Polytec

Software Manual

Polytec Scanning Vibrometer Software 7.3

PSV 200

PSV 300

MSV 300

Trademarks

Brand and product names mentioned in this manual could be trademarks or registered trademarks of their respective companies or organizations.

Contents

1 First Steps

Polytec's Scanning Vibrometer measures the two-dimensional distribution of vibrations. The software has the following functions in the measurement system:

- controlling the hardware
- defining scan points
- setting parameters for data acquisition
- generating excitation signals (as an option)
- acquiring digital data
- saving data
- displaying data and evaluating it.

Read the following sections to familiarize yourself with the basics of how to work with the software.

1.1 Integration of the Hardware

Caution!

Do not change any address or interrupt settings! This could prevent the software from running properly.

Using the software you can control two Polytec vibrometers and as an option, an external function generator as well. Connect up the devices as described in your hardware manual.

IEEE-488/GPIB If you control the devices via IEEE-488/GPIB, then we recommend you set the following addresses:

- Vibrometer controller 1: IEEE-488/GPIB address 5
- Vibrometer controller 2: IEEE-488/GPIB address 4
- Function generator: IEEE-488/GPIB address 10.

RS-232 If you control vibrometers via RS-232, then the software recognizes the controller connected to the lower COM port as Vibrometer Controller 1.

1.2 Starting the Software

To start the software, proceed as follows:

1. Check that the hardlock is plugged into the parallel interface of your workstation.

2. Switch the workstation on.

3. After a short while you will be asked to log in. See your operating system manual on this. In the condition the workstation is supplied in, you have to log in as administrator. A password has not been set prior to delivery.

4. Double-click the icon 

on the desktop or select Start > PSV. The application window appears. You may receive an error message and information on it (refer to section 1.3).

You can now work with the software.

1.3 Acquisition Mode and Presentation Mode

In the software you work in two different modes: acquisition mode and presentation mode. You can use some functions in both modes. Acquisition mode is not available in the desktop version.

Presentation mode

To go into presentation mode, click  or select View > Presentation.

Data acquisition

To go into acquisition mode, click  or select View > Acquisition.

When changing into acquisition mode, the software searches the interfaces of the workstation for external devices and initializes them. You may receive an error message and information on it. You can switch off this error message as described in chapter 2.

Figure 1.1: Error message when changing into acquisition mode

5. If you want to check the connection between the devices and correct it if necessary, select Set Up > Options and open the page Devices.

1.4 Windows in the Software

1.4.1 Application Window

After starting the software, the application window appears. At the upper edge you will find the title bar, menu bar and tool bar, at the bottom will be the status bar. In the menu View, you can show (hide) the tool bar and the status bar.

Initially the application window is empty. By clicking  you change over to data acquisition. There you will see the video window and an analyzer. (Both windows are described in the following sections in detail.)

By clicking  you can display further analyzers.

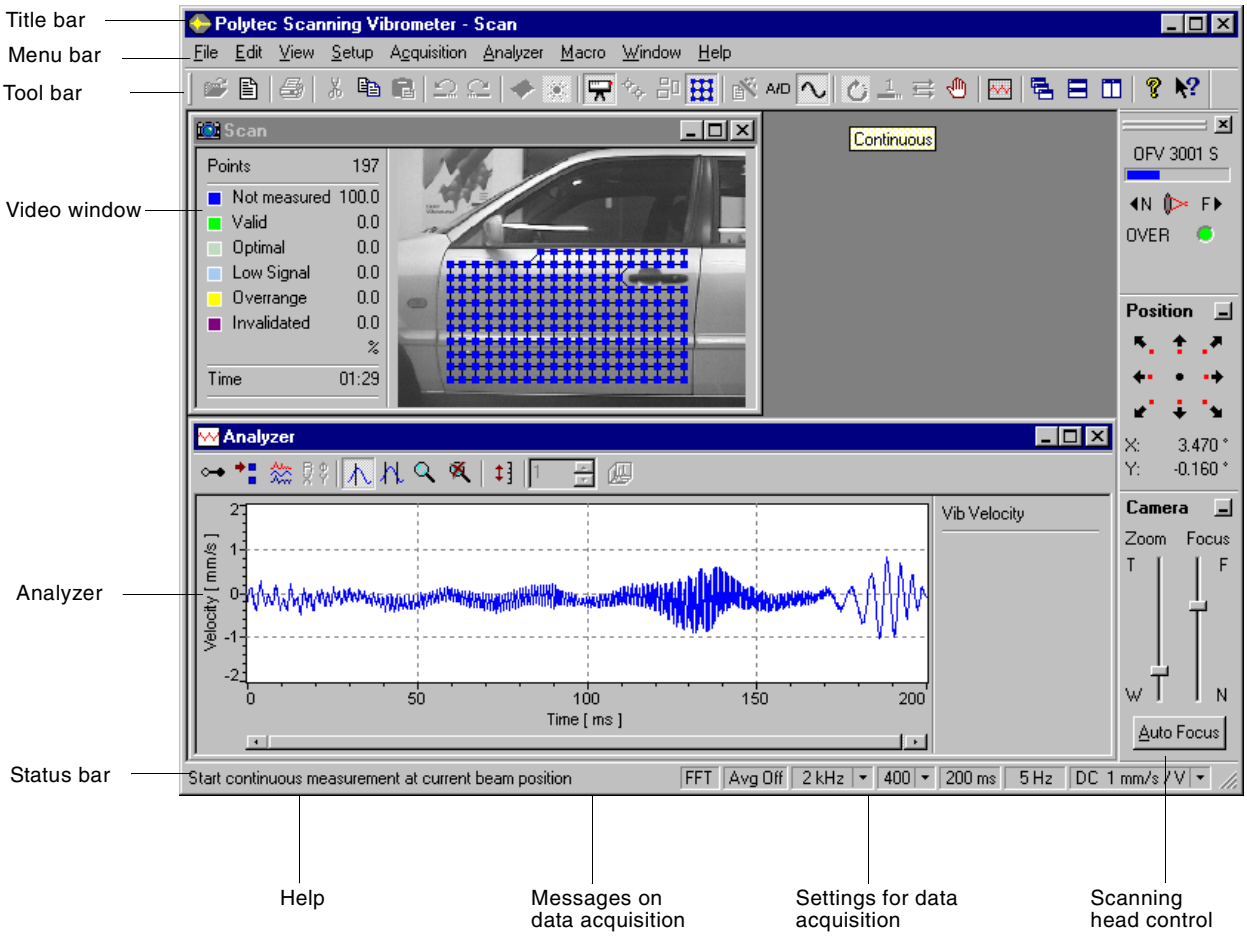


Figure 1.2: Example of the application window in acquisition mode

In the application window you can display and evaluate data. To do so, you can use various windows:

- in acquisition mode: a video window, any number of analyzers
- in presentation mode: any number of presentation windows and analyzers.

If you restart the software, then the arrangement and signal selection of the most recent windows will be restored.

Menu bar Every menu contains a group of commands for working with the software. Some menus are only visible if an analyzer or a presentation window is open.

Tool bar By clicking the icons, you can execute some commands quickly which you would otherwise have to select in the menus.

Status bar On the right, the status bar shows several settings for data acquisition. On the left, in certain situations, you will see a short help text. Point the mouse at one of the settings on the right, for example, and leave the cursor there for a moment. Then you will see on the left which parameter it is for. For more information on how to get help in the software, see section 1.6.

When making a measurement, messages on data acquisition can appear in the middle of the status bar. See section 5.3 on this.

Scanning head control In acquisition mode you can show (hide) the scanning head control in the application window. To do so, click  or select View > Scanning Head. You set the optics using the scanning head control. See section 3.1 on this.

1.4.2 Video Window

The video window displays the image of the video camera in the scanning head live.

Open The video window appears when you change to acquisition mode.

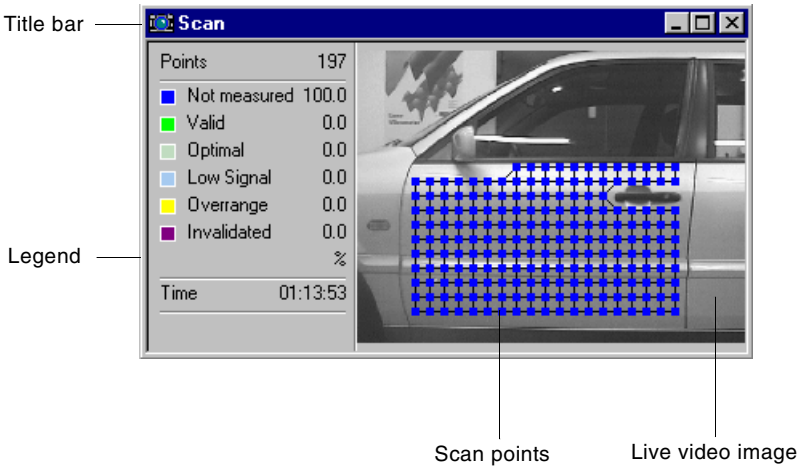


Figure 1.3: Example of the video window

Live video image In the live video image, you control the position of the laser beam, define scan points and follow the progress of a scan. You adjust the settings of the live video image (contrast, brightness, color) as described in section 3.3.

Scan points

If scan points are defined, you can show (hide) them in the video window.

To do so, click  or select View > Scan Points. You see the scan points as squares in different colors. These colors show the status of the scan points.

Legend

On the left you see a legend for the status of the scan points. The legend also shows the number of defined scan points and the estimated time needed for the scan.

1.4.3 Analyzers

Analyzers show measurement data and evaluated data in x-y diagrams. You can follow current measurements in analyzers or display measurement data from a file in them. In the evaluation an analyzer can also be part of a presentation window. See section 1.4.4 on this.

Open

An analyzer appears if you

- go into acquisition mode
- start a measurement
- open a single point measurement.

If an analyzer is open, the menu bar of the application window is extended by menus for working with analyzers. To open further analyzers, click  or choose Window > New Analyzer.

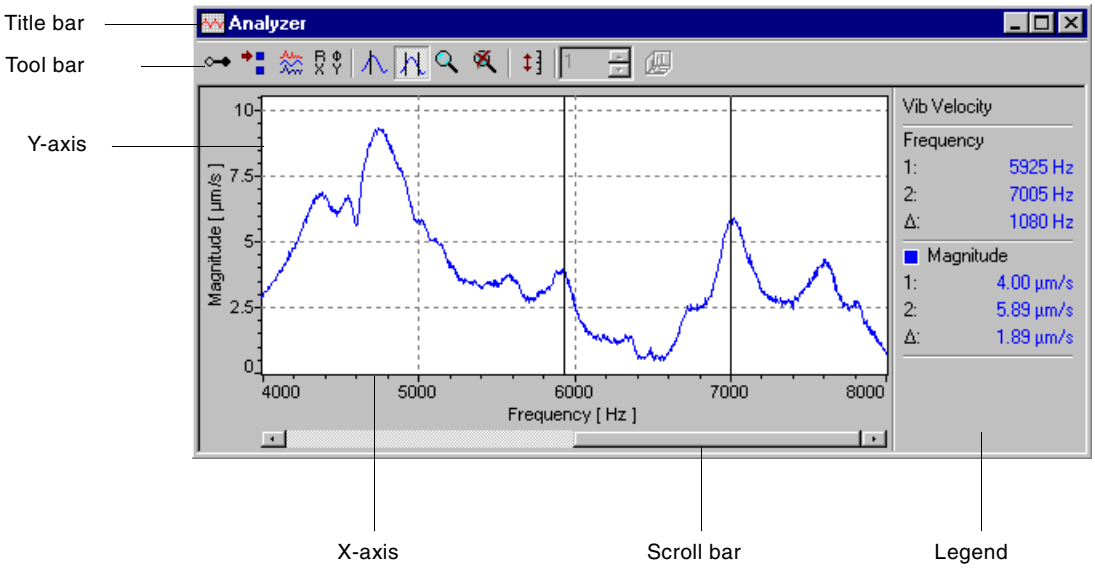


Figure 1.4: Example of an analyzer

Title bar

If an analyzer has the function of a measurement window, in the title bar you will see the name Analyzer. If you have already saved the measurement or have opened a measurement, you will see the file name in the title bar.

Tool bar You can show (hide) the tool bar by selecting Analyzer > Tool Bar. By clicking the icons you can execute some commands from the menu Analyzer quickly.

Diagram You see the data graphically in x-y diagrams. You can set up the look of the diagrams and the graphs yourself.

Legend On the right you can show (hide) a legend for the diagrams. To do so, select Analyzer > Legend.

Scroll bar You can use the scroll bar to scroll if you have zoomed the x-axis.

1.4.4 Presentation Window

In presentation windows you analyze scans in 2D and 3D diagrams.

Open A presentation window appears when you open a scan. The menu bar of the application window is then extended by menus for working with the presentation window. You can then open further presentation windows by selecting Window > New Window.

