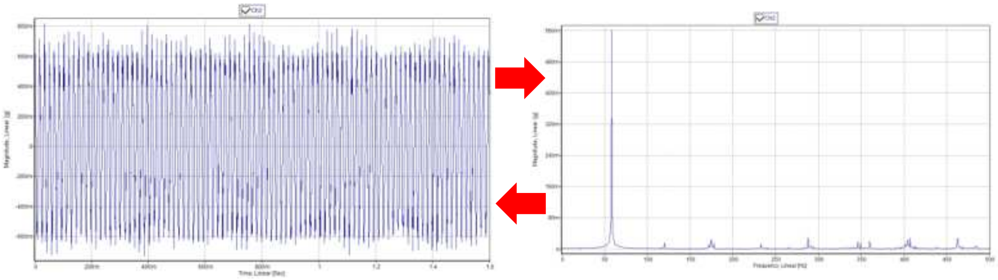


Figure 184 Fourier transform – example

6.10.3 Applying filters

Adding filters to the data is very helpful for eliminating or reducing noise in your measurement or looking at the data differently. Several filters are available.

- **Band pass filter** – Filters out frequencies outside the specified band. Place the cursor on the filter's edge to slide to desired placement on the plot.

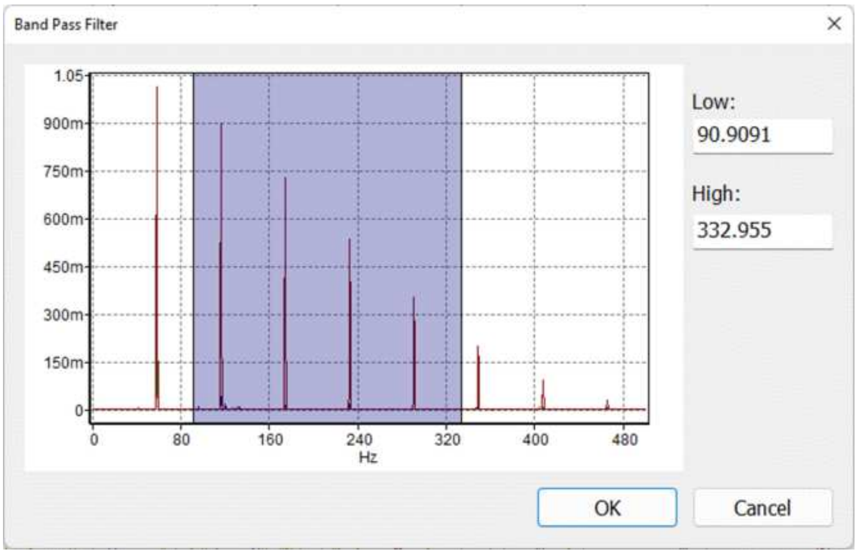


Figure 185 Band pass filter – example

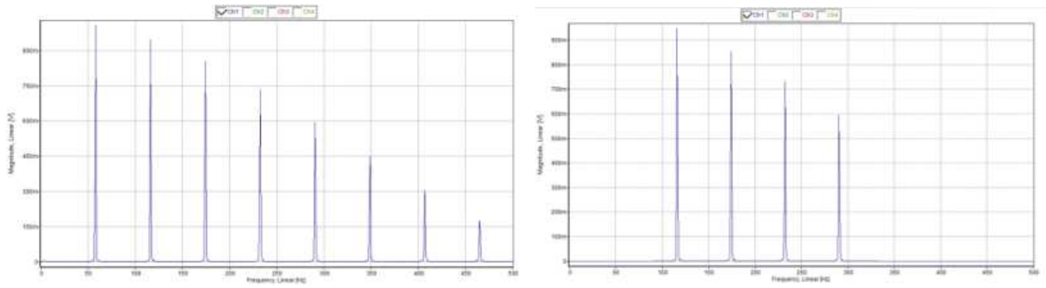
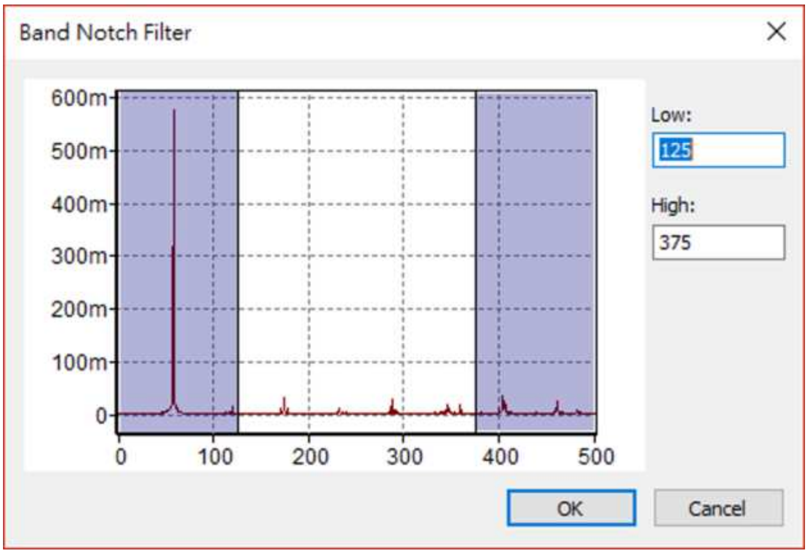


Figure 186 Before Band Pass applied (left) After Band Pass applied (right)

Band notch filter – Filters out the area between the specified band.



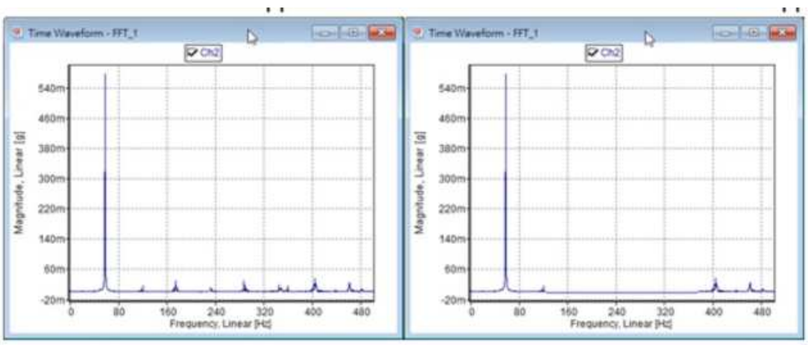
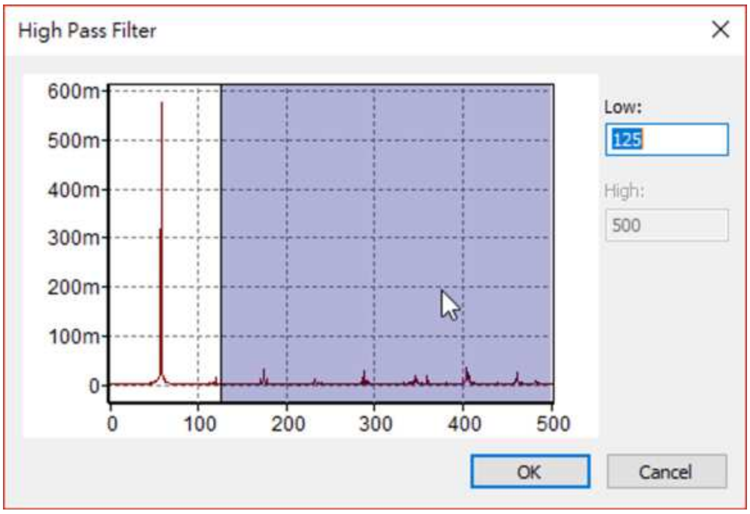


Figure 187 Before Band Notch applied (left) After Band Notch applied (right)

- **High pass filter** – Passes high frequencies well but attenuates (i.e., reduces the amplitude of) frequencies lower than the filter's cut-off frequency.



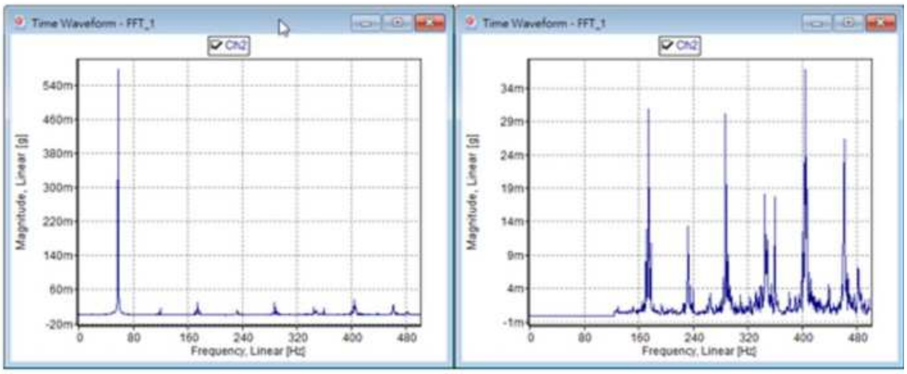
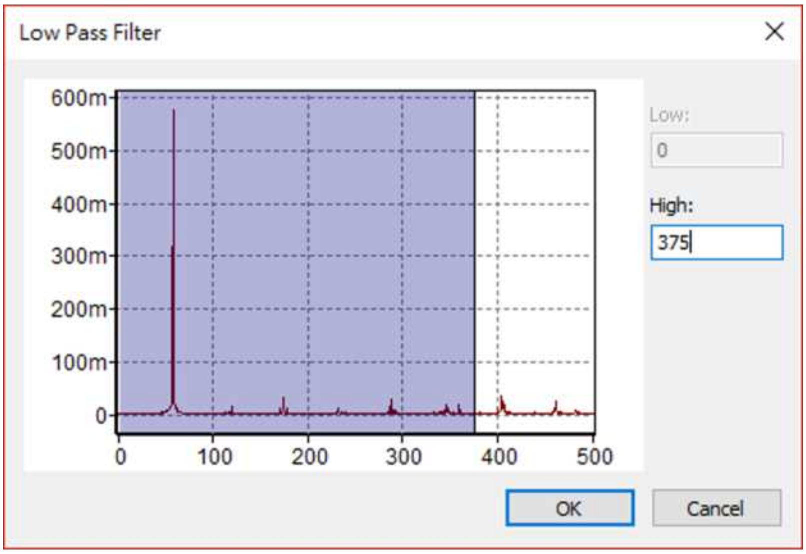


Figure 188 Before High Pass applied (left) After High Pass applied (right)

- **Low pass filter** – Passes low-frequency signals but attenuates (reduces the amplitude of) signals with frequencies higher than the cut-off frequency.