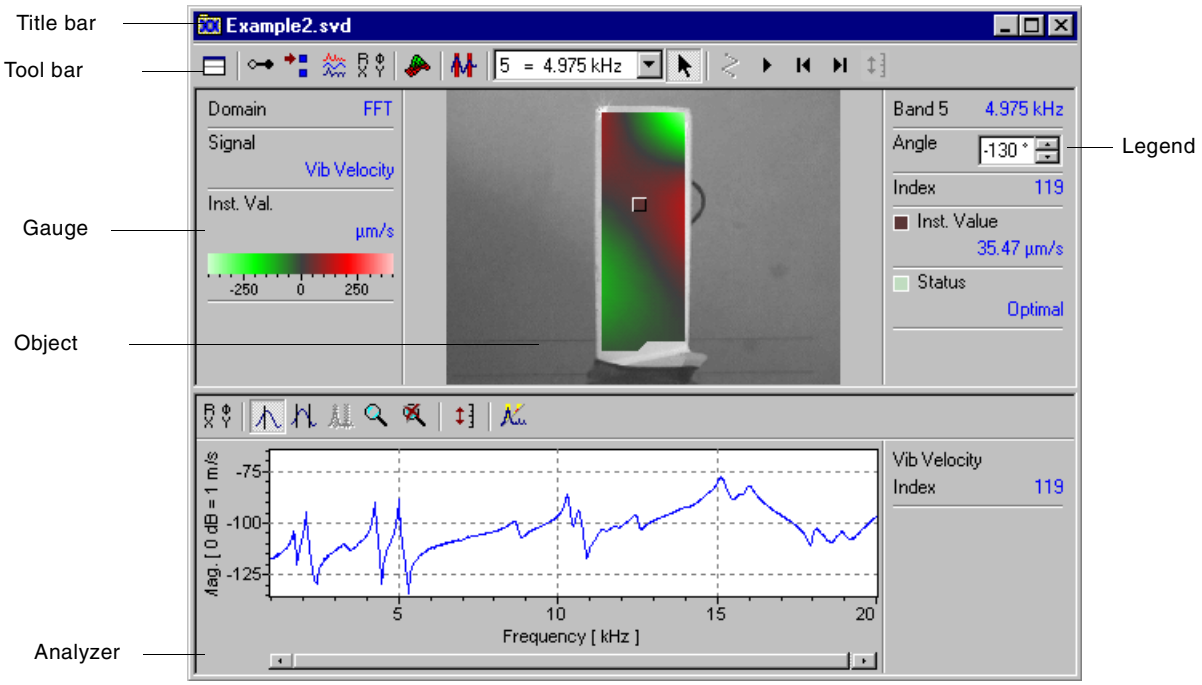


Figure 1.5: Example of a presentation window in the Single Scan Point view

Tool bar You can show (hide) the tool bar by selecting Analysis > Tool Bar. By clicking the icons you can execute some commands from the menus Setup, Presentation and Animation quickly.

View You can display the presentation window in four different views. To do so, click  and select the view required in the popup menu. You can also select Presentation > View.

Object You see the object in all views. You can present the object in five different ways. To do so, click  and select the view style required in the popup menu. You can also select Presentation > View Style.

Analyzer In the views Single Scan Point, Average Spectrum and Profile you see an analyzer at the bottom of the presentation window. See section 1.4.3 on this.

Gauge On the left you can show (hide) a gauge for color-coding. To do so, select Presentation > Gauge.

Legend On the right you can show (hide) a legend for the data displayed. To do so, select Presentation > Legend.

1.4.5 Arranging Windows

You can arrange all open windows in the application window using the commands in the menu Window.

New window To open further analyzers or presentation windows, click  or choose Window > New Analyzer.

Cascade You can arrange all windows over each other so that only the title bar and the y-axis name are visible respectively. To do so, click  or select Window > Cascade.

Tiling horizontally and vertically You can display all open windows so that they are completely visible and maximum size. To do so, click  or  . You can also select Window > Tile Horizontally or Window > Tile Vertically.

Close all To close all open windows, select Window > Close All.

Window list At the bottom of the menu Window you will find a list of open windows. To activate a window – i.e. move it to the foreground – click it in the list.

1.5 Printing

You can print out analyzers and presentation windows. The title of the window as well as the date and time are automatically printed as a header.

Print setup To set the paper size, feed and orientation, select File > Print Setup. The dialog Print Setup appears. Here you can select the current printer and set up the page.

Preview To see a print preview, select File > Print Preview.

Start To print out the active window, click  or select File > Print. The dialog Print appears. Here you can select the printer and set up further print properties. To start printing, then in the dialog Print click OK.

1.6 How to Get Help

You can get help in the software as online help and through short explanations of the commands. You can also display the version of the software and the software options which are installed.

Online help

To open the online help, click  or select Help > Contents.

Context-sensitive help

For dialogs and windows you can get context-sensitive online help. You have several possibilities to do this:

1. Activate the window or dialog you need help with. To do so, click it.
2. Press the key F1. The corresponding page of the online help appears.

or

When a dialog is open, then click Help in this dialog. The corresponding page of the online help appears.

or

1. When a dialog is open, click  at the top right in the dialog. Next to the cursor a question mark will appear.
2. Click an active element in the dialog – for example, an icon or a list.

Explanation of commands

For icons and commands you can also get explanations without opening the online help. To do so, point the mouse at the command or the icon and leave the cursor there for a moment. The explanation appears on the left in the status bar. For icons you will also see a brief description on a yellow background next to the cursor.

Software version and options

To display the version of the software, select Help > About PSV. The dialog About Polytec Scanning Vibrometer appears. To display the installed software options, then in the dialog click Options.

You can also display the version and serial number of the software as well as the memory and available hard disk space. To do so, click System info.

2 Setting up the Software

Before making a measurement, you have to set up the software for data acquisition. To do this, proceed as follows:

1. Go into acquisition mode. To do so, click  or select View > Acquisition.

☞ You can also set some options in presentation mode.

2. Select Setup > Preferences. The dialog Preferences appears.

3. The dialog contains several pages each with a group of parameters. To display a certain page, click the name of the group.

4. Set the parameters on all pages. You will find information on this in section 2.1, section 2.2 and section 2.3.

5. When you have set all parameters, click OK.

2.1 Devices

On the page Devices you set which measurement system and which external devices you want to control with the software.

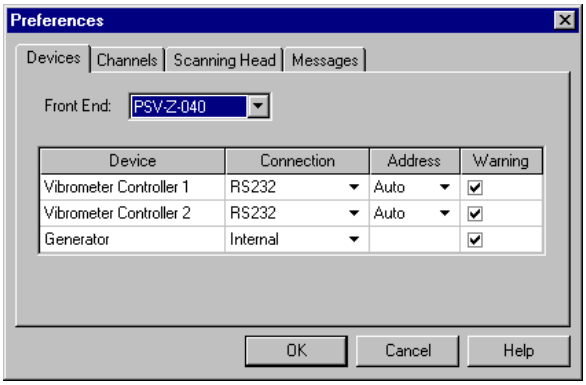


Figure 2.1: Page Devices

Front End: This list is only active for certain hardware configurations. Here you select which junction box your workstation is connected to.

Connection: Here you can select which interfaces should be searched for external devices.

For the vibrometer controller you can either have the available interfaces automatically searched for devices (Auto) or you can state where the devices are connected (RS232 or GPIB). You will speed up the start of the application with a fixed standard.

If you are using a generator, you can also state whether the device is integrated into the workstation (internal) or whether it is an externally connected device.

Address: If you have given a fixed connection, then select the address here or carry out an automatic search for it.

Warning: If you change into acquisition mode or have changed any settings, then the software will search the workstation's interfaces for external devices and initialize them. Here you mark the devices a warning message is to be issued for if the software can not find them.

☞ These warnings are not shown if under Set Up > Options on the page Messages you have selected Errors or Show None (refer also to section 2.4).

2.2 Channels

On the page Channels you set which channels you want to use to control Polytec vibrometers. Apart from that, you can allocate a name and an abbreviation for each channel.

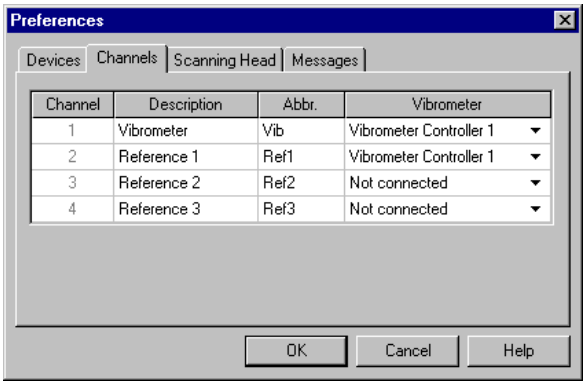


Figure 2.2: Page Channels

Channel: The number of channels displayed depends on which data acquisition board you are using and which junction box has been selected on the page Devices (refer to section 2.1). However, the number of channels is only updated if you close the dialog Preferences by clicking OK and then reopen it.

Description: You can enter a meaningful name for every channel here.

Abbreviation: You can enter a meaningful abbreviation for every channel here.

Vibrometer: In this column one channel is only active if a Polytec vibrometer is connected, the vibrometer had already been switched on before changing into acquisition mode and on the page Devices the correct connection has been set. Here you select which vibrometer provides the signal for this channel. You can use the software to control the connected vibrometers as described in section 5.5.6. For certain hardware configurations and measurement modes, channels are already permanently connected.

In table 2.1 and table 2.2 you will find the allocation of the BNC jacks on the different junction boxes to the digital channels.

Table 2.1: Allocation of digital and analog channels

Channel	Junction box				
	PSV-Z-040-H	PSV-Z-040-F, -U	MSV-Z-040	VDD-Z-010	VDD-Z-011
1	VELO	VELO	VELO	vibrometer signal[1]	vibrometer signal[1]
2	REF 1	REF	REF	REF IN	REF 1
3	REF 2	-	-	-	-
4	REF 3	-	-	-	-

[1] Only VDD 650/660: The vibrometer signal is not fed in through the junction box, it is only available directly in the software.

Table 2.2: Allocation of digital and analog channels

Channel	Junction box				
	VIB-Z-010	VIB-Z-012	VIB-Z-014	VIB-Z-016	VIB-Z-017
1	SIG IN	VELO	VELO	VELO	VELO
2	REF IN	REF 1	REF 1	REF 1	REF 1
3	-	-	REF 2	REF 2	REF 2
4	-	-	REF 3	REF 3	REF 3

With the junction boxes VIB-Z-012 and VIB-Z-016 you can select ICP for the reference channels and also switch the differential input for all channels on or off. You will find additional information on this in section 6.2.2 below the keyword ICP and in your hardware manual.

2.3 Scanning Head

On the page Scanning Head you set up the optics of the measurement system.

Figure 2.3: Page Scanning Head

Scanning Head: Select the scanning head of your measurement system here. Only PSV 200: Select the OFV-055F, if your scanning head is equipped with a color camera.

Pan-Tilt Control: This line only appears if you have selected the scanning head OFV-056 above. Tick the box if you want to control the optional pan-tilt stage using the software.

Fast Scan Axis: Here you select how the laser beam is to be moved when scanning. If the scan points are to be hit row by row, select the x-axis. If the scan points are to be hit column by column, select the y-axis. In the latter case the scanning time is shorter, as the scanner mirror for the y-axis is smaller and thus has a shorter settling time.

Angle Correction: The vibrometer measures vibration parallel to the laser beam. If the laser beam is at an oblique angle to the measurement surface, then an angle error occurs. Here you select whether the software is to correct the angle error for the respective axis. The angle correction only provides correct measurement values if you are measuring a flat surface at a right angle to the scanning head. For curved surface you should deactivate the angle correction.

Flip Video: The box is only automatically activated for certain video boards. Here you can mirror the live video image on the x- or y-axis and this if necessary set it up properly again. Normally the boxes appropriate for the sensor head have already been selected. However you can change the setting as appropriate, if, for example, you redirect the laser beam with external mirrors and thus obtain a flipped image or if the scanning head has been mounted in such a way that it generates a flipped image.

2.4 Messages

On the page Messages you can select which system messages you would like to have displayed:

- If you would like to see all messages (notes, errors and warnings), then select Show All.
- If you would like to be informed of warnings and errors, then select Warnings.
- If you would only like to see error messages, then select Errors.
- If you do not want to receive any system messages at all, then select Show None.

☞ Message windows which contain queries or advise you of faulty input will appear in any case.

Figure 2.4: Page Messages

3 Set the Optics

Depending on the model of your measurement system, you can control certain optics functions using the software. See section 3.1 on this.

To define scan points on the live video image and to move the laser beam using the mouse, you have to align the coordinates on the live video image with the measurement plane. See section 3.2 on this.

You adjust the settings of the live video image (contrast, brightness, color) as described in section 3.3.

3.1 Controlling the Hardware

For the PSV 300 you can use the software to control all the optics and the optional pan-tilt stage.

For the MSV 300 you can only use the software to position the laser beam. You set the other functions using the hardware. See your hardware manual on this.

For the PSV 200 you can only use the software to focus and position the laser beam. You control the video camera and the optional pan-tilt stage with the video box. See your PSV 200 manual on this.

To control the optics, proceed as follows:

1. Go into acquisition mode. To do so, click  or select View > Acquisition.
2. If you do not see the scanning head control in the application window, then click  to display it. You can also select View > Scanning Head.
3. Only MSV 300: From the list Lens, select the magnification which your microscope is set at.

☞ If you can not see the list Lens in the scanning head control, then first of all set up the software for your measurement system. See chapter 2 on this.

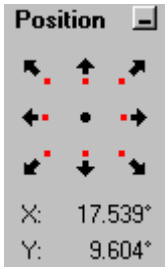
4. If necessary, maximize the video window. To do so, in the title bar of the video window on the right, click  .
5. If necessary, adjust the focus of the live video image. See section 3.3 on this.

6. Using the icons in the scanning head control you can now work as described below.

Focus the laser beam

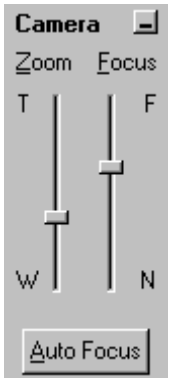
You focus the laser beam with the icons N (close-up) and F (infinity). Above the icons you will see the optical signal level as a bar. This makes focusing easier.

Position the laser beam



You move the laser beam with the icons Position. To do so, click the icon for the direction required. If you click •, then the laser beam is positioned so that it is emitted at a right angle to the front panel of the scanning head (position 0; 0).
Below the icons you can see the current position of the laser beam.

Zoom the video camera

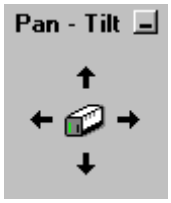


You zoom the video camera with the slider Zoom. Using the mouse you can roughly set the slider. If the slider is active, you can fine-tune it with the arrow keys ↑ and ↓. (To activate the slider, click it.)

Focus the video camera

You focus the video camera with the slider Focus. Using the mouse you can roughly set the slider. If the slider is active, you can fine-tune it with the arrow keys ↑ and ↓. (To activate the slider, click it.)
You can also autofocus. To do so, click Auto Focus. Autofocusing can take several seconds during which the software is inactive.

Pan-tilt stage (as an option)



You control the motorized pan-tilt stage with the icons Pan-Tilt. If you can not see the icons in the scanning head control, then first of all activate the pan-tilt stage for the software. See chapter 2 on this.

To pan the scanning head, click ← or →. To tilt the scanning head, click ↑ or ↓.

3.2 Aligning Coordinates

To define scan points on the live video image and to move the laser beam using the mouse, you have to align the coordinates on the live video image to the measurement plane. As you are aligning two-dimensional coordinate systems, you have to align at least two points with different x- and y-coordinates.

When?

You have to realign when you

- are working at a different stand-off distance
- have zoomed the video camera.

Tips

The following tips make it easier to align:

- The more points you align, the more precisely the software positions the laser beam.

- Distribute the alignment points evenly over the whole measurement surface. If, for example, you want to scan a rectangular surface, it would make sense to chose four alignment points near the corners.
- Align those points which you want to position precisely when scanning.

Alignment

To align the coordinates, proceed as follows:

1. Click  or select Setup > Align Coordinates. The software maximizes the video window and displays the scanning head control.
2. Move the laser beam with the icons Position in the scanning head control to an alignment point on the object, refer also to section 3.1.
3. In the live video image, point the mouse precisely at the laser beam and click. You can now see a target there.
4. If the laser beam is not precisely in the middle of the target, repeat step 3.
5. Repeat step 2 to 4 for at least one other alignment point.
6. When you have aligned all points, then click  again or select Setup > Align Coordinates.

Test

Now test the alignment as follows:

7. If the scan points are shown in the live video image, click  to hide them.
8. Point at the live video image and click a point on the measurement plane. The laser beam moves to this point on the object.
9. Check if the position of the laser beam corresponds to that of the point clicked. If this is not the case, repeat the alignment.

☞ The live video image shows a two-dimensional image of the three-dimensional object. Therefore, the laser beam does not hit points exactly which are not on the measurement plane.

10. Repeat steps 8 and 9 for a few more points.

3.3 Settings of the Live Video Image

You adjust the settings of the live video image in the dialog Display Properties. To open the dialog, double-click the live video image. You can also activate the video window (to do so, click it) and then select Edit > Properties.

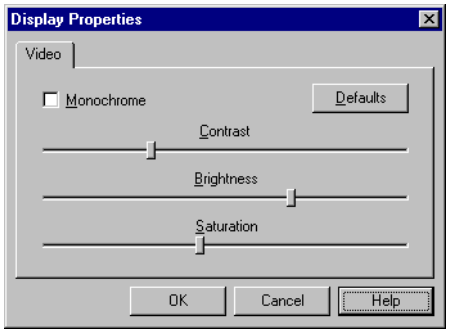


Figure 3.1: Dialog Display Properties

Monochrome: Here you can change color images to black and white. This is recommended for measurements with the optional close-up attachment OFV-056-C, as it is equipped with a red filter.

Standard: Click here to load the default settings for the live video image.

Contrast: Here you set the contrast of the live video image.

Brightness: Here you set the brightness of the live video image.

Saturation: For color images you set the color saturation here.

4 Defining Scan Points (APS)

Before a scan you first of all have to define the scan points. To do this, you draw geometric figures on the live video image as you would in graphics software. For this purpose, the software offers a standard mode and two further modes as an option.

Standard mode

In standard mode you can draw rectangles, ellipses and polygons. The scan points in all figures are on a global right-angled grid.

Professional mode (as an option)

In professional mode, apart from rectangles, ellipses and polygons, you can also draw lines. You can set up the grid for every figure individually and it can also be polar or hexagonal. Professional mode is only available if you have the option APS Professional.

Point mode (as an option)

In point mode you do not draw figures but set and edit individual scan points and their connections. You can take over scan point definitions from the other modes and fine-tune them. Point mode is only available if you have the option APS Professional.

Define

To define scan points, proceed as follows:

1. Go into acquisition mode. To do so, click  .
2. If necessary, align the coordinates of the live video image to the measurement plane. See section 3.2 on this.

3. In the tool bar of the application window, click  or select Setup > Define Scan Points. The video window is maximized and a graphics tool bar appears. Areas of the live video image can be crosshatched. These areas are outside the range you can scan.

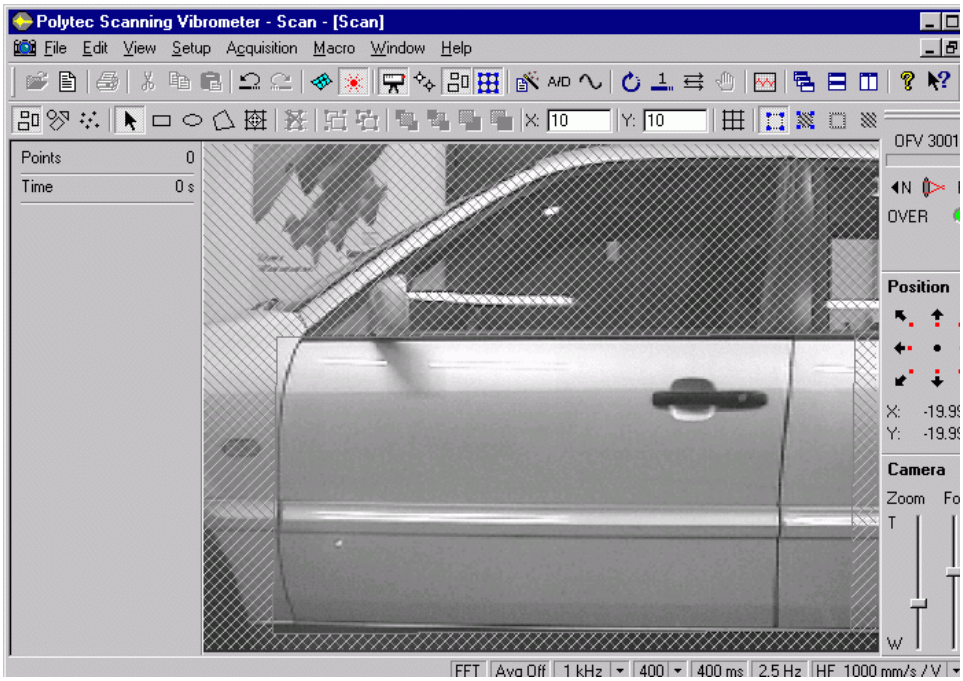


Figure 4.1: Definition of scan points

☞ The possible scanning area depends on the scanning head, zoom of the camera and correct alignment.

4. Go into the required mode. To do so, in the graphics tool bar, click  ,  or  . The icons  and  are only active if you have the option APS Professional.

5. Define scan points as described in section 4.1, section 4.2, section 4.3 and section 4.4. In the legend of the live video image you can see how many scan points you have defined. Below that you see the estimated time needed for the scan. The estimation is based on the time for data acquisition at all scan points and the settling time of the scanner mirrors.

☞ You can also define scan points in the crosshatched area of the live video image. However, when scanning, these scan points themselves are not hit directly but the next attainable point respectively.

6. When you have defined all scan points, in the tool bar of the application window, click  again.

Test

As the live video image shows a two-dimensional image of the three-dimensional object, it is possible that the laser beam does not hit some of the scan points exactly. So check the scan points as follows:

7. Point at the live video image and click a scan point. The laser beam moves to this point on the object. Only PSV 300: You can also position the laser beam on scan points using the hand set PSV-Z-051. See your hardware manual on this.

8. Check if the position of the laser beam on the object corresponds to that on the live video image.

9. Repeat steps 7 and 8 for a few more scan points.

4.1 Standard Mode

In standard mode you draw geometric figures on a global, right-angled grid. You can

- draw rectangles, ellipses and polygons
- determine the density of the grid points
- move the grid.

To go into standard mode, in the graphics tool bar, click  .

4.1.1 Drawing Rectangles and Squares

To draw a rectangle or a square, proceed as follows:

1. Click  .

2. Point at the live video image. The cursor becomes a cross.

Rectangle

There are two ways of drawing a rectangle:

3. Point at a corner of the rectangle.

4. Press the mouse button and drag to the diagonally opposite corner.

or

5. Point at the center of the rectangle.

6. Press the control key and the mouse button simultaneously and drag across to a corner point.

Square

There are two ways of drawing a square:

7. Point at a corner of the square.

8. Press the shift key and the mouse button simultaneously and drag to the diagonally opposite corner.

or

9. Point at the center of the square.

10. Press the control key, shift key and the mouse button simultaneously and drag across to a corner point.

4.1.2 Drawing Ellipses and Circles

To draw an ellipse or a circle, proceed as follows:

1. Click  .
2. Point at the live video image. The cursor becomes a cross.

Ellipse

There are two ways of drawing an ellipse:

3. Point at a corner of the rectangle enclosing the ellipse.
4. Press the mouse button and drag to the diagonally opposite corner.

or

5. Point at the center of the ellipse.
6. Press and hold the control key and then the mouse button and drag across to a corner point of the rectangle enclosing the ellipse.

Circle

There are two ways of drawing a circle:

7. Point at a corner of the square enclosing the circle.
8. Press and hold the shift key and then the mouse button and drag to the diagonally opposite corner.

or

9. Point at the center of the circle.
10. Press the control key, shift key and the mouse button simultaneously and drag across to a corner point of the square enclosing the circle.

4.1.3 Drawing Polygons

To draw a polygon, proceed as follows:

1. Click  .
2. Point at the live video image. The cursor becomes a cross.
3. Click at least three corners of the polygon.
4. Double-click the last corner. The software connects it to the first corner.

4.1.4 Deleting Scan Points within a Figure

You can delete all scan points within a figure. Thus you can hide areas which you do not want to scan. To do this, proceed as follows:

1. Select the figures as described in section 4.4.

2. Click  .

4.1.5 Setting up the Grid

In standard mode there is a global grid over the whole live video image.

Show and hide the grid

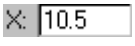
To show (hide) the grid in the live video image, click  .

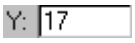
Density of grid points

The density of the grid points $d_{x,\,y}$ determines the distance between the grid points $a_{x,\,y}$ on the live video image as follows:

$$a_{x,\,y} = \frac{h_y}{d_{x,\,y}} \qquad \text{Equation 4.1}$$

h_y … Height of the live video image.

You can enter the density of the grid points in the fields 

and 

. You can also enter decimals.

You can also scale the grid at the same time as all the figures. To do this, proceed as follows:

1. If you cannot see the grid on the live video image, it would be useful to display it. To do so, click  .

2. Click  . All figures are selected.

3. Point at one of the white handles which mark the edge of the figures. The cursor becomes a double arrow.

There are now three ways of scaling:

4. Press the mouse button and drag in one of the directions the double arrow is pointing in. The anchor point is opposite the point you are dragging.

or

5. Press the control key and the mouse button and drag in one of the directions the double arrow is pointing in. The anchor point is the center of the figures.

or

6. Press the shift key and the mouse button and drag across to a corner point. The figures and the grid are scaled in proportion horizontally and vertically.

Move the grid

You can move the grid on the live video image. To do this, proceed as follows:

1. Click  .
2. Point at the live video image.

To move the grid without the figures:

3. Point at a dot outside the figures.
4. Press the mouse button and drag in the direction required.

To move the grid at the same time as all the figures:

5. Point at a dot within a figure.
6. Press the mouse button and drag in the direction required. (You can also use the arrow keys.)

4.2 Professional Mode (as an Option)

Professional mode is only available if you have the option APS Professional. In professional mode you can draw rectangles, ellipses and polygons as described in section 4.1.1, section 4.1.2 and section 4.1.3. Apart from that you can

- set up the grid for every figure individually and it can also be polar or hexagonal
- draw polylines
- rotate figures
- edit figures.

To go into professional mode, in the graphics tool bar, click  . The dialog Object Properties appears.

4.2.1 Drawing Polylines

To draw a polyline, proceed as follows:

1. Click  .
2. Point at the live video image. The cursor becomes a cross.
3. Click the corners of the polyline.
4. Double-click the last corner.

4.2.2 Rotating Figures

To rotate figures, proceed as follows:

1. Select the figures as described in section 4.4.

There are now two ways of rotating figures:

2. Enter the absolute angle of rotation in the dialog Object Properties, group Object in the field Rotation.

or

3. Click  .
4. Point at one of the white handles which mark the edge of the figures.
   The cursor becomes a  .
5. Press the mouse button and drag in the direction you want to rotate in. The anchor point is the center of the figures.

4.2.3 Changing the Shape of a Figure

You can retrospectively insert corners in figures and move corners individually. To do this, proceed as follows:

1. Select the figures as described in section 4.4.
2. Click  .

To insert a corner:

3. Point at the edge of the figure. The cursor becomes a  .
4. Click the edge of the figure. The software inserts a corner at this point.

To move a corner:

5. Point at the corner. The cursor becomes a .
6. Press the mouse button and drag in the direction required.

4.2.4 Distributing Scan Points Among Figures

Using the icons  ,  ,  and  in the dialog Object Properties you can distribute scan points on figures. To do this, first select the figures as described in section 4.4. Then click

 : You define scan points inside and on the edges of the figures (not active for polylines).

 : You define scan points only within the figures (not active for polylines).

 : You only define scan points on the edges of the figures.

 : You delete the scan points inside and on the edges of the figures.

4.2.5 Setting up the Grid

In professional mode, every figure has got its own grid. To set up the grid, first of all select the figure as described in section 4.4. In the dialog Object Properties you can then

- select the type of grid
- set the density of the grid points
- rotate the grid.

For polylines and figures which only have scan points on the edge (refer to section 4.2.4), you cannot set up a grid.

Type of grid

To select the type of grid, click

 : You select a right-angled grid.