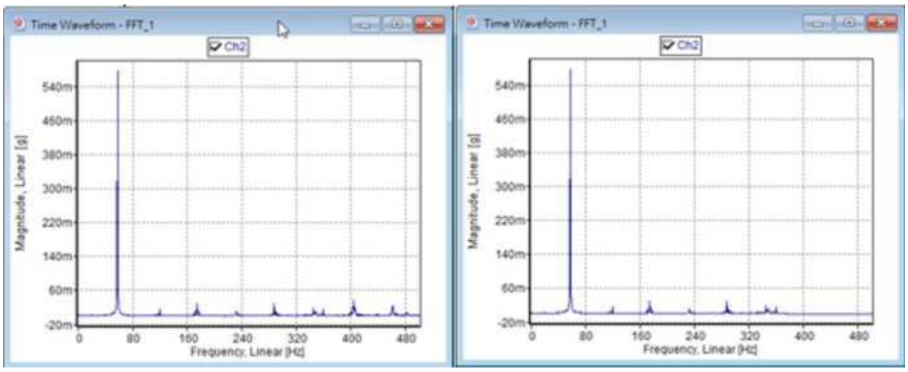
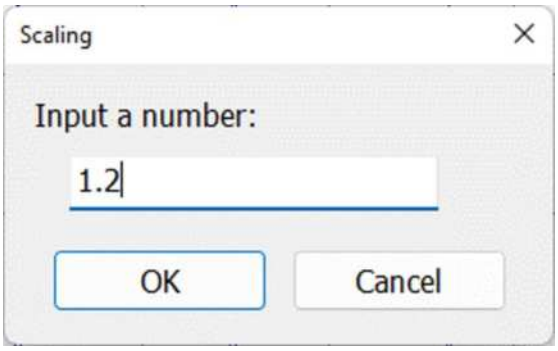


Figure 189 Before Low Pass applied (left) After Low Pass applied (right)

- **Scaling** – Enter a value to scale the data in the active window.



6.10.4 Window functions for Time Waveform

To remove this leakage error, it may be necessary to apply a window function to the time waveform before conducting FFT. Because there are many shapes of signals, there are different windowing functions to best suit the situation.

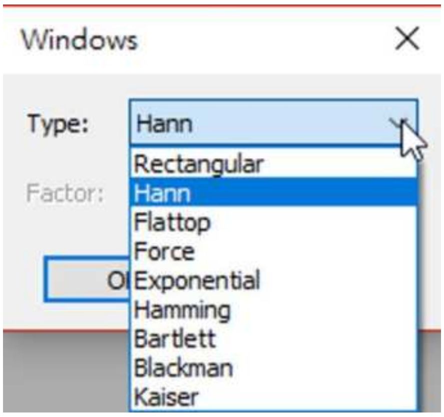


Figure 190 Windows type selection

From the dropdown menu, select **Rectangular**, **Hann**, **Flattop**, **Force**, **Exponential**, **Hamming**, **Bartlett**, **Blackman,** or **Kaiser** window.

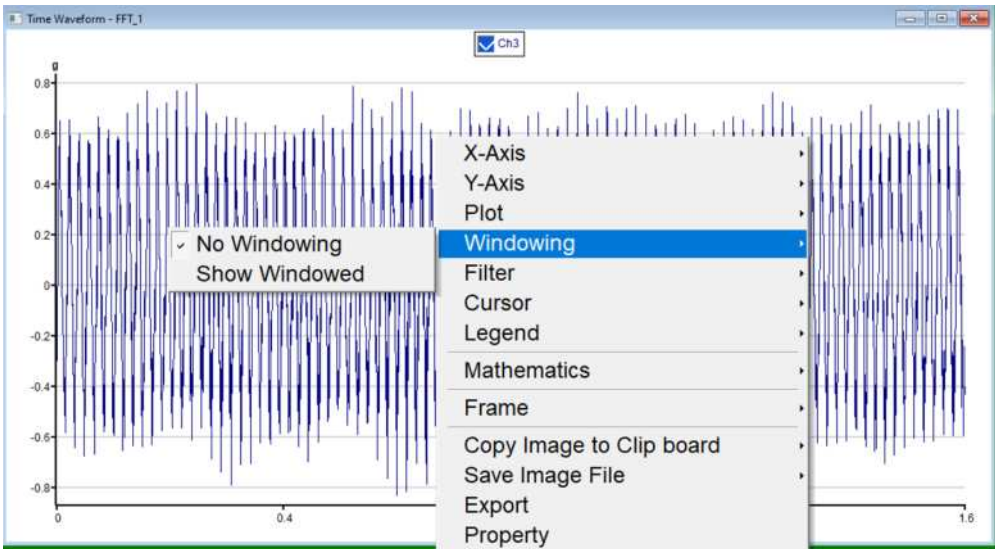


Figure 191 Right-click menu – Windowing

After a window is applied to a time waveform, the waveform display remains unchanged. To see the windowed waveform, press the right-click key and select **Windowing** > **Show Windowed** to display it as follows.

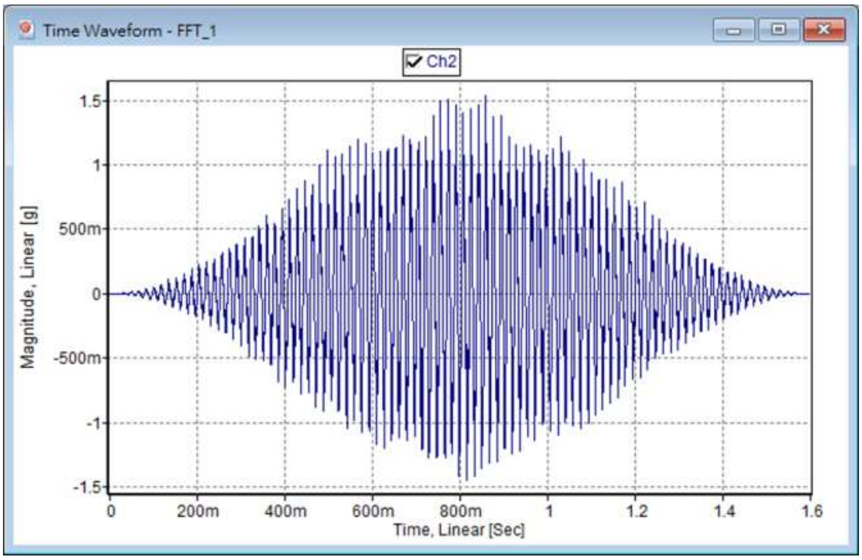
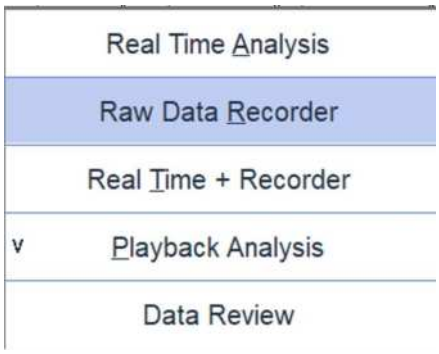


Figure 192 Apply windowing view – example

6.11 Mode menu

Use the Mode menu to select the measurement mode you wish to perform. Select one of the five modes: **Real Time Analysis**, **Raw Data Recorder**, **Real Time + Recorder**, **Playback Analysis**, or **Data Review**.



The selected mode will also work with different types of measurements. To measure FFT real time with Octave analysis, select **Real Time Analysis**.

6.11.1 Real Time Analysis

In Real Time Analysis mode, data is measured real time.

6.11.2 Raw data recorder

A Raw Data Recorder measurement records the raw data without post-processing. It is useful to capture the real event compared to taking a snapshot as an FFT analysis does.
Raw Data Recorder is designed to capture data continuously and transfer the raw signal to the hard drive. The raw data file can be replayed in the playback mode to perform post-analysis.

Select **File** > **Save data** to save the raw data file for playback analysis in the future.

A Recorder raw data file consists of two parts:

- *.nrd* files contain the header information
- *.nra* file(s) contains the measured data

6.11.3 Real time + Recorder

Real Time + Recorder combines Real Time FFT, Order Tracking and a Raw Data Recorder. This mode intends to take FFT and/or Order Tracking data in real time and capture the Raw data of your measurement as a raw data file at the same time.

6.11.4 Playback analysis

Use this mode for review and analysis of raw data record files. Use the playback mode to **replay** a saved raw data file as **live** data. The playback mode allows you to recreate the event as if you were on-site measuring in real time.

6.11.5 Playback analysis by adding a raw data file

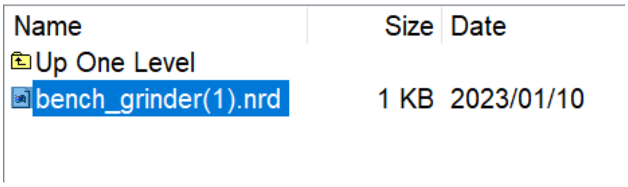
To perform a playback analysis, follow the instructions below.

1. To create a new project, select **File** > **New Project**.

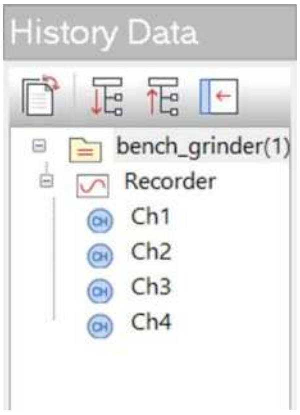
2. Select **Mode** > **Playback Analysis**.

3. To add a raw data file to the project, select **File** > **Add data**.
   Note that the raw data consists of two parts:
   a. The *.nrd* file contains the header and file information
   b. The *.rna* file contains the raw data itself

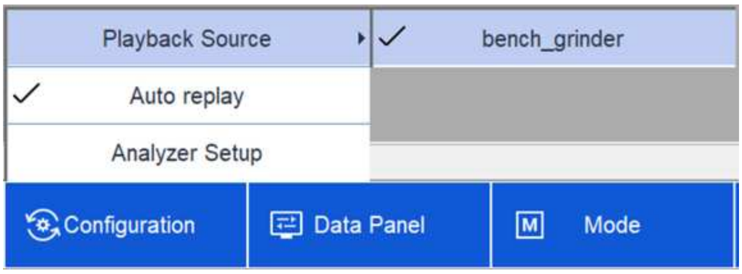
   Always select the *.nrd* file and add it to the project.



4. The added raw data file(s) is now visible in the **History data** panel.



5. From **Setup** > **Playback Source** select the raw data file.



6. Go to the **Setup** menu and set the parameters (or load a setup file from the **File** menu) for your analysis.

7. Press **Start** / **Stop** to perform the playback analysis.

6.12 Playback mode

Playback mode is used for review and analysis of raw data record files. Playback mode is used to replay a saved file as if it was live data. Playback mode allows you to recreate an event as if you were on-site performing the measurement in real time.

6.12.1 Performing a playback:

1. Set the mode to playback analysis.

2. Add a raw data file.

3. Adjust the parameters: go through the analyzer Setup menu and set up FFT or Order tracking parameters.

4. Create plots from the **Live Data** panel and set up the display layout(s).

5. Press **Start** on the measure toolbar to begin the playback of your raw data recorder file.

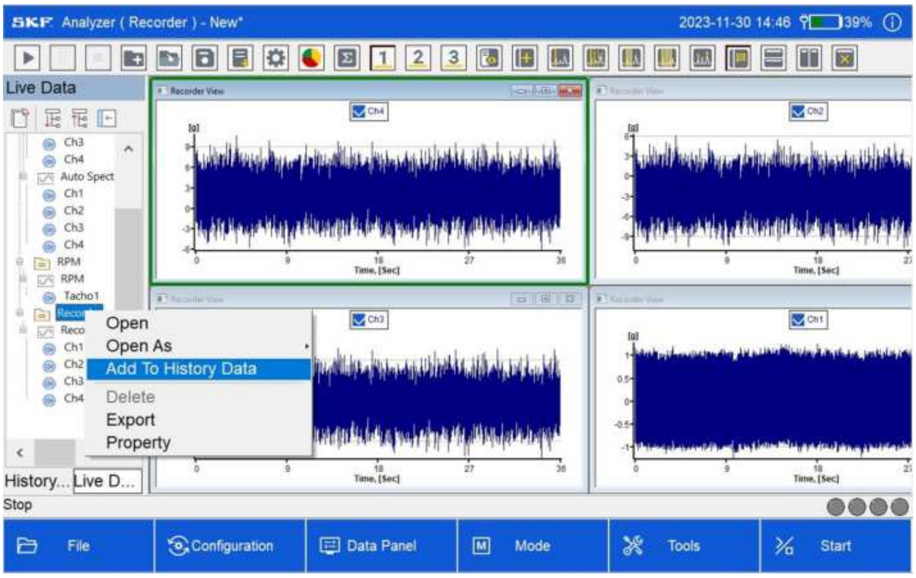
6. Press **Stop** on the measure toolbar to stop the measurement.