 : You select a polar grid.

 : You select a hexagonal grid.

Density of grid points

You enter the density of the grid point in the dialog Object Properties in the respective field Density. You can also enter decimals. The density is independent of the dimensions of the figure, it always refers to the height of the live video image.

For a figure with a right-angled grid, in the dialog the fields Density X and Density Y appear. The density of the grid points $d_{x,y}$ determines the distance between the grid points $a_{x,y}$ as follows:

$$a_{x,y} = \frac{h_y}{d_{x,y}} \qquad \text{Equation 4.2}$$

h_y ... Height of the live video image.

For a figure with a polar grid, the fields Density radial and Density tang appear. The radial density d_{radial} determines the distance of the grid points on the beams a_{radial} as follows:

$$a_{radial} = \frac{h_y}{d_{radial}}$$ Equation 4.3

h_y … Height of the live video image.

The tangential density d_{tang} is equal to the number of grid points on the circles.

For a figure with a hexagonal grid, the field Density appears. The density of the grid points d determines the distance between neighboring grid points a as follows:

$$a = \frac{h_y}{d}$$ Equation 4.4

h_y … Height of the live video image.

Rotate grids You can rotate a grid relatively to its figure. To do so, enter the angle of rotation in the field Rotation, group Grid.

4.2.6 Vertex Points of Ellipses

The software approximates ellipses by polygons (at least quadrangles). You can set the number of vertex points. To do this, first select the ellipse as described in section 4.4. In the dialog Object Properties, the field Vertex Points then appears, in which you can enter the required number of vertex points. The number of vertex points then also applies for all ellipses which you subsequently draw.

☞ If you define scan points on the edge of an ellipse (refer to section 4.2.4), then every vertex point becomes a scan point.

4.2.7 Properties of Lines

Polylines and figures which only have scan points on the edge (refer to section 4.2.4), do not have a grid. However, you can set the density of the scan points on the lines. To do this, first select the figure as described in section 4.4.

Resolution In the dialog Object Properties, the field Resolution appears. The resolution is inversely proportional to the distance between the scan points on the line.

Thickness For polylines, in addition the field Width appears. The width determines the size of a shadow around the polyline. Within the shadow, the scan points which are behind the polyline are hidden. This means, for example, you can delete scan points on cables which obscure the view of the object.

4.3 Point Mode (as an Option)

Point mode is only available if you have the option APS Professional. In point mode you do not draw figures but edit individual scan points and their connections. You can take over scan point definitions from the other modes and fine-tune them. To do this, proceed as follows:

1. Draw figures in standard or professional mode as described in section 4.1, section 4.2 and section 4.4.

2. Go into point mode. To do so, in the graphics tool bar, click  . A message window appears. To separately save the scan points which have already been defined and to then set new ones, click Save. To add more to those which already exist, click Continue.

3. Click Continue. The software groups all scan points which have already been defined and selects the group.

4. Undo grouping. To do so, click  . The software selects all scan points.

5. Undo selection. To do so, click the live video image outside the scan points.

6. You can now edit scan points as described in section 4.4. Apart from that, you can define individual scan points as described in section 4.3.1. Only PSV 300: You can edit scan points with the hand set PSV-Z-051, see your hardware manual on this.

7. You can edit connections of scan points as described in section 4.3.2 and section 4.3.3.

4.3.1 Defining an Individual Scan Point

There are two ways of defining a scan point:

1. Click  .

2. Point at the live video image and click. A scan point is defined at this point.

or

1. Move the laser beam on the object to the point at which you want to define a scan point. See section 3.1 on this.

2. Click  .

4.3.2 Defining Connections

There are two ways of connecting scan points. You can connect two scan points and thus define a line. You can also connect three scan points and thus define a triangle.

☞ In point mode it is useful to select the view  (refer also to section 4.4). You can then see which areas are enclosed by scan points.

To connect two scan points:

1. Click  .
2. Point at the first scan point. The cursor becomes a cross.
3. Click the scan point.
4. Point at the second scan point.
5. Double-click.

To connect three scan points:

1. Click  .
2. Point at the first scan point. The cursor becomes a cross.
3. Click the scan point.
4. Point at the second scan point and click.
5. Point at the third scan point and click. The software connects this scan point to the first one.

4.3.3 Deleting Connections

To delete connections, proceed as follows:

1. Click  .
2. Point at a connection. The cursor becomes a  .
3. Select the connection. To do so, click it.
4. To select further connections, click them while holding the shift key pressed.
5. To remove a connection from the selection, click the connection holding the shift key pressed.
6. To select all connections, press the key combination Ctrl+A or select Edit > Select All.

7. To delete the selected connections, press the Delete key or select Edit > Delete.

☞ To delete a triangle, you only have to delete one of the connections. The software automatically deletes the other connections.

4.4 Common Functions in All Modes

Show and hide scan points

To show and hide the scan points in the live video image, in the tool bar of the application window, click  or select View > Scan Points. You see the scan points as blue squares. You see the connections between scan points as lines.

Present figures

Using the icons  ,  ,  and  , you can present figures in different ways. To do this, first select the figures as described below. Then click

 : You see the scan points and their connections.

 : You see the scan points, their connections and cross-hatching of the areas enclosed by the scan points. As a general rule, these areas are different from the geometric figures you have drawn.

☞ In presentation mode, areas which are enclosed by scan points are shown as enclosed areas.

 : You see the connections between scan points.

 : You see cross-hatching of the areas enclosed by scan points.

Select figures

To edit figures, you have to select them first. To do this, proceed as follows:

1. Click  .
2. Point at a figure. At the right below the cursor, a cross appears. ✣
3. Click. The software marks the edge of the figure with white handles.
4. To select further figures, click them while holding the shift key pressed. The software marks every selected figure with white handles.
5. To select all figures, select Edit > Select All or press the key combination Ctrl+A.
6. To remove a figure from the selection, click the figure holding the shift key pressed. The white handles disappear.

Move figures

To move figures, select them as described above and then use the mouse to drag them in the direction required. You can also use the arrow keys.

You can move scan points which you defined in point mode either individually or in groups. You move individual points, connected or freestanding in exactly the same way as you move figures (see above). If you want to move several points at the same time, then you first have to select them. To do this, you have the following possibilities:

- Hold the shift key pressed and click the individual points. Then release the shift key.
- Mark the points you would like to move. To do so, use the mouse to draw a rectangle which encloses these points.

Move the points with the mouse.

Resize figures

To resize figures, proceed as follows:

1. Select the figures as described above. If you want to resize scan points which you defined in point mode, then first of all select them as described above under Move figures. Then click  , to group the selected points.

2. Point at one of the white handles which mark the edge of the figures. The cursor becomes a double arrow.

There are now different ways to scale:

3. Press the mouse button and drag in one of the directions the double arrow is pointing in. The anchor point is opposite the point you are dragging.

or

Press the control key and the mouse button and drag in one of the directions the double arrow is pointing in. The anchor point is the center of the figures.

or

Press the shift key and the mouse button and drag across to a corner point. The figures are resized in proportion horizontally and vertically.

Duplicate figures

To duplicate figures, proceed as follows:

1. Select the figures as described above.

2. Click  or select Edit > Copy. The figures are copied onto the clipboard.

3. Click  or select Edit > Paste.

Delete figures

To delete figures, proceed as follows:

1. Select the figures as described above.
2. Press the Del key or select Edit > Delete.

Group figures

To group figures, proceed as follows:

1. Select the figures as described above.
2. Click  .
3. To undo grouping, click  .

Arrange figures

Using the four icons  ,  ,  and  , you can arrange figures behind each other (not in point mode). To do this, first of all, select one of the figures as described above. Then click

 or  : The figure moves into the foreground or background.

 or  : The figure moves one layer forwards or backwards.

Undo and redo

You can undo up to 20 operations and then redo them. To do so, click  and  or select Edit > Undo and Edit > Redo.

5 Making Measurements

You can make single point measurements and scan using the software. In section 5.1 and section 5.2 the different measurements are described. There you will also find some explanations of the respective procedure.

When a measurement is running, the status bar can display messages on it. See section 5.3 on this.

In section 5.4 is described what you have to pay attention to if you stop a measurement.

In section 5.5 you will find a few example applications which give you detailed instructions on various measurement tasks.

5.1 Single Point Measurement

You can perform single point measurements as single shots or continuous measurements.

Single shot

With a single shot, the software makes a single measurement and then ends data acquisition. You can also stop single shots manually.

Continuous measurement

With a continuous measurement, the software repeats single shots until you manually stop data acquisition. You can only save the last single shot.

To make a single point measurement, proceed as follows:

Preparation

1. Go into acquisition mode. To do so, click  or select View > Acquisition.
2. Set the software up for data acquisition as described in chapter 2.
3. Set the optics and position the laser beam at the point at which you want to make a measurement. See chapter 3 on this.
4. Set the parameters for data acquisition. See chapter 6 on this.

Start

5. Start a single shot or a continuous measurement. To do so, click  or  . You can also select Acquisition > Single Shot or Acquisition > Continuous.

Follow the measurement

6. During the measurement, you can display data in analyzers as described in section 7.2 and section 7.4.
7. During the measurement, the status bar and scanning head control can display messages on the data acquisition. See section 5.3 on this.

Stop

8. To stop the measurement, click  or select Acquisition > Stop. Please see section 5.4 on this as well.

5.2 Scanning

A scan is a sequence of single point measurements. The order in which the software approaches the scan points is determined by an internal algorithm. For every scan point, the software carries out the following steps:

- positions laser beam at scan point
- waits for end of settling time of scanner mirrors
- makes single shot
- allocates scan point status
- saves measurement data.

After a scan, the software can automatically remeasure certain scan points or you can start remeasuring manually.

To scan, proceed as follows:

Preparation

1. Go into acquisition mode. To do so, click  or select View > Acquisition.
2. Set the software up for data acquisition as described in chapter 2.
3. Set the optics. See chapter 3 on this.
4. Define the scan points as described in chapter 4.
5. Set the parameters for data acquisition. See chapter 6 on this.

Start

6. Start the scan. To do so, click  or select Acquisition > Scan. If the function generator is active (refer to section 6.2.9), the signal output starts. The software creates a file for the scan. The dialog Save As appears.

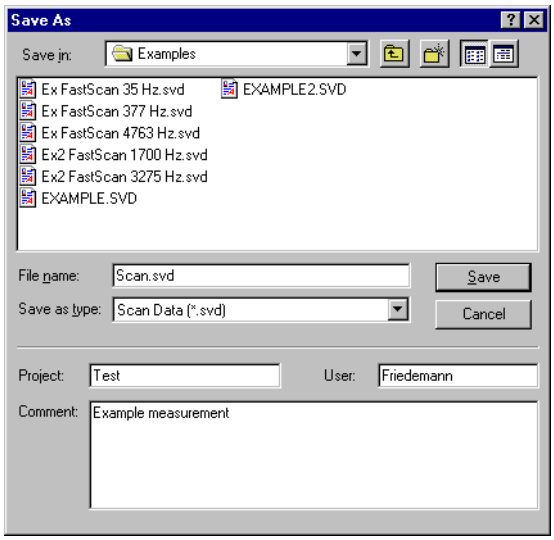


Figure 5.1: Dialog Save As

Save

7. Navigate to the saving location and enter the file name.
8. At the bottom in the dialog you can enter additional information to give the file more precise properties. This information is part of the file properties which you can also view and edit in the Explorer.
9. Click Save. The scan starts. The software immediately saves every scan point measured.

Follow the scan

10. You can follow the progress of the scan in the video window as described in section 7.1. During the scan, you can display data in analyzers as described in section 7.2 and section 7.4.
11. During the scan, the status bar and scanning head control can display messages on the data acquisition. See section 5.3 on this.

Stop the scan and continue

12. Once the total sample time has expired, the software ends data acquisition and displays a message. However, you can also stop the scan manually. You will find more details on this in section 5.4.
13. You can continue a stopped scan. To do so, select Acquisition > Continue.

Remeasure

14. After a scan you can remeasure scan points with the status Low Signal or Overrange at a slightly different position. To do so, select Acquisition > Remeasure. If Signal Enhancement is active for at least one channel (refer to section 6.2.7), then scan points with the status Valid are also remeasured.

☞ The software can also remeasure automatically, see section 6.2.1 on this.

5.3 Messages while Measuring

During a measurement, the status bar and scanning head control can display messages on the data acquisition. A list of messages, their causes and information on prevention can be found in table 5.1.

Table 5.1: Messages on data acquisition

Message	Cause	If the message appears permanently:	See
OVERRANGE VIB (Status bar)	The voltage on the vibrometer channel exceeds the input range set.	Click A/D, display the page Channels and set to the next highest input range of the vibrometer channel. Increase the input range on the page Channels step by step up to 10V. Then display the page Vibrometer and increase the measurement range for the velocity step by step.	section 6.2 .2
☞ The message OVERRANGE VIB can also appear briefly, caused by noise p dropouts. In this case, do not reset the input range.			
OVERRANGE REF (Status bar)	The voltage on the reference channel exceeds the input range set.	Click A/D, display the page Channels and set to the next highest input range of this reference channel.	section 6.2 .2
NO TRIGGER (Status bar)	Triggering is set but the software can not detect a trigger signal.	Click A/D, display the page Trigger and check the trigger settings. Also check the cabling.	section 6.2 .6, hardware manual
Display OVER is red (Scanning head control)	The measurement range for the velocity has been exceeded.	Click A/D, display the page Vibrometer and set to the next highest measurement range for the velocity.	section 6.2 .8
☞ Even if you are only measuring and evaluating the displacement signal, the range for the velocity must not be exceeded.			

5.4 Stopping Measurements

With single shots and with scanning, the software ends data acquisition once the total sample time has expired. You can also stop measurements manually. Continuous measurements have to be stopped manually.

To stop a measurement, click  or select Acquisition > Stop.