7. Perform mathematics, place cursors, notes etc. save/export the file or create reports as needed.

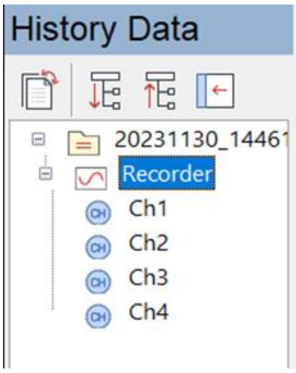
6.12.2 Playback analysis after raw data recording

Another way to perform a playback analysis is to change the instrument mode to Playback Analysis right after the raw data recording from **Recorder** or **Real-time + Recorder** mode.

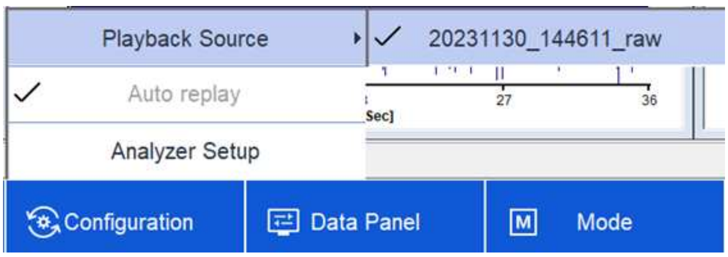
1. After the recording, the recorder function is available in the **Live data** panel. Move the cursor to highlight the **Recorder**, right-click to display the options and select **Add to History Data**.



2. In the **History data** panel, you can see the added raw data.



3. Change instrument mode to Playback Analysis and select the Source data file from **Setup** > **Playback source**.



4. Set up the parameters and start the playback analysis.

6.12.3 Data review

Use the Data review Mode to review and perform analysis on the previously measured data files.

6.13 Configuration menu

The Configuration menu contains three setup menus related to performing a measurement:

- **Analyzer Setup**
- **Channel Setup**
- **Engineering Unit**

Specific module setups are **FFT**, **Recorder**, **Octave,** and **Order Tracking**.

6.13.1 Analyzer setup

Use the Analyzer setup to assign the selected channels to desired analysis module(s), determine the tachometer channel, and set up the tacho sensor. You will notice available analysis modules under the authorization column, depending on what was purchased.

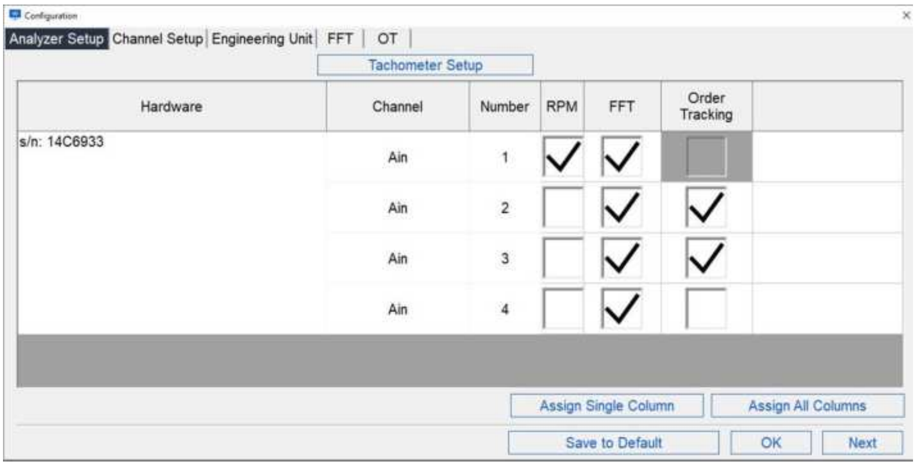


Figure 193 Analyzer Setup view – example

With this Analyzer setup menu, you can assign an analysis software module to each channel.
In the example above, Channels 1, 2, 3 and 4 are assigned for FFT analysis, and Channels 2 and 3 for Order Tracking analysis.
The tachometer channel is used for measuring a tachometer signal.

Note: To perform Order Tracking analysis, at least one tachometer channel must be enabled.

As the real-time measurement is started, both analysis modules will run parallel at the same time.

- **Save to Default** – Save all the current settings to a default file (*default.nst*).
- **Tacho Setup** – Opens the Tachometer Setup screen.

6.13.2 Channel setup

- **Channel ID** is divided into two groups:
    - **Modal** uses channels IDs for modal analysis or ODS analysis, where you assign the Point number and the direction to each channel.

Figure 194 Channel Setup view – Modal – example

- **Machine** – Labels are given for the type of machine used.

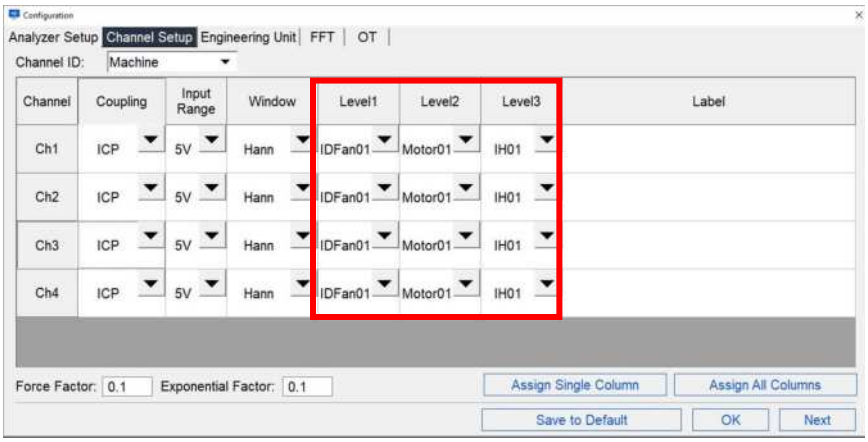


Figure 195 Channel Setup view – Machine – example

- **Channel** – Displays the channel number.
- **Coupling** – Set sensor coupling types. Select **AC**, **DC** or **ICP**.
- **Input Range** – Set input voltage range (5 V or 20 V).
- **Window** – Set the window function: **Uniform**, **Hanning**, **Flattop**, **Force**, **Exponential**, **Hamming**, **Bartlett**, **Blackman**, **Kaiser**.
- **Node** – Set measurement location number.
- **Direction** – Set measurement location direction (Combo: +X/-X/+Y/-Y/+Z/-Z+R/-R/+T/-T/+P).
- **Advance Number** – Sets the advance number for the next measurement.
- **Label** – Enter the channel label as memo.
- **Force Factor** – Enter the force window width. In the above example, the Force window width is 0.1 = 10%
- **Exp. Factor** – Enter the attenuation rate at the end of the time block when an exponential window is selected.
  In the above example, Exp. Factor = 0.1 = 10%. That means at the end of the time block, the weighting value is equal to 0.1

Definition of Exponential function	x(t) = EXP (-Bt)
Definition of Exponential factor X	X = EXP (-Bt) at t = T Where **T** is the length of the time block. **X** is the attenuation rate of the exponential window at the end of the time block. This should be a real number between 0 and 1

- **Assign Single Column** – Repeats selected data in that column.
- **Assign all columns** – Repeats all columns with data in the selected channel.
- **Save to Default** – Save the current setup as the default, same as selecting **File** > **Setup/Save to Default**.
- **OK** – Save the setup and exit the setup view.
- **Next** – Enter the next setup page.

6.13.3 Tachometer Setup

Understanding the Tachometer input
A pure TTL signal is typically a 0–5 V signal that uses a tachometer and optical tape to generate the “tach” or signal for the 1x per rev signal. Microlog dBX is equipped with a tachometer input channel and a built-in power supply, which can power SKF laser tachometer sensor directly. Besides that, you may use one of some of the analogue input channels as tachometer input. Each revolution is calculated based on the zero crossing of the phase angle. The tachometer waveform can be observed in the channel designated in the FFT. You can have more than one tachometer input if desired.

Tap **Tacho Setup** and in the pop-up window you can enter various parameters for proper triggering.

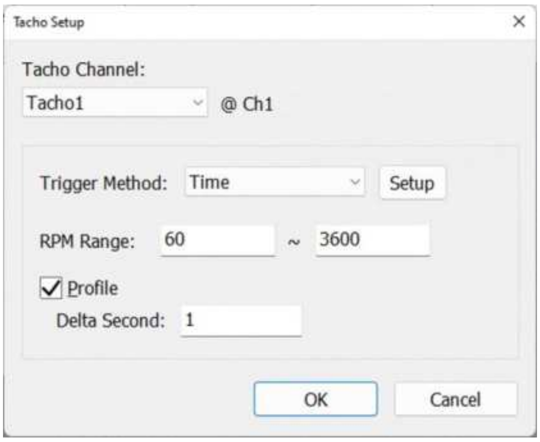


Figure 196 Tacho setup – example

- **Tacho Channel** – Select a channel from the dropdown. The available Tacho Channels are defined in the Analyzer setup menu.

- **Trigger Method** – Time or Spectrum.
  If a tachometer sensor is used, always select Time and the Tacho Channel will be triggered from the time waveform. If no physical tachometer sensor is used, and you still need to measure the rotation speed, use **Spectrum** triggering.

6.13.4 Time triggering setup

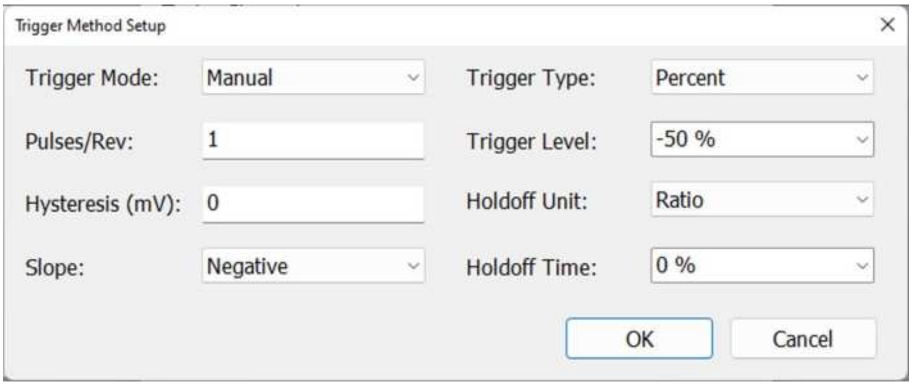


Figure 197 Trigger Method Setup view – example

- **Trigger Mode** – **Manual** for manual setup, **Auto** for Auto-detection. For using a standard TTL tachometer sensor, Auto detection is recommended. If you are using a non-TTL sensor, which also contains noise, select **Manual**, and Analyzer provides a very flexible setup to detect the tachometer correctly.