Single point measurement

If you stop a single point measurement, the following applies:

- If there are less than three seconds to the end of the sample time, the software carries out the measurement completely and then ends data acquisition.
- If there are more than three seconds to the end of the sample time, the software aborts the measurement. In the measurement modes FFT and Zoom FFT, you can always display and save the time signal for aborted measurements, but possibly not a spectrum. For long sample times, single shot is recommended if you want to evaluate and save spectra.

Scan If you stop a scan, the following applies:

All completely measured scan points are already saved. You can evaluate the scan the same as you can a completed one.

5.5 Example Measurements

Here you will find a few example applications which give you detailed instructions on various measurement tasks.

5.5.1 Optimizing Input Signals with Signal Enhancement and Speckle Tracking

When scanning, with Signal Enhancement and Speckle Tracking, you get an approximately even noise level for all scan points.

☞ Speckles occur when laser light is scattered back from optically rough surfaces. The light scattered back from a certain point can cause constructive or destructive interference. Correspondingly, the detector sees a bright or a dark speckle. Dark speckles lead to dropouts in the vibrometer signal. Dropouts increase the noise level significantly.

With Signal Enhancement and Speckle Tracking, you suppress dropouts in averaged spectra. In addition, the software evaluates the noise level of the individual scan points. You will find more information on this in your theory manual.

To activate Signal Enhancement and Speckle Tracking, proceed as follows:

1. Click A/D or select Acquisition > Settings. The dialog Acquisition Settings appears.
2. Display the page SE.
3. On the left you mark the channels which Signal Enhancement is to be active for. You will find information on this in section 6.2.7.
4. If Signal Enhancement is active for the vibrometer channel, then you can activate Speckle Tracking for this channel on the right.
5. Set the parameters on all other pages of the dialog. You will find information on this in section 6.2.
6. Click OK.

You can now start a measurement as described in section 5.1 or section 5.2.

5.5.2 Controlling an External Function Generator (as an Option)

Using the software you can control certain function generators via an IEEE-488/GPIB interface. To do this, you need the option External Function Generator.

☞ You can set which generator you are using via Set Up > Options on the dialog page Devices.

The software supports the following function generators:

- HP 33120 A
- Agilent 33120 A
- Agilent 33250 A
- PREMA ARB 1000.

To control an external function generator, proceed as follows:

Make a connection

1. Connect your workstation to the function generator via an IEEE 488/GPIB or RS-232 interface.
2. Switch the function generator on.
3. Only 33120 A: Configure the interface of the function generator as described in its manual. Set HPIB/488 or RS-232 as the interface and as HPIB address 10.
4. Start the software or go into acquisition mode.
5. Select Set Up > Preferences and open the page Devices. Here you set the correct connection to the generator (refer also to section 2.1).
6. Click OK. The generator is initialized.

Set the parameters

7. Click A/D or select Acquisition > Settings. The dialog Acquisition Settings appears.
8. Display the page Generator.
9. At the top left, tick the box Active to activate the function generator.
10. Set the parameters for the signal you want to emit. You will find information on this in section 6.2.9.
11. Set the parameters on all other pages of the dialog. You will find information on this in section 6.2.
12. Click OK.

Signal output

13. Start (stop) the signal output of the function generator. To do so, click  or select Acquisition > Generator.

☞ When scanning, the signal output starts automatically.

You can now start a measurement as described in section 5.1 or section 5.2.

5.5.3 Controlling the Internal Function Generator (as an Option)

Using the software you can control the internal function generator of the PSV 300 or MSV 300. To do this, you need the option Internal Function Generator. The output signal of the internal function generator is available at a BNC jack on the junction box (refer to the hardware manual).

☞ You can set which generator you are using via Set Up > Options on the dialog page Devices.

To control the internal function generator, proceed as follows:

Set the parameters

1. Click A/D or select Acquisition > Settings. The dialog Acquisition Settings appears.
2. Display the page Generator.
3. At the top left, tick the box Active to activate the function generator.
4. Set the parameters for the signal you want to emit. You will find information on this in section 6.2.9.
5. Set the parameters on all other pages of the dialog. You will find information on this in section 6.2.
6. Click OK.

Signal output

7. Start (stop) the signal output of the function generator. To do so, click  or select Acquisition > Generator.

☞ When scanning, the signal output starts automatically.

You can now start a measurement as described in section 5.1 or section 5.2.

5.5.4 FastScan (as an Option)

You can fast scan at individual frequencies. To do this, you need the option FastScan. The principle of FastScans is a regression in the time domain.

To carry out a FastScan, proceed as follows:

Set the parameters

1. Click A/D or select Acquisition > Settings. The dialog Acquisition Settings appears.
2. Display the page General and select the measurement mode FastScan. In the dialog a page with parameters for FastScan appears. The pages Frequency, Window and Trigger disappear.
3. Display the page FastScan and set the parameters. You will find information on this in section 6.2.11.
4. Set the parameters on all other pages of the dialog. You will find information on this in section 6.2.

5. Click OK.

Making measurements

6. You start and stop FastScans the same way as normal scans. See section 5.2, section 5.3 and section 5.4 on this.

☞ Leakage hardly occurs with FastScans. Triggers are not active. If you are controlling a function generator with the software, then a sinusoidal signal with the frequency of the FastScan is used for excitation.

Display data

7. You can display data for FastScans as described in chapter 7 but no spectra.

Evaluate

8. You can evaluate FastScans as described in chapter 8 but only at the frequency of the FastScan. You can not define any other frequency bands.

5.5.5 MultiFrame Measurement (as an Option)

Using the software you can divide measurements on combustion engines into frames and analyze the frames individually. To do this, you need the option MultiFrame and a trigger signal (TTL) which determines the start of an engine cycle.

To make a MultiFrame measurement, proceed as follows:

Cabling

1. You can determine the boundaries of the frames automatically or manually (refer also to section 6.2.13). If you set the boundaries of every individual frame automatically, then you will need another trigger signal (TTL). Connect the trigger signals to the front panel of the junction box as described in table 5.2.

Table 5.2: Connection of the trigger signals for MultiFrame measurements

Trigger	Manual	Automatic frame boundaries	Use pulses for individual frames
Trigger signal for the start of the engine cycle	TTL INPUT TRIGGER or TTL TRIG IN	REF 3	TTL INPUT TRIGGER or TTL TRIG IN
Trigger signal for the boundaries of the frames	Not used	Not used	REF 3
Settings on the page Trigger	Source: External (TTL)	Source: Analog, Reference3 Conditions: Rising, 20% Level	Source: External (TTL)

Set the parameters

2. Click A/D or select Acquisition > Settings. The dialog Acquisition Settings appears.

3. Display the page General and select the measurement mode MultiFrame. In the dialog a page with parameters for the MultiFrame measurement appears.

4. Set all other parameters on the page General. You will find information on this in section 6.2.1.

5. Display the page MultiFrame and set the parameters. You will find information on this in section 6.2.13.

6. Display the page Frequency. Set the parameters as described in the steps 7 to 9.

7. At the top left, set the bandwidth.

8. At the bottom right, set the number of samples. Above that the software shows the lower RPM_{min} limit which it can acquire with the bandwidth and number of samples set. Set the number of samples so that the lower RPM limit is 5 to 40% below the RPM the engine is running at.

9. At the bottom left, set the number of FFT lines per frame to be as low as possible.

10. Display the page Window and set the window functions. As a general rule, the window function Tapered Hanning with a parameter of 10 is suitable for MultiFrame measurements. You will find more information on windowing in section 6.2.5 and in your theory manual.

11. Display the page Trigger and set the parameters according to table 5.2.

12. Set 0% pre-trigger.

13. Set the parameters on all other pages of the dialog. You will find information on this in section 6.2.

14. Click OK.

If you set the boundaries of the frames manually (refer also to step 1), now follow steps 15 to 20. If you set the boundaries automatically, you can skip these steps and continue with step 21.

Set frame boundaries manually

15. Start a single point measurement. To do so, click  .

16. In the analyzer you see a MultiFrame tool bar and in the diagram, the boundaries of the frames. To do so, in the MultiFrame tool bar, click  .

17. Point at a boundary in the diagram. The cursor becomes a ↔.

There are two ways of moving the boundaries:

18. To move a boundary, use the mouse to drag it to the right or left into the required position. You can only move each boundary in a certain range which depends on the neighboring boundaries.

19. To move all boundaries at the same time, press the shift key and drag to the right or left into the required position. All boundaries move at the same time and retain the same respective distances between them.

20. When you have adjusted all boundaries, click  again. Initially the boundaries jump back into their old positions in the analyzer. The new boundaries are not shown until you start a new measurement.

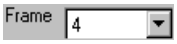
Making measurements

21. You start and stop MultiFrame measurements as described in section 5.1, section 5.2, section 5.3 and section 5.4.

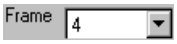
Display data

22. You can display data for MultiFrame measurements as described in chapter 7.

23. In analyzers you can also display the spectra of the individual frames and the average of all frames. To (de)activate the display of a spectrum, click the corresponding icon in the MultiFrame tool bar.

24. If you display a spectrum in the analyzer and have set a cursor, then in the legend you can also see the list Frame 

. From the list, select the data set you want to read.

25. In presentation windows you can either display an individual frame or the average respectively. Select the data set you want to display in the list Frame 

in the legend for the object.

Export a Universal File

26. If you export a MultiFrame measurement as a Universal File (refer to section 9.2.3), the software generates an individual file for each individual frame and the average respectively.

5.5.6 Controlling a Second Vibrometer

Apart from the PSV or MSV, you can control a second Polytec vibrometer at the same time. To do this, you need an IEEE-488/GPIB board or another RS-232 board in your workstation. You can set the parameters of the controller for data acquisition via the connection.

Supported controllers

You will find an overview of the controllers and interfaces supported as well as information on cabling in table 5.3.

Table 5.3: Supported controllers and interfaces

Vibrometer	IEEE-488/GPIB	RS-232	RS-232 cable
CLV-1000/2000/3000	-	x	Null modem (cross-wired)
HLV-1000	-	x	Null modem (cross-wired)
NLV-1232	-	x	Null modem (cross-wired)
OFV-3000/3001/3001 S	x	x	1:1
OFV-3020/3020 S	x	x	1:1
OFV-3300-1/3300-2	x	x	1:1
OFV-3310/3320	x	x	1:1
OFV-4000	x	x	1:1

To control a vibrometer, proceed as follows:

Make a connection

1. Connect the workstation to the controller via an IEEE-488/GPIB or RS-232 interface, refer also to table 5.3.
2. Switch the controller on.
3. Only OFV-3300-1, OFV-3310 and OFV-4000: Use the key ↑ on the front panel of the controller to switch its display to the menu SETTINGS. Then you can monitor the current settings on the display.
4. Start the software or go into acquisition mode.
5. Select Setup > Preferences. The dialog Preferences appears.
6. Display the page Devices. Make the necessary settings here (refer also to section 2.1).
7. Display the page Channels.
8. In the column Vibrometer, connect the controller to the digital channel it is providing the signal for. You will find information on this in section 2.2.
9. Click OK. The controller is initialized.

Set the parameters

10. Click A/D or select Acquisition > Settings. The dialog Acquisition Settings appears.
11. Display the page Vibrometer.
12. Set the parameters of the controller for data acquisition. You will find information on this in section 6.2.8 and in your vibrometer manual.
13. Set the parameters on all other pages of the dialog. You will find information on this in section 6.2.
14. Click OK.

You can now start a measurement as described in section 5.1 or section 5.2.

6 Parameters for Data Acquisition

There are three ways for you to set the parameters for data acquisition. See section 6.1 on this.

You will find a description of all parameters in section 6.2.

6.1 Setting the Parameters

You can set the parameters for data acquisition as follows:

- in the status bar
- in the dialog Acquisition Settings
- by loading the settings from a file.

Status bar

You can set some parameters on the right in the status bar. The parameters which are shown there depend on the measurement mode set. To see which parameters they are, point at a setting and leave the cursor there for a moment. The parameter is then shown on a yellow background next to the cursor and also on the left in the status bar. To set a parameter, click the arrow on the right of it and select the required setting in the popup menu.

Dialog

You can set all parameters in the dialog Acquisition Settings. To do this, proceed as follows:

1. Click  or select Acquisition > Settings. The dialog appears. It consist of several pages each with a group of parameters respectively. To display a certain page, click the name of the group.

2. Display the page General and set the parameters. You will find information on this in section 6.2.1. The pages in the dialog can change with the measurement mode selected.

3. Set the parameters on all other pages of the dialog. You will find information on this in section 6.2.

4. When you have set all parameters, click OK.

File

You can load settings (from setting files) and import them from measurements. See section 9.1.3 on this. You can save the current settings as described in section 9.2.6.

6.2 Description of the Parameters

6.2.1 General

On the page General, you set the measurement mode, averaging and automatic remeasuring.

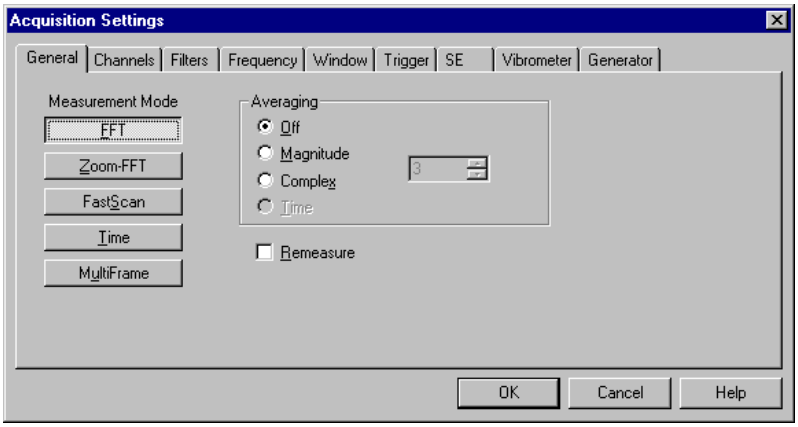


Figure 6.1: Page General

Measurement mode

On the left you select the measurement mode. The measurement mode FFT is available as standard, all others are optional. The pages in the dialog can change with the measurement mode selected. If, for example, you select the measurement mode FastScan, then a page appears with parameters for FastScan. Other pages disappear.

Averaging

On the right you set the averaging. In measurement mode FastScan you can't average. By averaging you can improve the signal-to-noise ratio of spectra. You will find detailed information on averaging in your theory manual.

Off: You don't average.

Magnitude: Select magnitude averaging if you

- measure without a reference signal or trigger

or

- use stochastic excitation – for example, in a wind tunnel.

Complex: Select complex averaging if you

- measure a reference signal or trigger

and

- use deterministic excitation.

On the right you enter the number of averages. The time needed for a measurement will increase by this factor.

Time: In the measurement mode Time you can only select time averaging. A prerequisite of this is that you trigger the measurements.

Remeasure

At the bottom you can activate automatic remeasuring. After a scan, all scan points with the status Low Signal or Overrange are remeasured at a slightly different position. If Signal Enhancement is active for at least one channel (refer to section 6.2.7), then scan points with the status Valid are also remeasured.

☞ You can also start remeasuring manually. See section 5.2 on this.

6.2.2 Channels

On the page Channels you activate the required measurement channels and set their parameters for digital data acquisition. For certain hardware configurations and measurement modes, parameters on the page are not available or are set permanently.

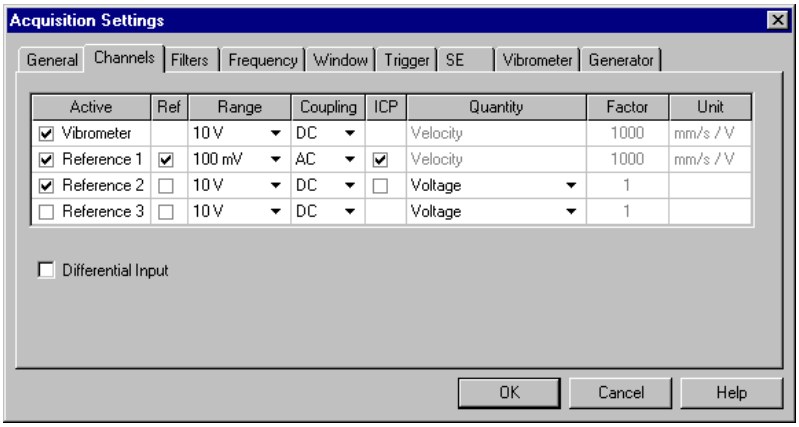


Figure 6.2: Page Channels

Active: Here you mark the channels you want to make measurements on.

Ref: This column only appears if there are more than two input channels available in the software. A channel is either an answer channel or a reference channel. The channel vibrometer is always an answer channel.

For systems with two channels the channel Reference 1 is always a reference channel. For systems with more than two channels you can select which channel is the reference channel (apart from Vibrometer). To do so, mark it in the column Ref. If for a channel the column Ref is not marked, it is used as an answer channel, even if it is called Reference.

The channel reference 1 is given special treatment. It serves as a phase reference for the other channels if you

- average magnitudes (refer to section 6.2.1) or
- measure without a trigger (refer to section 6.2.6).

When calculating the frequency response function, all possible combinations of type answer channel/reference channel are formed.

Example:

Vibrometer & Reference 1: answer channels; Reference 2 & Reference 3: reference channels

The following signals are available as frequency response function:

Vibrometer/Reference 2
Vibrometer/Reference 3
Reference 1/Reference 2
Reference 1/Reference 3

When using more than one reference channel, frequencies are respectively formed as a total answer signal/reference signal. This is correct with an excitation signal and several reference signals (e.g. self-excited system, vibrometer and microphone as reference).

☞ When using the measured frequency responses for experimental modal analysis, please note that these can not easily be fed in to a multi-reference procedure in the modal analysis software package. To do this, it is necessary to determine the transmission function. To calculate this the total answer signal of every sample point has to be divided up into the proportions of the individual references. This is only possible if the reference signals are uncorrelated (refer also to section 6.2.9).

Range: Here you select the input range of the data acquisition board for every channel.

Coupling: Here you select the input coupling for every channel. For frequencies below 200 Hz, DC coupling is recommended. Only PSV 200-1: For certain hardware configurations, select the input coupling via the cabling of the data acquisition board. See your PSV 200 hardware manual on this.

ICP: If you are using the junction box VIB-Z-012 or VIB-Z-016 and you have set them up correspondingly under Set up > Preferences on page Devices, then for the channels REF 1 and/or REF 2, you can switch ICP on and off. The box Differential Input allows you to switch the differential input on or off for all channels.

☞ If you use the junction box VIB-Z-016 and have activated ICP, then you should select AC coupling.
Pay attention to the cutoff frequency (refer to the hardware manual).

Quantity: Here you select the physical quantity which you want to measure on the respective channel.

☞ Using a digital filter, the software can integrate or differentiate the time signal of some measured quantities. See section 6.2.3 on this.
If the quantity displacement, velocity or acceleration is selected, then the spectra of both the others are calculated by integration or differentiation. This also applies to the quantities angle, angular velocity and angular acceleration.

Factor: Here you enter the calibration factor for every channel, i.e. the measurement value which corresponds to an analog input signal of one volt. If a vibrometer is connected (refer to section 2.2), then the software enters the calibration factor for this channel corresponding to the measurement range set on the page Vibrometer (refer to section 6.2.8).

Unit: Here, the software enters the SI-unit of the set quantity.

6.2.3 Filters

On the page Filters you set digital filters for the input signals

- to limit the bandwidth

and

- for differentiation or integration.

The page Filters is divided into two parts. In the first part you select the type of filter to limit the bandwidth – high pass, low pass etc. – and select the integration or differentiation filter. In the second part you set the filter parameters to limit the bandwidth, i.e. cutoff frequency and quality. You can also see the frequency response there.

Select

In the first section of the page Filters you select the types of filters you want for each channel.

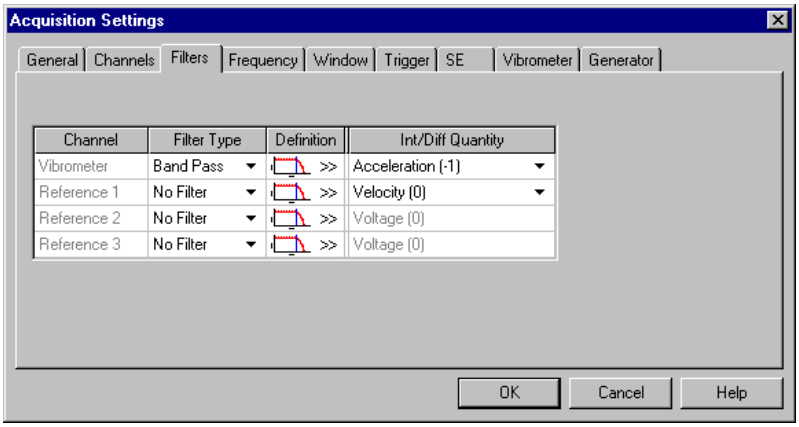


Figure 6.3: Page Filters

Filter Type: Here you select the type of filter to limit the bandwidth.

Definition: You click here to set the filter parameters to limit the bandwidth. The second part of the page Filters appears (see below).

Int/Diff: Using digital filters, the software can integrate or differentiate the time signal of some measured quantities. Here you select the physical quantity the software is to calculate. The number on the right next to the quantity tells you which arithmetic operation the filter is carrying out, on this, refer to table 6.1.

Table 6.1: Description of the calculations carried out by digital filters

Number	Calculation
2	Integrate twice
1	Integrate once
0	No filter
-1	Differentiate once
-2	Differentiate twice

Set the Parameters

In the second part of the page Filters you set the filter parameters to limit the bandwidth. There you can define the filters for several channels one after the other.

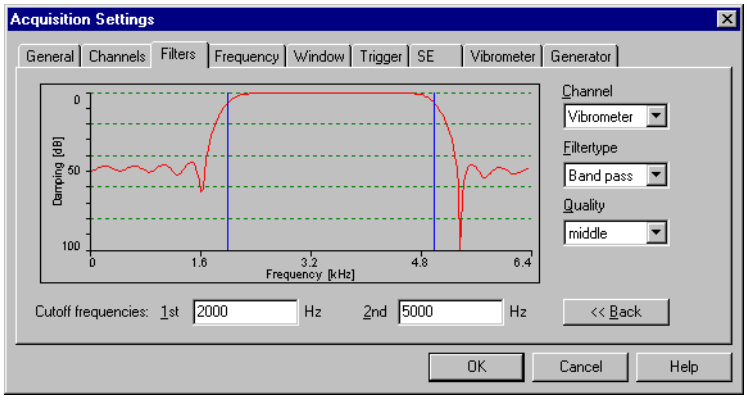


Figure 6.4: Page Filters, part 2 to set the parameters

Channel: Here you select the channel you want to set the filter for.

Filter Type: Select the type of filter here.

Quality: Here you select the quality of the filter.

Cutoff Frequencies: Enter the cutoff frequency(/ies) of the filter here in Hertz.

The software uses the parameters set to calculate the frequency response and displays it.

To apply all settings, at the bottom right, click Back. The software returns to the first part of the page Filters (see above).

6.2.4 Frequency

You set the parameters for calculating the frequency spectra on the pages Frequency and Window. In the measurement mode Zoom FFT, the page Frequency is adapted to the zoom FFT. See section 6.2.10 on this. In measurement mode MultiFrame, additional parameters appear for the MultiFrame measurement. See section 6.2.14 on this.

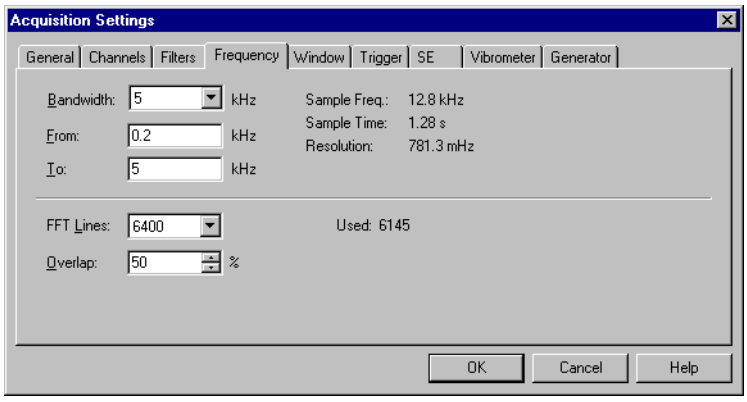


Figure 6.5: Page Frequency in the measurement mode FFT

Frequency range

At the top left you set the frequency range to be measured.

Bandwidth: Here you select the bandwidth in kilohertz.

From and **To:** Here you can limit the frequency range which is displayed and evaluated. If you control a function generator using the software, then with periodic chirp and pseudo random, excitation only takes place in this frequency range.

FFT

At the bottom you set the parameters for the Fast Fourier Transformation (FFT).

FFT Lines: Here you select the number of FFT lines that you want to analyze.

Used: The number of FFT lines shown here corresponds to the frequency range between From and To (see above).

Overlap: This parameter only appears for certain hardware configurations in the measurement modes FFT and Zoom-FFT. The software only takes the parameter into account for measurements with averaging and without trigger. With overlap you can significantly reduce the time for a measurement, in particular for narrow bandwidths. As a rule of thumb, for a two-channel measurement, the following applies:

- Up to 2.5 kHz bandwidth: 75% overlap
- 2.5 to 5 kHz: 50%
- Over 5 kHz: 0%.

You will find more information on overlap in your theory manual.

Sample frequency and resolution

The software calculates the following parameters and shows them at the top right.

Sample Freq. $f_{Sample} = 2.56 \cdot BW$ Equation 6.1

BW … Bandwidth

Sample Time: $t_{Sample} = \frac{n_{FFT}}{BW}$ Equation 6.2

n_{FFT} … Number of FFT lines
BW … Bandwidth

Resolution: $\Delta f = \frac{1}{t_{Sample}}$ Equation 6.3

t_{Sample} … Sample time

6.2.5 Window

You set the parameters for calculating the frequency spectra on the pages Frequency and Window. You set windowing of the input signals on the page Window. With suitable windowing you can prevent leakage. You will find information on this in your theory manual.

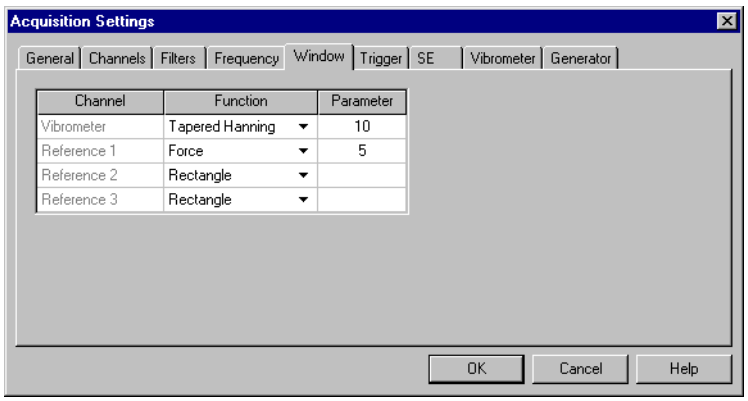


Figure 6.6: Page Windows

Function: Here you select a window function which is suitable for your application. See your theory manual on this.

Parameter: The window functions Tapered Hanning, Force and Exponential can be fine-tuned with a parameter. In table 6.2, the parameters are described.

Table 6.2: Parameters for window functions

Window function	Description of the parameter	Default
Tapered Hanning	Rise and decay time in percent of the sample time	10
Force	Length of the window in percent of the sample time	5
Exponential	Quotient from window length and decay constant	4

6.2.6 Trigger

You set the parameters for triggering on the page Trigger. You will find information on the influence of the trigger in your theory manual.