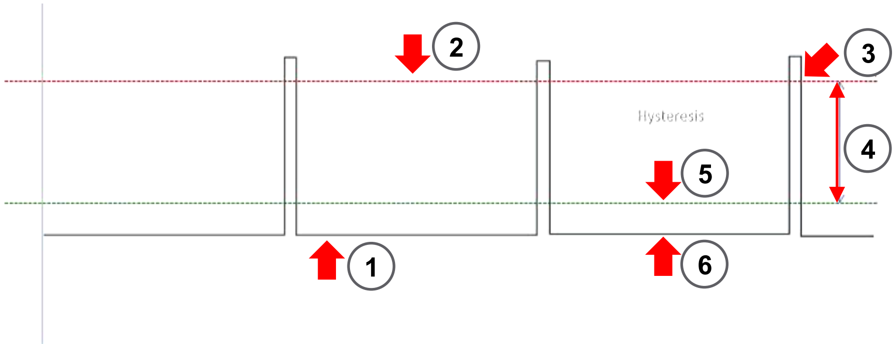


Figure 198 Tachometer detection – example

1. Tacho signal
2. Trigger level
3. Trigger point
4. Hysteresis
5. Hysteresis level
6. Reference level

- **Hysteresis(mV)** – Type the Hysteresis voltage level.
  This setting is used to exclude fake signals when there is noise from the tachometer sensor. In the illustration plot above, the tachometer signal is raised from the **Reference level** and reaches the **Trigger level** at the **Triggering point**. The Hysteresis defines the minimum “depth of drop” after a triggering. If does not drop down to the hysteresis level again, any new triggering will be ignored.

- **Slope** – Select trigger slope as positive (ascending), negative (descending) or both.

- **Trigger Type** – Select by millivolt level or by percentage.
- **Trigger Level** – Select a percentage of full range, or a voltage level.
- **Holdoff Unit** – Select millisecond or ratio.
- **Holdoff Time** – Select a **ms** (milliseconds) number or ratio number. Holdoff time is used to exclude Noise too. It defines the minimal time span between two triggering points.

  For example, if Holdoff time is set as **20 ms**, the maximum rotation speed would be **1/20 ms = 50 Hz = 3000 RPM**.

  Any tachometer signal providing a number higher than **3000 RPM** would be considered as Noise and ignored.

6.13.5 Spectrum Triggering setup

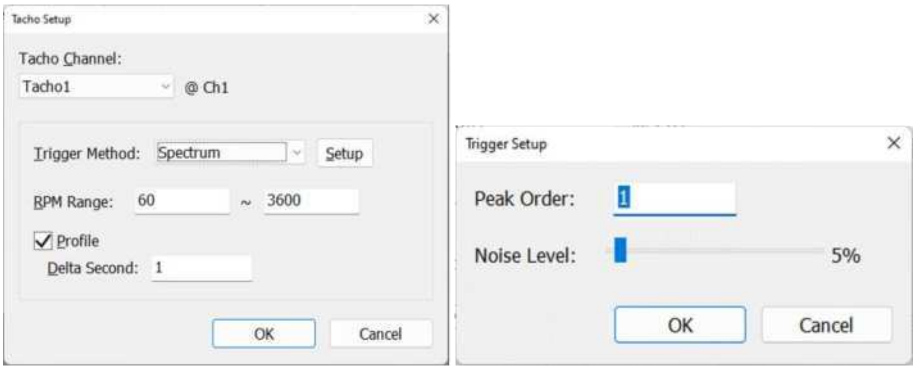


Figure 199 Trigger setup – example

When here is no physical tachometer sensor connected to your dBX Analyzer:

1. Select the **Trigger method** as **Spectrum**
2. Click **Setup** to show the setup menu

Input the **Peak Order** with a number. For example, if Peak Order is specified as **1**, dBX Analyzer will take the frequency of the maximum peak as order **#1** and then calculate the rotation speed from the frequency of that peak.

The following example shows how to measure the rotation speed when imbalance vibration is the dominating signal. Another example for conducting order tracking analysis on a 4-cylinder 4-stroke engine, which has a dominant signal in the 2nd order, you can set Peak Order as 2.

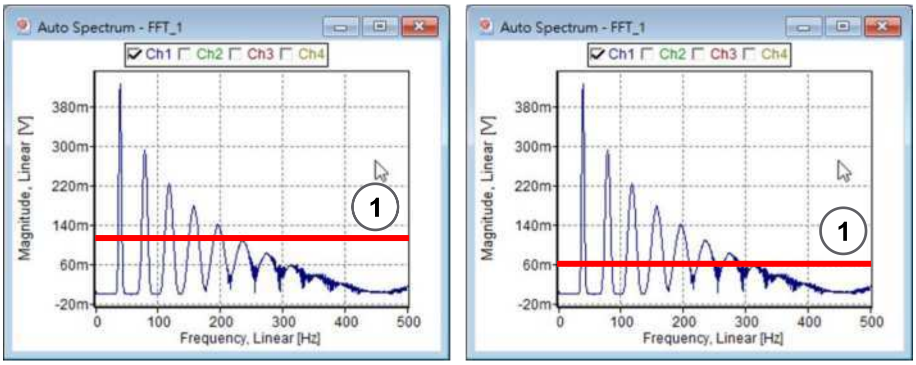


Figure 200 Auto Spectrum Rotation Speed – example

1. Noise level

Note: When Spectrum trigger is selected, there is **no phase information** available.

6.13.6 Engineering unit setup

From the toolbar, select **Setup** > **Engineering Setup**.
Engineering Unit setup is used for selecting the sensor type, sensor units, sensitivity, and other parameters for your measurement.

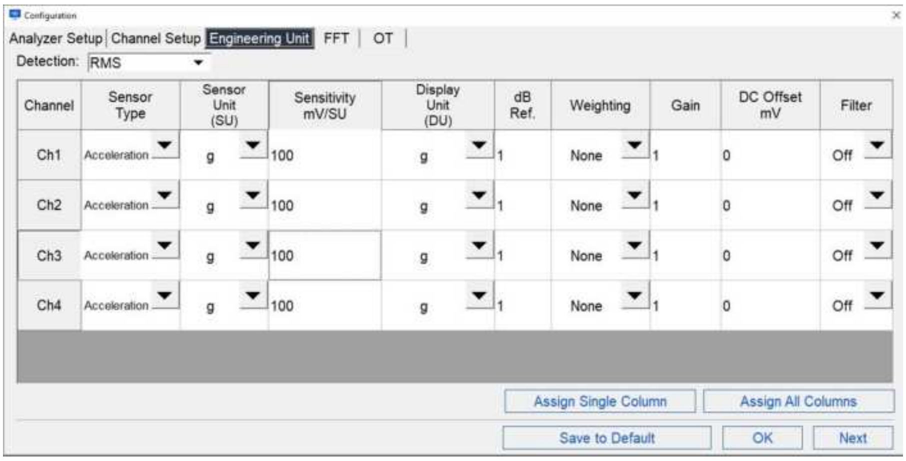


Figure 201 Engineering Unit view – example

- **Channel** – Shows channel number generated by the **Analyzer Setup**.
- **Sensor Type** – Set sensor type: **Voltage**, **Acceleration**, **Velocity**, **Displacement Force**, **Pressure**, **Temperature** or **Custom**.
- **Sensor Unit (SU)**
  - Acceleration: g, m/s^2
  - Velocity: mm/s, in/s, cm/s

  - Displacement: mil, µm (um)
  - Force: N, g
  - Sound: Pa
  - OFF: Volt

- **Sensitivity** – Enter the sensors' sensitivity in millivolts per sensor unit (SU).

- **Display Unit** – The displayed unit can be different from the sensor unit to display the units in something other than what is calibrated. Select the Display unit based on the Engineering Unit Table.

- **dB Reference** – dB reference value A_0: e.g. A_0 of Sound signal: 2e-05 Pa.

- **Weighting** – Add a weighting function to a selected channel. For example, weighting A, B, C and D are used for acoustic measurements. ISO 2631 is used for complete body vibration measurements, and ISO 6954 is used for vibration evaluation concerning habitability on passenger and merchant ships.
- **Gain** – Sets the Gain

- **DC Offset** – Set DC Offset

- **Filter** – Applies a high pass filter to the measurement. Select **OFF**, **0.1Hz**, **0.5Hz**, **1Hz**, **2Hz**, **5Hz**, **10 Hz** or **20Hz**). If the display unit is derived from integration, use a high pass filter to filter out the ski slope noise at the low-frequency zone.

- **Assign Single Column** – Repeats selected data in that column.

- **Assign All Columns** – Repeats all columns with data in the selected channel.

- **Detection** – Set signal detection to: **RMS, Peak,** or **Pk-Pk**.

6.13.7 FFT setup

From the toolbar, select **Setup** > **FFT setup**. In this section, you can select **Function**, **Frequency**, **Averaging**, **Trigger** and **Map**.

- **Function**

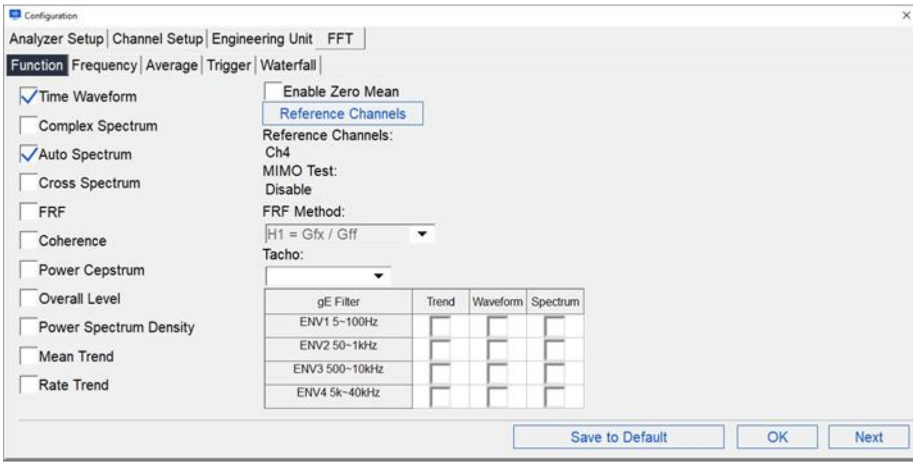


Figure 202 FFT – Function view – example

Select **Time Waveform**, **Complex Spectrum**, **Auto Spectrum**, **Cross Spectrum**, **FRF**, **Coherence**, **Power Cepstrum**, **Overall Level**, **Power Spectrum Density**, **Mean Trend** or **Rate Trend**.

- **Auto Spectrum** – Also known as the Power Spectral Density (PSD), represents the distribution of power across the frequency components that make up the signal. It is derived from the Fourier transform of a signal and provides information about the energy present at different frequencies within a single time series.

  Use: It is primarily used to identify dominant frequencies and their amplitudes in a vibration signal.

- **Cross Spectrum** – Used to analyse the relationship between two different time series. It provides information on how the two signals are correlated in the frequency domain.

  Use: It helps in identifying common frequencies between two signals and understanding the phase relationship between them. This is particularly useful in identifying how different parts of a system interact with each other.

- **Complex Spectrum** – Includes both the amplitude and phase information of a signal. It is represented as a complex number, combining the real and imaginary parts of the Fourier transform.