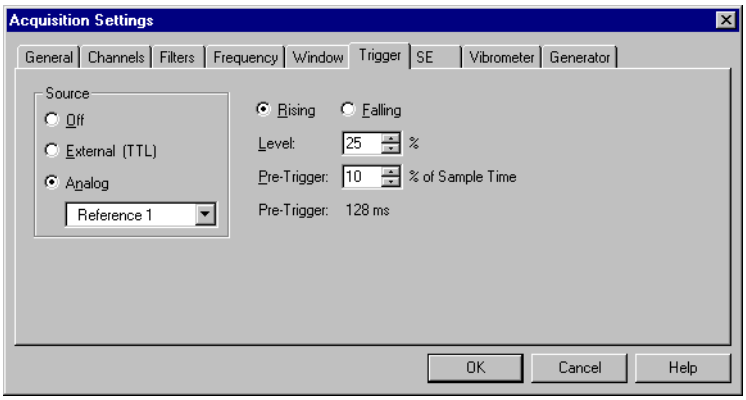


Figure 6.7: Page Trigger

Source

On the left you select the trigger source.

Off: You measure without a trigger.

☞ To measure phases you need a trigger or a reference signal.

External (TTL): You trigger using an external trigger signal.

> **Caution!**
>
> Only connect voltages in the range of 0 to 5 V to the input for the external trigger! Otherwise you could damage the data acquisition board.

Analog: You trigger using one of the analog input signals. Select the channel the trigger signal is connected to below.

☞ When scanning, don't trigger on the vibrometer signal.

Conditions

On the right you set the trigger conditions.

Rising or **Falling:** Here you select which edge is used to trigger.

Level: If you trigger using one of the analog input signals, you set the trigger threshold here. 100 % threshold corresponds to the input range of the channel (refer to section 6.2.2).

Pre-Trigger: Here you set the trigger time. If the data acquisition is to start **before** the trigger time, enter a positive value. If data acquisition is to start **after** the trigger time (Posttrigger), enter a negative value. Only PSV 300-F: You can not set a pre-trigger for an external trigger signal.

Gate signal (as an option)

As an option, you can control data acquisition with an external gate signal. You will find information on this in your hardware manual. You don't have to set special parameters for this.

6.2.7 Signal Enhancement (SE)

On the page SE you set the parameters for Signal Enhancement (SE) and Speckle Tracking. When scanning, with Signal Enhancement and Speckle Tracking, you get an approximately even noise level for all scan points. You will find information on measuring with Signal Enhancement in section 5.5.1 and in your theory manual.

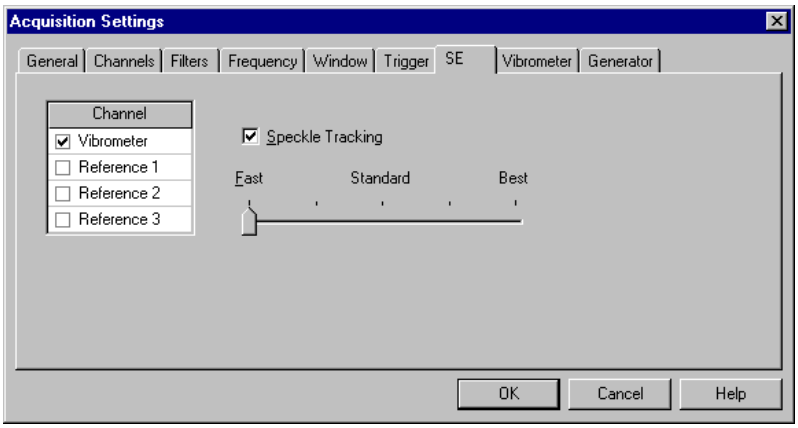


Figure 6.8: Page SE

The measurement mode Time is an exception. The dialog page SE only appears, if on the page General you have set time for Averaging.

Activate On the left you can activate Signal Enhancement.

Measurement mode Time: With active SE a median algorithm is used for averaging. This makes higher demands on your PC with regards to memory and calculation speed.

Channel: Here you mark the channels which Signal Enhancement is to be active for.

☞ Activate Signal Enhancement for all channels which may have dropouts. This is generally the case for the vibrometer channel. For reference channels, the following applies: If a vibrometer is connected, measure on treated surfaces if possible and switch Signal Enhancement off. To treat surfaces, stick reflective film on them or apply metallic paint. If you are measuring with the vibrometer, but not on treated surfaces, activate Signal Enhancement.

Speckle Tracking: If Signal Enhancement is active for the vibrometer channel, then you can activate Speckle Tracking for this channel on the right. Without Speckle Tracking, every scan point is measured at the same position when averaging. So it is possible that you will always measure a dark speckle. With Speckle Tracking the position of every scan point is changed slightly - at a meter stand-off distance by approx. 50µm. You are then very likely to measure some bright speckles as well.

In the measurement mode Time, the number of times averaging is carried out always correspond to the figure given on the page General.

Averaging

At the bottom right you set how often additional averaging is carried out with Signal Enhancement.

Fast: Averaging is only carried out as often as set on the page General (refer to section 6.2.1).

Standard: Depending on the optical signal level, averaging is carried out up to three times as often as set.

Best: Averaging is carried out up to five times as often as set.

6.2.8 Vibrometer

You set the parameters of the controller for data acquisition on the page Vibrometer. You will find information on suitable settings in your hardware manual. If you are controlling a second vibrometer, then you set the parameters for both controllers on the page Vibrometer. You will find information on controlling a second vibrometer in section 5.5.6.

The parameters displayed on the page Vibrometer depend on which controllers are connected (refer to section 2.2). The parameters for the controllers OFV-3001 and OFV-3001S are described below. If you would like any information on other controllers, refer to you vibrometer manual. There you will also find information on suitable settings.

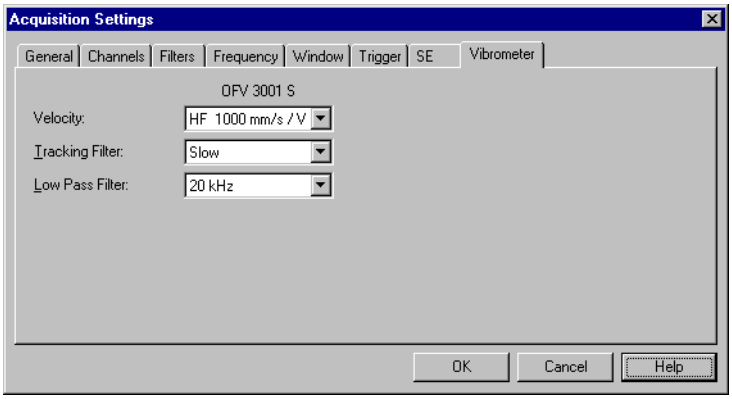


Figure 6.9: Page Vibrometer for the controller OFV-3001

In brackets is the corresponding name of the parameter on the controller's display.

Velocity (Velocity Range): Here you select the measurement range for the velocity. If a measurement range is available in several velocity decoders, then to the left of the measurement range you can see an abbreviation for the active velocity decoder. The abbreviations are explained in table 6.3.

Table 6.3: Abbreviations for velocity decoders

Abbreviation	Velocity decoder
HF	OVD-02 OVD-04
LF	OVD-06
DC	PLL-DC
PLL	OVD-01

Displacement (Displacement Range): This line only appears if the controller is equipped with an optional displacement decoder. Here you select the measurement range for the displacement.

☞ Even if you are only measuring and evaluating the displacement signal, the measurement range for the velocity must not be exceeded (see above).

Tracking Filter: Here you set the tracking filter.

Low Pass Filter (Velocity Filter): Here you set the low pass filter. Only PSV 300-H and PSV 200-1: If you select the setting User Defined, then the field Cutoff Frequency appears at the bottom. There you can enter multiples of 0.4 kHz, up to a maximum of 102.4 kHz.

6.2.9 Generator (as an Option)

As an option, you can control certain function generators using the software. You will find information on this in section 5.5.2 and section 5.5.3. You set the parameters for signal output on the page Generator.

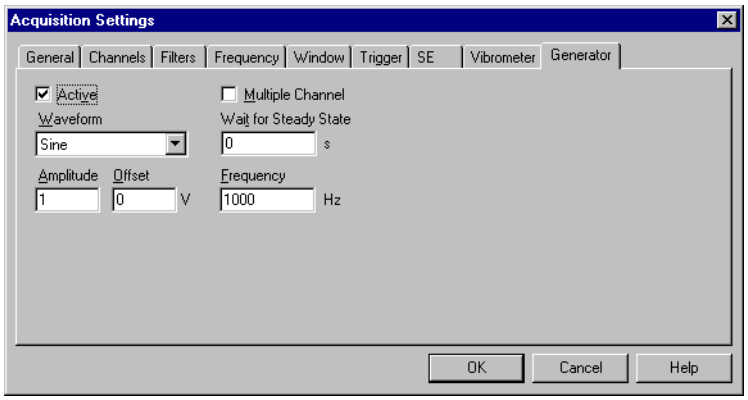


Figure 6.10: Page Generator with user defined waveform

General parameters

Active: If sometimes you are not using the function generator, you can deactivate it here for the software. Otherwise signal output starts automatically when scanning. Apart from that, you will suppress corresponding warnings and error messages.

Multiple Channels: Only PSV300-H: Here you activate three generator channels. For signals with randomly generated numbers (pseudo random, white noise, burst random), uncorrelated signals are generated using various random number sequences, otherwise identical signals on all three channels. You will find more information on this in section 6.2.2.

Waveform: Here you select the waveform you want to emit. Some parameters on the page Generator change with the waveform selected. You will find an explanation of these parameters and information on some of the waveforms below. In the measurement mode FastScan, a sinusoidal signal with the frequency of the FastScan is always used for excitation.

Wait for Steady State: Here you enter the time in seconds which the excited object needs to attain steady-state condition. The software delays data acquisition by this time after you have

- started signal output or
- closed the dialog Acquisition Settings with OK.

Amplitude and **Offset:** Here you enter the peak amplitude and DC offset of the signal in volts for high-impedance termination of the function generator. The following conditions must be fulfilled:

$$\text{Amplitude} + |\text{Offset}| \leq 10\ \text{V} \qquad \text{Equation 6.4}$$

Only 33120 A and 33250 A:

$$|\text{Offset}| \leq \text{Amplitude} \qquad \text{Equation 6.5}$$

If one of the conditions is not fulfilled, the software changes the offset.

☞ The function generator itself has an output impedance of 50 Ω. If you terminate the function generator with 50 Ω, then the output voltages are only half the size set.
Only for external function generators: The display shows Peak-Peak amplitude and offset with 50 Ω for the termination. Thus with high-impedance termination, the display only shows half of the offset set.

Sine, rectangle, triangle, ramp, user defined

Frequency: For the waveforms sine, rectangle, triangle, ramp and user defined, you enter the repeat frequency of the signal in hertz here.

Sweep

Start Frequency, End Frequency and **Sweep Time:** For a sweep, you enter the start and end frequency in hertz here. The start frequency can be higher than the end frequency. You also enter the duration of the sweep in seconds.

Periodic chirp, pseudo random

For the waveforms periodic chirp and pseudo random, sinusoidal signals are emitted to all FFT lines at the same time, but only in the frequency range between From and To on the page Frequency (refer to section 6.2.4). Only 33250A: The waveforms are not available in the measurement mode Zoom-FFT. Only 33120A: The waveforms are not available in the measurement mode Zoom-FFT or if 12,800 FFT lines are set.

With periodic chirp, the phases of the sinusoidal signals are adapted so that the energy of the resulting signal is maximized. With pseudo random, the phase of the sinusoidal signals is random.

Amplitude Correction File: As standard, with periodic chirp and pseudo random all sinusoidal signals have the same amplitude. If you need other amplitudes or frequencies, you can generate a correction file and list it here. To do this, tick the box and then click Browse. The dialog Open appears, in which you can navigate to correction files. You will find a short description of the format required in the file AmplitudeCorrectionSample.txt in the software's installation directory. You can also use this file as a template for your own correction files.

Burst chirp, burst random

The waveforms burst chirp and burst random are only available for certain hardware configurations.

Burst Start and **Burst End:** Here you enter when the burst is to start and when it is to end (input in percent of the sample time).

User defined

If you need other waveforms, you can generate them yourself. You will find a short description of the format required in the file UserDefSample.txt in the software's installation directory. You can also use this file as a template for waveforms you want to generate. To load a waveform you have generated, select the waveform User Defined and then click Browse. The dialog Open appears, in which you can navigate to files you have generated.

6.2.10 Frequency in the Measurement Mode Zoom-FFT (as an Option)

In the measurement mode Zoom-FFT you can also make high-resolution FFT measurements. To do this, you need the option Zoom-FFT. The page Frequency is then adapted to the Zoom-FFT.

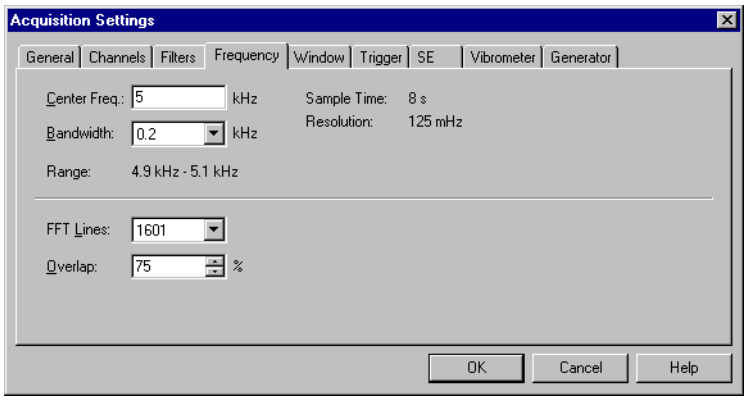


Figure 6.11: Page Frequency in the measurement mode Zoom-FFT

Frequency range

At the top left you set the frequency range to be measured.