Use: It provides a more comprehensive view of the signal, allowing for the reconstruction of the original time-domain signal from its frequency components.

- **Coherence** – A function used to examine the relationship between two signals. It is a real function with values ranging between 0.0 and 1.0.

  Use: It is commonly used in frequency response function measurement to check the quality of signals and signal measurements.

- **Frequency response** – Frequency response of a system is a quantitative measure of the magnitude ratio and phase difference between the output and input signals.

  Use: It is widely used in investigating resonance problems of a structure, conducting experimental modal testing, and ODS testing

- **Power Cepstrum** – A tool used in signal processing to analyse periodic structures within a signal. It is defined as the inverse Fourier transform of the logarithm of the power spectrum of a signal.
  The Power Cepstrum is particularly useful in identifying periodicities in the frequency domain, which can be related to echoes or reflections in the signal.

  Use: To detect periodic components in mechanical vibrations.

Select the **gE Envelope** filter(s) and the corresponding functions: overall trend, gE waveform or gE spectrum. You can see the selected functions in the **Live data** panel.

- **Enable Zero Mean** – Enabling Zero Mean will remove the DC offset of the time waveform digitally.
- **Reference Channels** – dBX Analyzer always uses **channel 4** as the reference channel. The reference channel is used to calculate the FRF and Cross-Spectrum, which are used by the ODS analysis.

- **Tacho** – Select a tachometer for the FFT measurement. This feature will be used for controlling the RPM step during a Waterfall 3D measurement. Note that the tachometer needs to be selected from the Analyzer setup in advance.

- **gE Filter** – Select gE Filter(s) for Enveloping analysis. You can select to measure gE Overall Trend, gE Envelope Time Waveform or gE Envelope Spectrum for each gE filter.

- **Frequency** – Sets the Frequency Band, Resolution, Zoom FFT, and Overall Frequency Band.

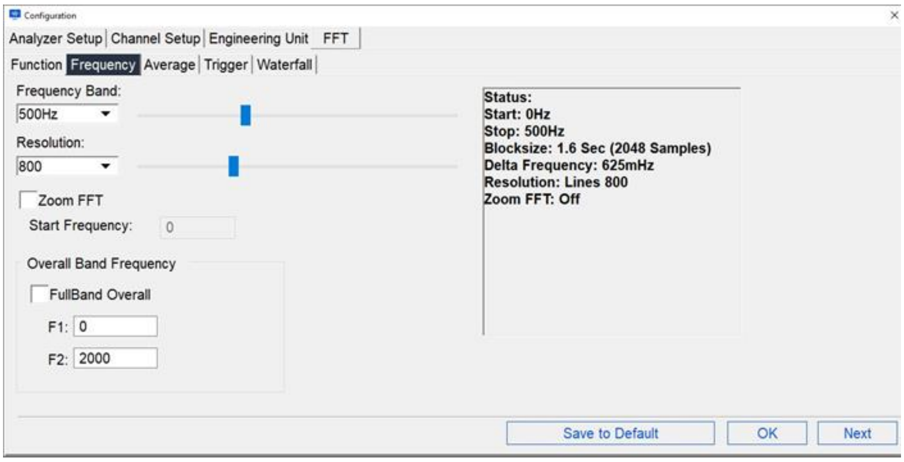


Figure 203 FFT – Frequency view – example

- **Frequency Band** – Set the Frequency Bandwidth: 50 Hz, 100 Hz, 200 Hz, 500 Hz ,1 kHz, 2 kHz, 10 kHz, 20 kHz or 40 kHz.
- **Resolution** – Select the number of resolution lines.
- **Zoom FFT** – Check to enable zoom FFT.
- **Start Frequency** – Enters the initial start frequency for zoom FFT (It will not work if the Zoom FFT is not selected.
- **Overall Frequency Band** – Enters the frequency band of overall level. The overall level is calculated within the band between F1 and F2 from an FFT Spectrum.
- **Status**
  - **Start** – Displays the start frequency.
  - **Stop** – Displays the ending frequency.
  - **Delta f** – Frequency resolution.
  - **Resolution** – Lines of resolution.
  - **Zoom FFT** – Zoom FFT start frequency.

- **Average** – Set the averaging for the FFT measurement**.**
  Select: **Spectral** or **Time Average**, **Average Type**, **Average Count** and **Overlap Level**. When performing modal measurements, select **Overload Reject** and **Preview**.

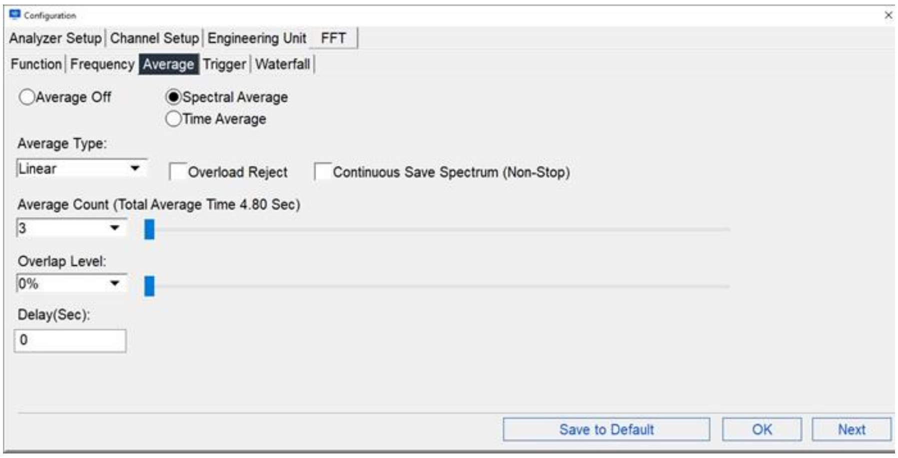


Figure 204 FFT – Average view – example

- **Average**
  - Average Off – No averages taken.
  - Spectral Average – Averages data in the frequency domain.
  - Time Average – Averages data in the time domain.

- **Average Type** – Select **Linear**, **Exponential** or **Peak Hold**.

- **Average Count** – Use the dropdown menu or the slider to set the average number.

- **Overlap Level** – Use the dropdown menu or the slider to set the overlap percentage.

- **Overload Reject** – Check the box to reject an overloaded signal.

- **Continuous save spectrum (non-stop)** – Enabling this option will auto-save the spectrum to a specified folder after the average number is achieved, and then it will perform measurements repeatedly. Press the **Start** / **Stop** key to terminate the procedure. The specified folder can be set from the **File** > **Save option**.

- **Delay (Sec)** – Tap to enter the time value in seconds for measurement delay. When the signal is not stable at the beginning of a measurement, set a delay time to skip that unstable phase.

- **Trigger** – The Trigger setup is used to trigger or begin a measurement. A trigger signal is also used as the reference of phase measurements.

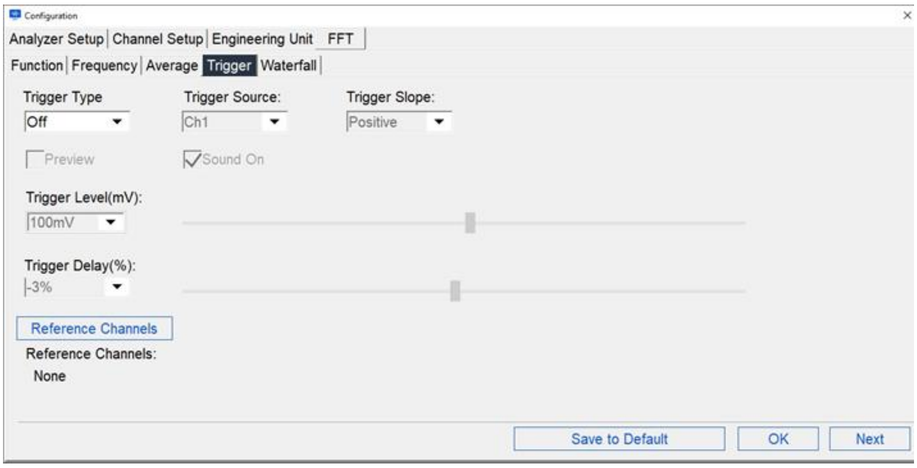


Figure 205 FFT – Trigger view – example

- **Type of Trigger** – Select **Off**, or **Channel.**
- **Trigger Source** – Select channels for the trigger source.
- **Trigger Slope** – Select either positive or negative slope.
- **Trigger Level** – Use the dropdown menu or the slider to set the trigger level.
- **Preview** – Check the box to open the Signal Preview function. A new window for every measurement will open, showing the Time Waveform data. Select **Accept** or **Reject** the measurement during the averaging process. This feature is useful when doing a bump test and checking the signal quality before accepting it.

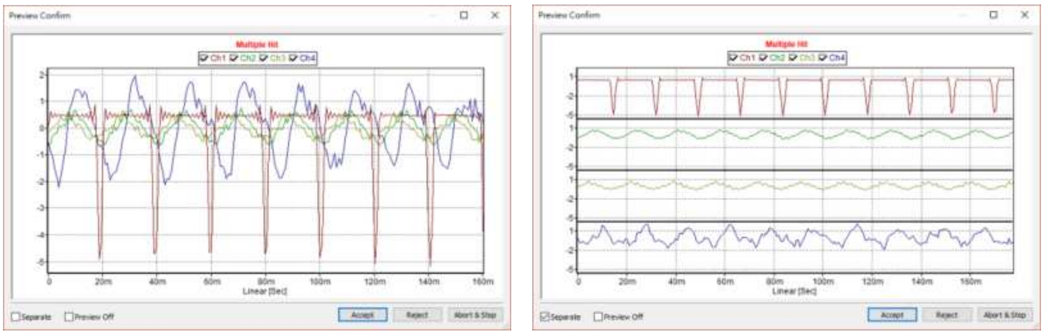


Figure 206 Channels Overplayed (left) Channels Separated (right)

- **Separate** – Each channel will be separated instead of overlaid.
- **Trigger Delay (%)** – Use the dropdown menu or use the slider to set the trigger delay. A negative trigger delay is useful for capturing the complete waveform of a bump test. -3% to -5% trigger delay is suggested for such applications.

- **Waterfall** – 3D Waterfall functions for the FFT measurement can be set for Time Waveform, Complex Spectrum, Auto Power Spectrum, and Cross Power Spectrum. The sampling rate is chosen based on the RPM and bandwidth.

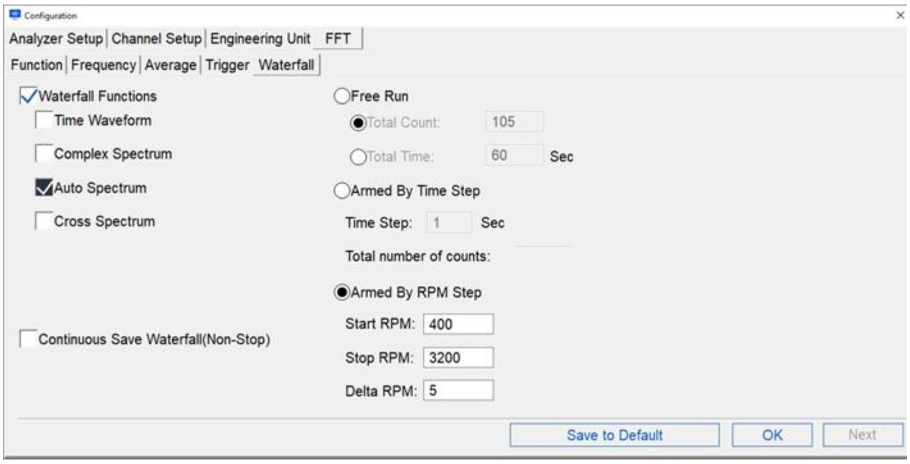


Figure 207 FFT – MAP view – example

  - **Waterfall functions** – When the Waterfall Function box is checked, it will enable 3D waterfall measurements for **Time Waveform**, **Complex Spectrum**, **Auto Spectrum**, **Cross Spectrum** and **Envelope Spectrum**.

  - **Free Run** – If the Free Run box is checked, it will cause a measurement to continue until the entered total number of counts or time is reached.

  - **Armed by Time Step** – Checking the Armed by Time step will cause measuring every specified period. The measurement will continue until the time entered or the total number of counts is reached.

  - **Armed by RPM Step** – If the Armed by RPM Step box is checked, a measurement is taken whenever it reaches a specified RPM step. It will begin when the Start RPM entered is reached and will stop the measurement when the Stop RPM entered is reached. Users can set the Start, Stop and Delta RPM.

  - **Armed by both** – Enables the measurement controlled by both time / rpm.

6.13.8 Order Tracking setup

From the toolbar, select **Setup** > **OT Setup**.
In this view, you can select **Function**, **Frequency**, and **Average**.

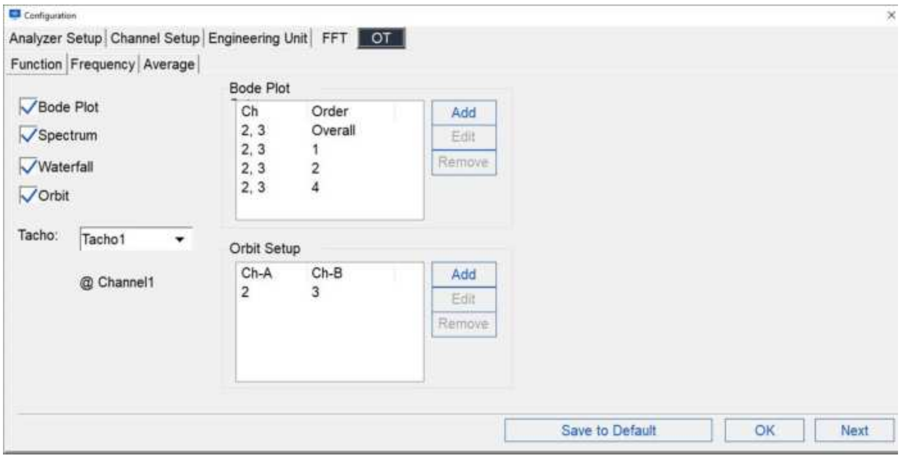


Figure 208 Order Tracking view – example

- **Function** – Select measurement functions from Bode Plot, Spectrum, Waterfall, and Orbit. And select a tachometer channel, which was selected previously in the Analyzer setup menu.
    - **Bode Plot Setup** – When Bode Plot function is selected, you may edit the trace properties from the sub-menu.
        - **Add** – Add Bode Plot.
        - **Edit** – Edit the selected Bode Plot.
        - **Remove** – Remove the selected Bode Plot.
    - **Add Bode Plot** – Select **Add Bode Plot** to open the **Bode Plot Setup** menu. Then select the channel(s) and type of Bode Plot you want to add to the measurements. A Bode Plot could be the overall trend, or an order component measured over the specified speed range. Note that an overall trace does not contain phase information.

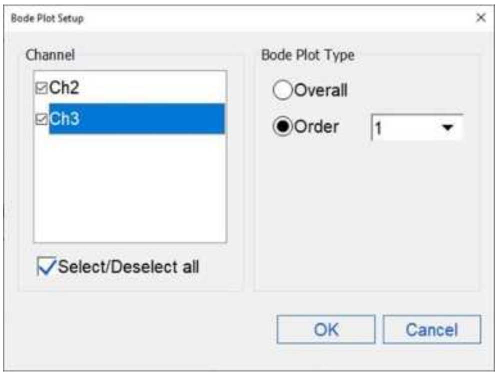


Figure 209 Bode Plot Setup – example

    - **Orbit Setup** – The three symbols to the right are **Add an orbit**, **Edit selected orbit** and **Remove orbit**.
        - **Add** – Add orbit.

- **Edit** – Edit the selected orbits.
- **Remove** – Remove the selected orbits.

Select **Add** or **Edit** to enter the Orbit setup. An orbit is the **XY** plot of signals from selected **channel A (X)** and **channel B (Y)**. To make the orbit and centreline plots show measured data in accordance with the actual geometry of the rotating machine, you need to specify the angular location of the sensors by entering the Angle A and Angle B.

Enter the Initial (Init) Gap value if shaft centerline is to be measured.

Figure 210 Angular location of the sensors – example

- **Frequency**

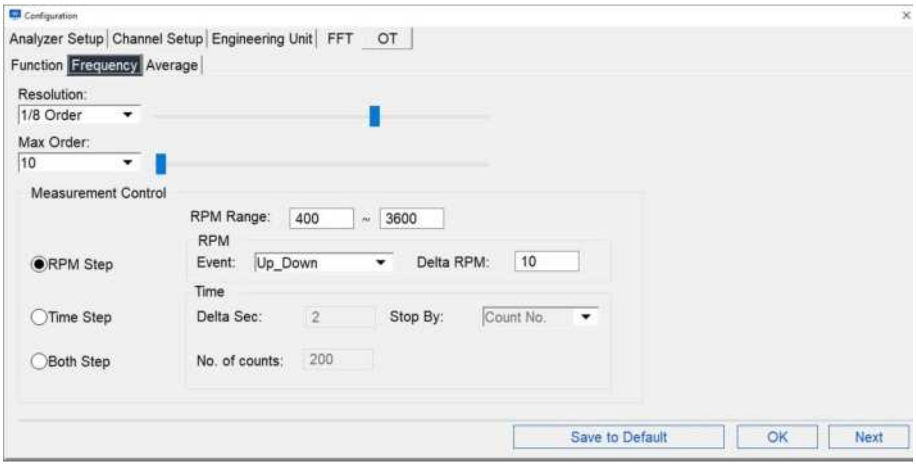


Figure 211 OT – Frequency view – example

- **Resolution** – Select resolution order from the dropdown menu options, **1/2**, **1/4**, **1/8** and **1/16 order**.

- **Max order** – Set the maximum order from the dropdown menu options, **5**,**10**, **20**, **50**, 100, **200**, **400** and **800** orders**.**

Note: The maximum sampling rate for order tracking is defined by **[Max Order] × [Max Speed]**. If the specified sampling rate exceeds your hardware limitation, a warning message will appear.

- **RPM Step** – If the Armed by RPM is checked, a measurement is taken whenever it reaches a specified RPM step. It will begin when the Start RPM entered is reached and stop when the Stop RPM entered is reached. You can set the **Start**, **Stop** and **Delta RPM**.

  Select **Event** as **Start Up**, **Coast down**, or **Up** and **Down**.
  If **Start Up** is selected, dBX Analyzer will take a measurement only when the speed is ascending.

- **Time Step** – Check to measure on specified period. The measurement will continue until the time entered or the total number of counts is reached.

- **Both** – Enables the measurement controlled by both time / RPM.

- **Average –** Set average of your Order Tracking measurement.

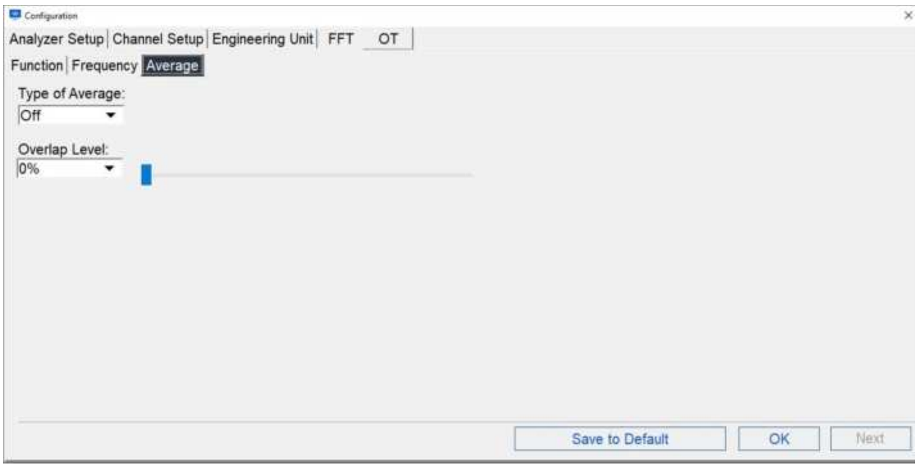


Figure 212 OT – Average view – example

- **Type of Average** – Select **Off**, **Linear**, **Exponential** or **Peak Hold**.

- **Overlap Level** – Use the dropdown or the slider to set the overlap percentage.

6.13.9 Recorder setup

Select **Recorder** or **Real-time + Recorder** mode from the main menu.
In the main toolbar, select **Setup** > **Recorder Setup**. There are two recorder measurement modes:

- **Recorder**– The measured raw signal is transferred to the hard drive continuously. During data recording, you can monitor basic Spectrum and Time Waveform.

- **Real time + Recorder** – Real-time measurements of FFT or Order Tracking run as in the real-time mode. The raw signal is saved to other raw data file(s).

In either of the above modes, use the following menu to set up the data recording. There are three sub-menus from the Recorder setup: **Frequency**, **Display** and **Auto Stop**.

- **Frequency**

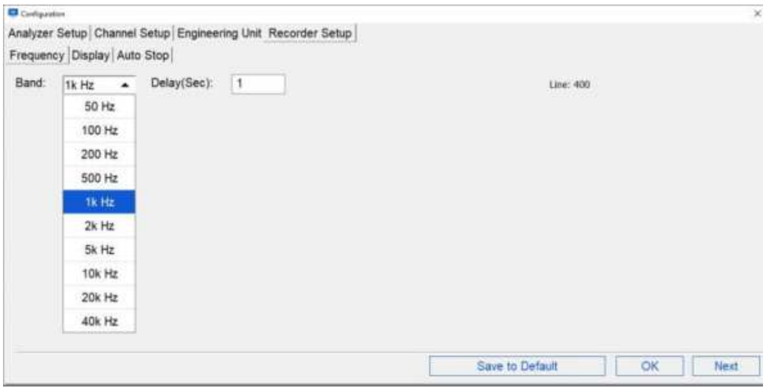


Figure 213 Recorder setup – Frequency view – example

  - **Band** – Select the Frequency Band.
  - **Delay** – Input the Delay time value (Sec).

- **Display**

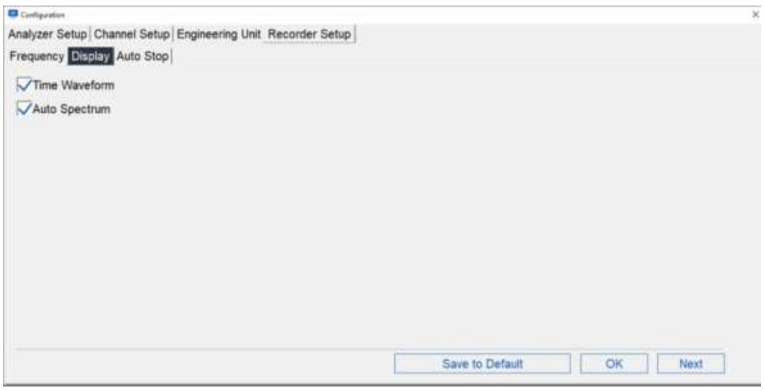


Figure 214 Recorder setup – Display view – example

Select **Time Waveform** and/or **Auto Spectrum** (400 lines) to be displayed during recording.

- **Auto Stop** – Set up the raw data recorder to automatically record start / stop settings, file size and where to record the data file. If all items are unchecked, you must stop the recording manually using the measure toolbar and press stop.

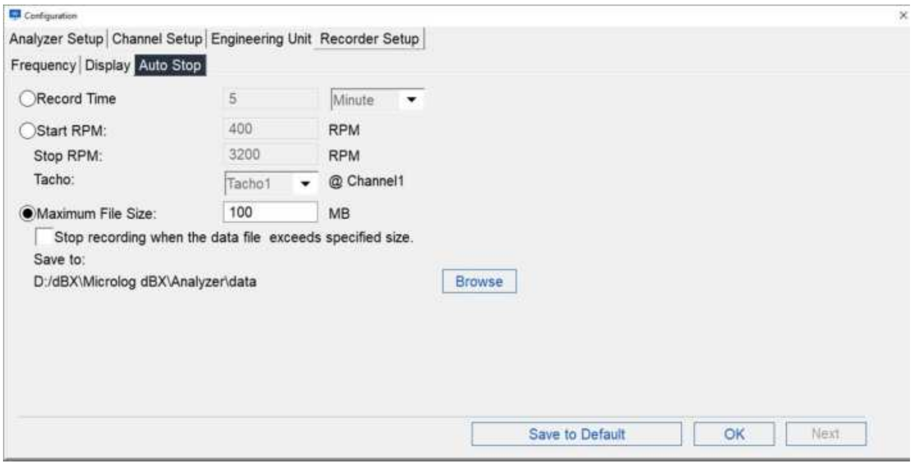


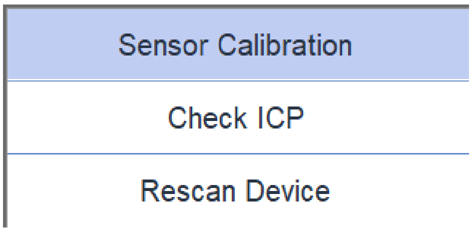
Figure 215 Recorder setup – Auto Stop view – example

  - **Record Time** – Enter the length of time to record.
  - **Start RPM** – Enter the starting RPM. It must be greater than 0.
  - **Stop RPM** – Enter the stop RPM.
  - **Tacho** – Select a Tacho channel.
  - **Maximum file size** – Enter the maximum file size you wish to record. This is helpful when concerned about large file sizes and storage space.
  - **Save to** – Click **Browse** and select path to save the recorder data files.

- **Real Time + Raw Data Recorder mode** – When measuring in Real time + Raw Data Recorder, a Recorder function is added to the Live data panel. As the real-time measurement starts, the continuous raw data is also taken simultaneously.

6.14 Tools menu

Use the Tools menu to perform **Sensor Calibration**, **Rescan Device** and **Check IPC**.



- **Sensor Calibration** – Calibrates and adjusts sensors' sensitivity with a calibrator. When a sensor requires calibration before measurement, this procedure is used.
  For example, an acoustic calibrator usually generates a 94 dB calibration signal at 1 kHz. Therefore, you can use this tool to calibrate its sensitivity very quickly.

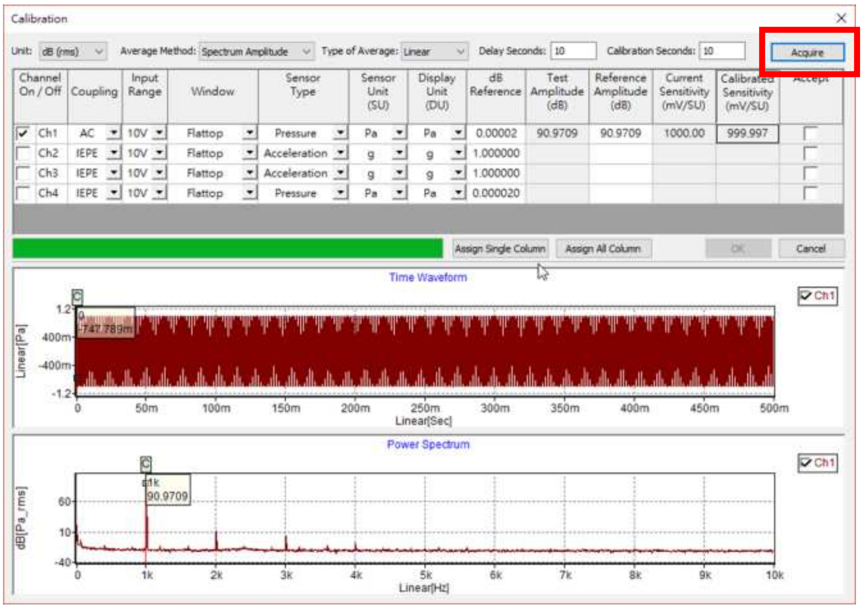


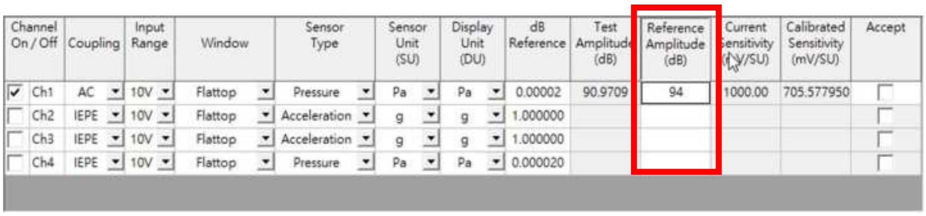
Figure 216 Sensor calibration view – example

1. Install the microphone to the 94 dB calibrator and connect it to Channel 1.

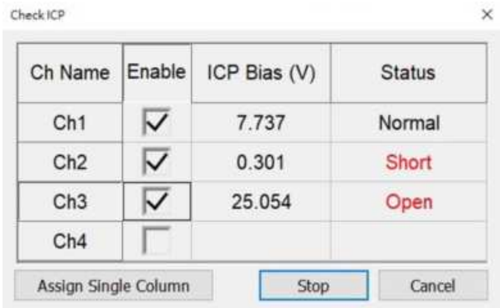
2. Turn the calibrator power on and select **Acquire** to start the measurement.

3. After data acquisition, the microphone sensitivity, in this example, is set as 1000 mV/Pa, and the measured dB level is 90.97 dB.

4. Input the **Reference Amplitude** to read 94. It will change the microphone sensitivity to 705.577 mV/Pa, which will make the measured level 94 dB.



- **Check ICP** – Check the ICP bias voltage when necessary. Select the channel for ICP bias checking and then select **Start** to perform the testing.



If the test result shows the status as **Open**, the ICP sensor is not connected, or the cable is faulty.
Note that the Open ICP bias should be in the range of **25+/-1 V** for normal conditions.
If the tested bias voltage is 0 V and the status **Shorted**, the cable connected to this channel is shorted.
If a normal ICP sensor and cable is connected to a channel, it will show the normal bias of that sensor and show the status as **Normal**.

- **Rescan Device** – Check the connection of the data acquisition hardware. Select **Rescan** device to check the connection of the data acquisition hardware. After conducting Rescan device, it will show the result as follows. A device called U404 will be found and checked. If no device was found, go to the **Utility application,** and perform a **reset DAQ** to restore the connection.

6.15 Version information

Tap the information icon  at the top right corner to display the version information.

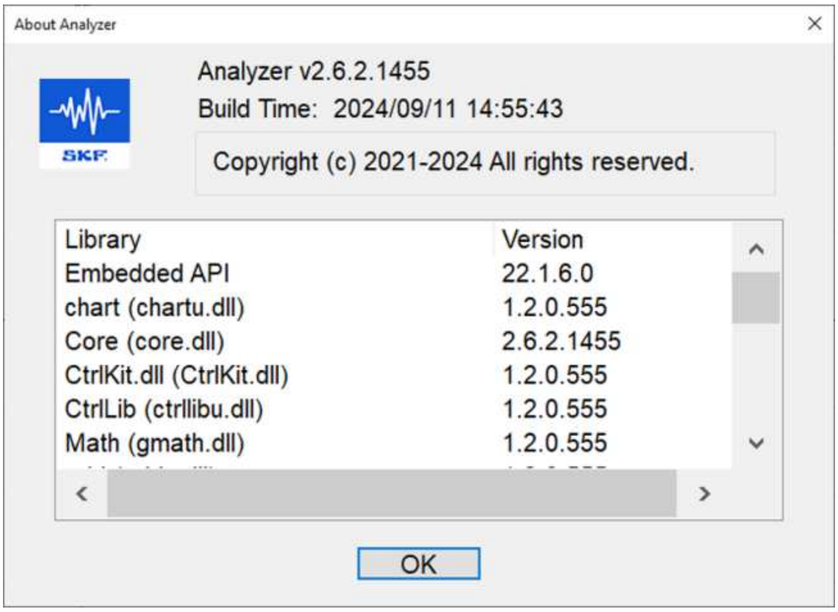


Figure 217 About window – example

7 dBX Documents

7.1 Introduction

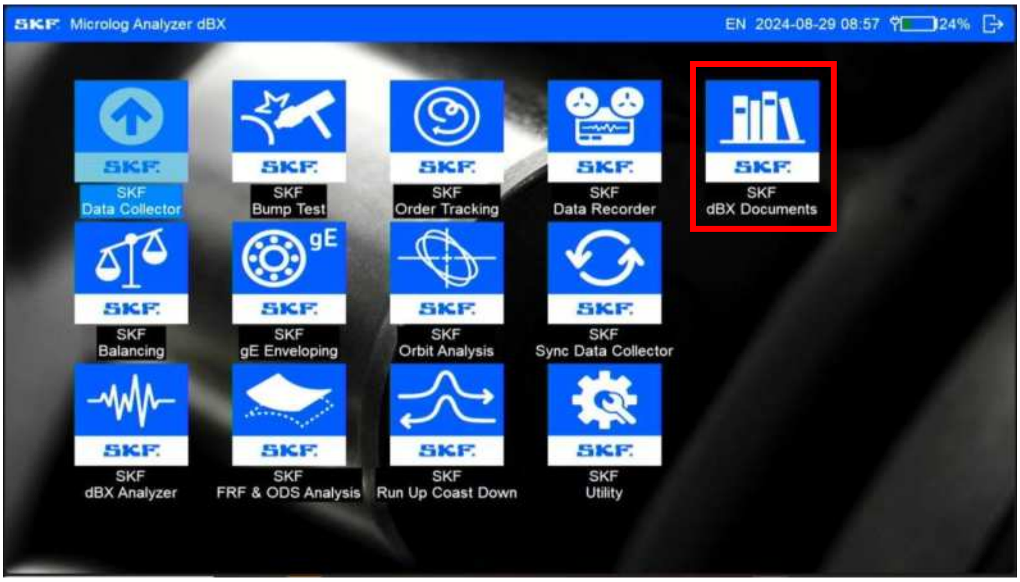


Figure 218 dBX Documents – example

Double tap the **dBX Documents** icon on the home screen to open the Documents application where Microlog dBX manuals and calibration files are stored.

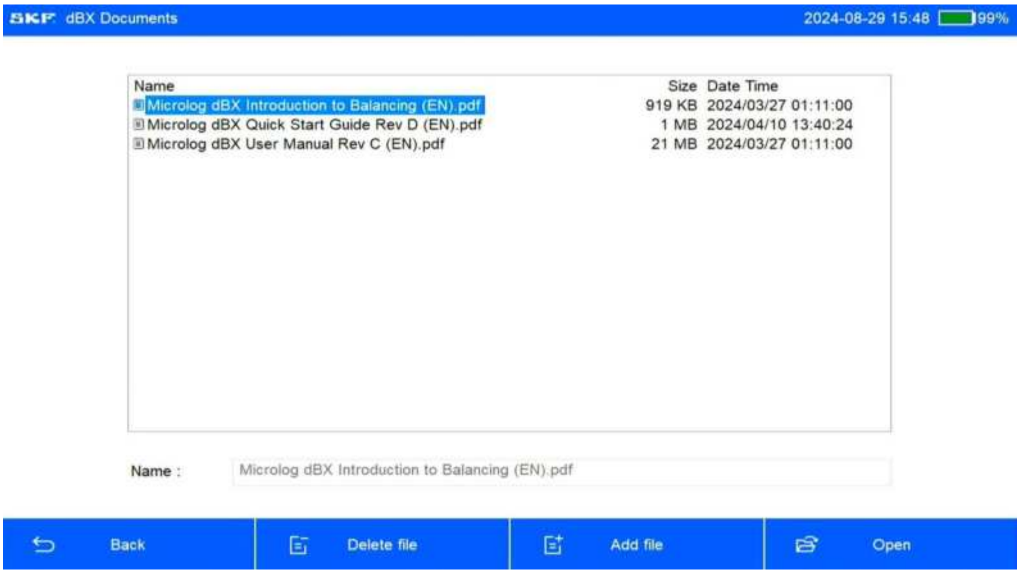


Figure 219 Home screen – dBX Documents icon

Open – To open a file, select it from the list and tap **Open**.

Add file – To add a file, tap **Add file** and select a file to add.

Delete file – To delete a file, select it from the list and tap **Delete file**.

Back – Tap to return to the previous view.

Appendix A Limited Warranty

SKF – Limited Warranty

If you have experienced an issue with your Microlog Analyzer dBX, to help expedite your concern, please contact your nearest SKF Technical Support representative and raise a support case.

Download the latest version from www.skf.com.

Appendix B Data Files

File Type	Name extension	Description
Project file	.npj	A project file contains both the setup parameters and the measured data.
Setup File	.nst	A setup file contains the setup parameters only. • Analysis Mode • Channel Setup • Engineering Unit • Recorder Setup • FFT Setup • Order Tracking Setup
Default File	default.nst	Default Setup File setting. When a new project is created, the default setting will be loaded to the new project.
Raw Data	.nrd & .nra	.nrd files contain the header information of a raw data file, while the .nra files contain the raw data itself. A complete set of raw data includes one nrd and one nra files.

Appendix C Unit Table Default Definitions

Acceleration	Display Unit	Int/Diff	Scale Factor
g	g	No	1
	mg	No	1000
	m/s^2	No	9.8067
	mm/s	int*1	9806.7
	in/s	int*1	386.091
	cm/s	int*1	980.67
	mil	int*2	386090
	µm (um)	int*2	9.8e6
m/s^2	g	No	0.101971
	mg	No	101.971
	m/s^2	No	1
	mm/s	int*1	1000
	in/s	int*1	39.37
	cm/s	int*1	100
	mil	int*2	39370
	µm (um)	int*2	1e6

Velocity	Display Unit	Int/Diff	Scale Factor
mm/s	mm/s	No	1
	cm/s	No	0.1
	in/s	No	0.03937
	m/s^2	diff*1	0.001
	g	diff*1	1.0197e-4
	µm (um)	int*1	1000
	mil	int*1	39.37
in/s	mm/s	No	25.4
	cm/s	No	2.54
	in/s	No	1
	m/s^2	diff*1	0.0254
	g	diff*1	2.59e-3
	µm (um)	int*1	2.54e4
	mil	int*1	1000
cm/s	mm/s	No	10
	in/s	No	0.3937
	cm/s	No	1
	m/s^2	diff*1	0.01
	g	diff*1	1.019711e-3
	µm (um)	int*1	1e4
	mil	int*1	393.7

Displacement	Display Unit	Int/Diff	Scale Factor
mil	mil	No	1
	µm (um)	No	25.4
	mm/s	diff*1	2.54e-2
	in/s	diff*1	0.001
	cm/s	diff*1	2.54e-3
	g	diff*2	2.59e-6
	m/s^2	diff*2	2.54e-5
	mg	diff*2	2.59e-3
µm	mil	No	3.937e-2
	µm (um)	No	1
	mm/s	diff*1	1e-3
	in/s	diff*1	3.937e-5
	cm/s	diff*1	1e-4
	g	diff*2	1.0197e-7
	m/s^2	diff*2	1e-6
	mg	diff*2	1.0197e-4

Force	Display Unit	Int/Diff	Scale Factor
lbf	lbf	No	1
	N	No	4.4482216
	g	No	453.59237
N	lbf	No	0.22480894
	N	No	1
	g	No	101.97162

Appendix D End User License Agreement

Terms & Conditions

By accessing or using this mobile application of the SKF Group an end user license agreement (“Agreement”) has been entered into between you and SKF Sverige AB (“SKF”), a company registered under the laws of Sweden, and you agree to be bound by the following terms and conditions. If a separate written end user license agreement for this mobile application has been agreed by you and an SKF entity, or if your employer has an agreement with an SKF entity regarding Microlog Analyzer dBX, such agreement will prevail over this Agreement. SKF may at any time update or change the Terms and Conditions without giving notice to you.

Limited license

Conditioned upon your compliance with the terms and conditions of this Agreement, SKF hereby grants to you a limited, revocable, non-exclusive, personal, non-sub licensable, non-transferable, non-assignable license to install and use the executable form of the mobile application solely for your personal use on a device (whether mobile device or other) owned or controlled by you. All rights not specifically granted by SKF under this Agreement are retained by SKF and withheld from you.

Restrictions of use

Except as expressly specified in this Agreement, you may not: (a) copy or modify the mobile application; (b) transfer, sublicense, lease, lend, rent or otherwise distribute the mobile application to any third party; or (c) use the mobile application in any unlawful manner, for any unlawful purpose, or in any manner inconsistent with this Agreement. You agree not to disassemble, decompile or otherwise reverse engineer any components of the App provided in object code or any other SKF products or services, in whole or in part, or permit or authorize a third party to do so, except to the extent such activities are expressly permitted by law notwithstanding this prohibition.

Warranty Disclaimer and Limitation of Liability

Although due care has been taken to assure the accuracy of the information in this mobile application, SKF provides this information "AS IS" and DISCLAIMS ALL WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. You acknowledge that your use of this mobile application is at your sole risk, that you assume full responsibility for all costs associated with use of this mobile application, and that SKF, or any of its group companies, shall not be liable for any direct, incidental, consequential, or indirect damages of any kind arising out of your access to, or use of the information or software made available in this mobile application.

Any warranties and representations in this mobile application for SKF products or services that you purchase or use will be subject to the agreed upon terms and conditions in the specific written contract for such product or service.

Further, for non-SKF sources that are referenced in our mobile application or where a hyperlink appears, SKF makes no warranties concerning the accuracy or reliability of the information in these sources and assumes no responsibility for material created or published by third parties contained therein. In addition, SKF does not warrant that this mobile application or these other linked sources are free from viruses or other harmful elements.

Copyright

Copyright in this mobile application and copyright of the information and software made available in this mobile application rest with SKF or its licensors. The licensed open source software and applicable license terms including credits and applicable terms for usage by you can be found on www.skf.com All rights are reserved. All licensed material will reference the licensor that has granted SKF the right to use the material. The information and software made available in this mobile application may not be reproduced, duplicated, copied, transferred, distributed, stored, modified, downloaded or otherwise exploited for any commercial use without the prior written approval of SKF. Under no circumstances may this information or software be supplied to third parties.

Trademarks and Patents

All trademarks, brand names, photos and corporate logos displayed in the mobile application are the property of SKF or its licensors, and may not be used in any way without prior written approval by SKF. All licensed trademarks published in this mobile application reference the licensor that has granted SKF the right to use the trademark. Access to this mobile application does not grant to the user any license under any patents owned by or licensed to SKF.

Changes and termination

SKF reserves the right to make changes or additions to this mobile application at any time. This Agreement will terminate automatically upon the earlier of: (a) your failure to comply with any term of this Agreement; (b) you deleting the mobile applications(s) from your devices; and (c) if you are using the mobile application(s) in connection with a paid service, the end of the time period specified at time of purchase. In addition, SKF may terminate this Agreement at any time, for any reason or no reason. If this Agreement terminates, you must stop using the mobile application(s) and delete them from your devices.

Applicable law

The laws of Sweden shall apply to this Agreement.

Export controls

You shall not export or re-export, directly or indirectly, the mobile application, or use the mobile application to export or re-export any information, without complying with export controls imposed by any applicable country (or any future export control legislation) and the regulations promulgated thereunder.

Privacy Notice

SKF will handle any personal data obtained from you in accordance with the [insert app name] Privacy Notice. Please read this Privacy Notice carefully before using this mobile application.

Appendix E Federal Communications Commission Interface Statement

This equipment has been tested and found to comply with the limits for a Class B digital device, pursuant to part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference in a residential installation. This equipment generates, uses and can radiate radio frequency energy and, if not installed and used in accordance with the instructions, may cause harmful interference to radio communications. However, there is no guarantee that interference will not occur in a particular installation. If this equipment does cause harmful interference to radio or television reception, which can be determined by turning the equipment off and on, the user is encouraged to try to correct the interference by one or more of the following measures:

-Reorient or relocate the receiving antenna.

-Increase the separation between the equipment and receiver.

-Connect the equipment into an outlet on a circuit different from that to which the receiver is connected.

-Consult the dealer or an experienced radio/ TV technician for help.

CAUTION:

Any changes or modifications not expressly approved by the grantee of this device could void the user's authority to operate the equipment.
This device complies with Part 15 of the FCC Rules. Operation is subject to the following two conditions: (1) this device may not cause harmful interference, and (2) this device must accept any interference received, including interference that may cause undesired operation.

RF exposure warning

The equipment complies with FCC RF exposure limits set forth for an uncontrolled environment.
The equipment must not be co-located or operating in conjunction with any other antenna or transmitter.

Appendix F Radio Frequency Identification Tag Standards

The following RF tags are supported by the built-in RF identification using near-field communication (NFC) located on rear of SKF Microlog Analyzer dBX.

Radio Frequency Identification	
RF Transmit Frequency	13.56MHz
Supported Transponder	ISO 15693(R/W) ISO 18092(R/W) ISO 14443-A(R/W) ISO 14443-B (Read UID only) (Depend on antenna type and antenna matching)
Antenna impedance	Loop Antenna, 50 ohms
Operating Temperature	-25°C to + 85°C
Storage Temperature	5 ~ 97% non-condensing