Center Freq: Here you enter the center frequency in kilohertz.

Bandwidth: Here you select the bandwidth in kilohertz.

Range: The software calculates the frequency range to be measured and shows it here.

FFT

At the bottom you set the parameters for the Fast Fourier Transformation (FFT).

FFT Lines: Here you select the number of FFT lines that you want to analyze. The number is odd because the frequency range measured is symmetrical about the center frequency.

Overlap: This parameter only appears for certain hardware configurations. You will find information on this in section 6.2.4 and in your theory manual.

Resolution

The software calculates the following parameters and shows them at the top right.

Sample Time:

$$t_{Sample} = \frac{n_{FFT} - 1}{BW} \qquad \text{Equation 6.6}$$

n_{FFT} … Number of FFT lines
BW … Bandwidth

Resolution:

$$\Delta f = \frac{1}{t_{Sample}} \qquad \text{Equation 6.7}$$

t_{Sample} … Sample time

☞ The highest attainable resolution depends on the frequency range measured (see above).

6.2.11 FastScan (as an Option)

In the measurement mode FastScan you can scan individual frequencies quickly. You will find information on this in section 5.5.4. You set the parameters for this on the page FastScan.

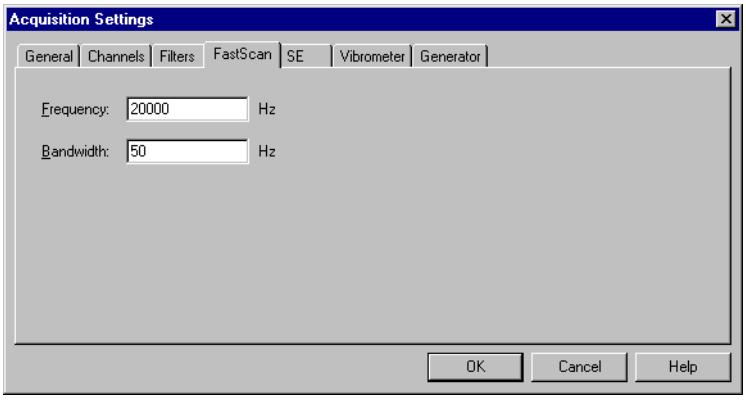


Figure 6.12: Page FastScan

Frequency: Here you enter the frequency in hertz at which you want to fast scan.

☞ If you control a function generator using the software, then in the measurement mode FastScan, a sinusoidal signal at the FastScan frequency is always used for excitation.

Bandwidth: Here you enter the bandwidth in hertz. As a general rule it can be said: The smaller the bandwidth, the better the signal-to-noise ratio, but the slower the scan.

6.2.12 Time (as an Option)

In the measurement mode Time you can save time signals instead of spectra for scans. To do this, you need the option Time Domain. You set the parameters for this on the page Time.

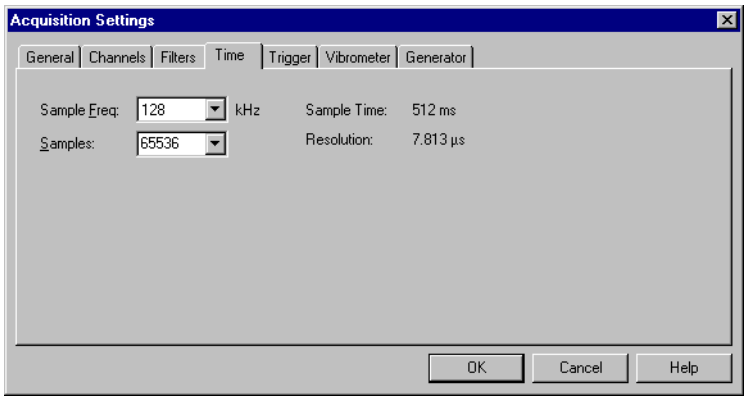


Figure 6.13: Page Time

Sample Freq: Here you select the sample frequency in kilohertz.

Samples: Here you select the number of samples. You can select in the list or enter any number.

The software calculates the following parameters and shows them on the right.

Sample Time:

$$t_{Sample} = \frac{n_{Sample}}{f_{Sample}}$$ Equation 6.8

n_{Sample} … Number of samples
f_{Sample} … Sample frequency

Resolution:

$$\Delta t = \frac{1}{f_{Sample}}$$ Equation 6.9

f_{Sample} … Sample frequency

6.2.13 MultiFrame (as an Option)

In the measurement mode MultiFrame you can divide measurements on combustion engines into frames and analyze the frames individually. You will find information on this in section 5.5.5. You set the parameters for this on the pages MultiFrame and Frequency.

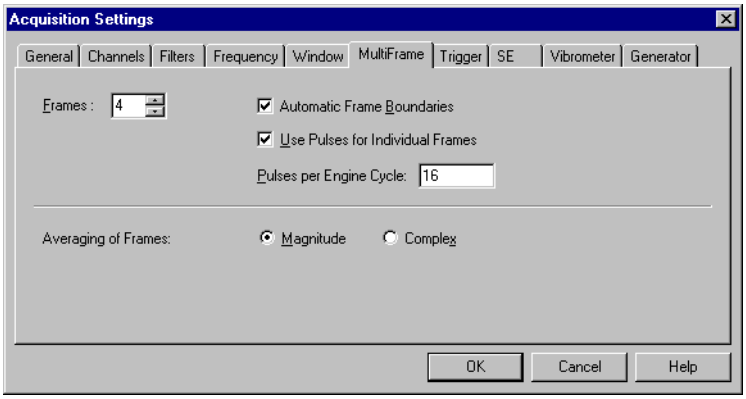


Figure 6.14: Page MultiFrame

Number of frames

At the top left you set the number of frames.

Frames: Here you enter how many frames an engine cycle has.

Frame boundaries

At the top right you set how you determine the frame boundaries.

Automatic Frame Boundaries: Tick the box if you want to determine the frame boundaries with a trigger signal. Do not tick the box if you want to determine them manually.

Use Pulses for Individual Frames: Tick the box if you want to determine each frame boundary with a trigger signal. You will need a trigger signal for this. Do not tick the box if you only want to determine the boundary of the last frame with a trigger signal. The software then calculates the boundaries of the other frames so that all frames are the same length.

Pulses per Engine Cycle: Here you enter how many trigger pulses are emitted per engine cycle. If Use Pulses for Individual Frames is active (see above), the number of pulses per engine cycle must be an integer multiple of the number of frames.

Averaging

You can analyze the spectra of the individual frames and the average of all frames. At the bottom you set how the software averages.

Magnitude or **Complex:** Here you select how the software averages over the frames. Averaging over the frames is independent of the averaging on the page General (refer to section 6.2.1).

6.2.14 Frequency in the Measurement Mode MultiFrame (as an Option)

In the measurement mode MultiFrame you can divide measurements on combustion engines into frames and analyze the frames individually. You will find information on this in section 5.5.5. On the page Frequency additional parameters then appear for MultiFrame measurement.

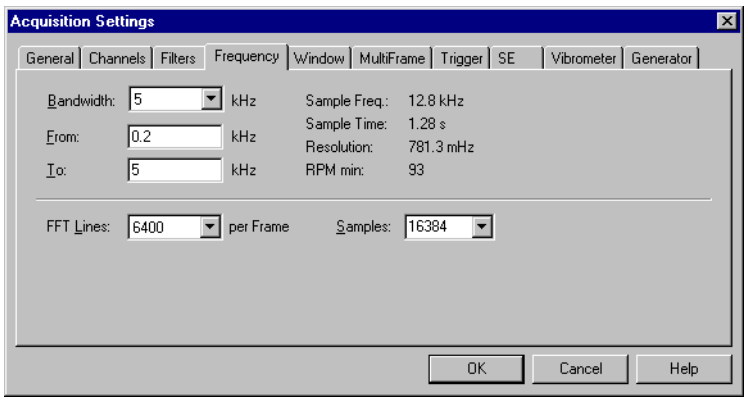


Figure 6.15: Page Frequency in the measurement mode MultiFrame

At the bottom right, set the number of samples.

Samples: Here you select the number of samples.

The software calculates the sample time and the lower RPM limit RPM_{min}, which it can acquire with the parameters set and shows them on the right.

Sample Time:

$$t_{Sample} = \frac{n_{Sample}}{2.56 \cdot BW} \qquad \text{Equation 6.10}$$

n_{Sample} … Number of samples
BW … Bandwidth

RPM min:

$$RPM_{min} = 2.56 \cdot \frac{120 \cdot BW}{n_{Sample}} \qquad \text{Equation 6.11}$$

BW … Bandwidth
n_{Sample} … Number of samples

You will find information on the other parameters on the page Frequency in section 6.2.4.

7 Displaying Data

In the software you work with the following windows to display data:

- in acquisition mode: a **video window**, any number of **analyzers**
- in presentation mode: any number of **presentation windows** and **analyzers**.

7.1 Following Scans in the Video Window

In the video window you follow the progress of scans. While scanning, you can see what status the scan points get. You will find a description of the video window in section 1.4.3.

Show and hide scan points

To show and hide the scan points in the video window, click  or select View > Scan Points. To the left of the live video image you see a legend for the status.

Status

Before a scan, all scan points have got the status Not Measured.

When scanning, the software gives every scan point the status Valid, Optimal, Low Signal or Overrange.

The status Optimal can only occur if Signal Enhancement is active for at least one channel (refer tosection 6.2.7). At these scan points, the signal-to-noise ratio is relatively high.

The status Low Signal can only occur if Signal Enhancement is switched off. At these scan points, the optical signal level was low.

At scan points with the status Overrange, the input range of the data acquisition board was exceeded on at least one channel.

If Signal Enhancement is active, scan points can change their status between Valid and Optimal at the end of the scan.

After the scan you can manually give the scan point the status Invalid. You will find details on this in section 8.1.

7.2 Working in Analyzers

Analyzers show measurement data and evaluated data in x-y diagrams. You will find a description of the analyzer in section 1.4.3.

You select the data set to be displayed in the analyzer as described in section 7.4.

Analyzers offer user-friendly functions to present data quickly and usefully. Directly in analyzers, you can zoom, set cursors, read data and autoscale the y-axis. You will find more functions in the dialog Analyzer Properties. There you can set the line style and color of the graphs, scale and set up the look of the diagrams.

7.2.1 Zooming

You can zoom in on sections of the diagrams.

Zoom the x- or y-axis

To zoom the x- or y-axis individually, proceed as follows:

1. Point at an edge point of the section required on the axis. The cursor becomes a magnifying glass.
2. Press the mouse button and drag horizontally or vertically to the other edge point.

Scroll the x-axis

The scroll bar below the diagram shows the position of the zoomed section on the whole x-range. You can scroll along the x-axis with the current zoom factor. To do so, move the bar in the scroll bar to the right or to the left. You can also click the arrows to the right and left of the scroll bar.

Zoom both x- and y-axis

To zoom both x- and y-axis simultaneously, proceed as follows:

1. Click  or select Analyzer > Zoom.
2. In the diagram, point at a corner of the section required.
3. Press the mouse button and drag to the diagonally opposite corner.

Zoom out

To undo the last zoom action or autoscaling, click  or select Analyzer > Zoom Out.

To undo all previous zoom actions, click  while holding the shift key pressed. You can also select Analyzer > Zoom Out holding the shift key pressed.

7.2.2 Scaling the Axes

You can scale the y-axis automatically or scale the x- and y-axis manually.

Autoscale the y-axis

When you have started a measurement or are displaying another data set, then the y-axis range is often unsuitable. You can then autoscale, i.e. the software selects the y-axis range displayed corresponding to the y-range of the data. To do so, click  or select Analyzer > Auto Scale. Even if the x-axis is zoomed, the whole y-range is autoscaled, not just the zoomed section.

Scale manually

In the dialog Analyzer Properties you can scale the manually and set further scaling properties. To open the dialog, point at an axis (the cursor becomes a magnifying glass) and double-click. You can also activate the analyzer (to do so, click it), select Edit > Properties and then display the page Ranges.

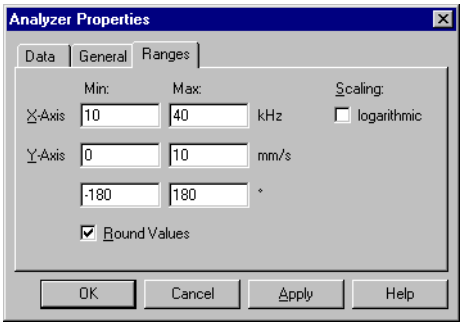


Figure 7.1: Page Ranges

Max and **Min:** Here you enter the x- and y-axis ranges you want to display.

Scale: This box only appears if the active analyzer is displaying a spectrum. Here you select linear or logarithmic scaling of the x-axis.

Round Values: Here you select whether the axis values displayed should be round. The software then rounds the values in Min and Max suitably. Round Values is also active when zooming. The zoomed section is then usually slightly larger than selected.

7.2.3 Setting Cursors and Reading Data

In the analyzer you can set cursors to read individual data points. Cursors are represented by thin vertical lines. In display type Nyquist you can not set any cursors. In section 8.4.2 you will find details on how to animate time data.

Single cursor

To set a single cursor, proceed as follows:

1. Click  or select Analyzer > Cursor.
2. Click in a diagram.

Read data

You can see the cursor's coordinates in the legend. If the legend is not visible on the right in the analyzer, select Analyzer > Legend to display it.

Differential cursor

You can set two cursors and read the difference between coordinates. To do this, proceed as follows:

1. Click  or select Analyzer > Differential Cursor.
2. In the diagram, point at the first point you require.

3. Press the mouse button and then drag to the right or left to the second point. The difference between the coordinates also appears in the legend.

Move the cursor

You can move a cursor to the right or left. To do this, proceed as follows:

1. Point at the cursor. The mouse cursor becomes a ↔.

2. Press the mouse button and then drag to the right or left to the required point. The data in the legend is updated simultaneously.

You can also move a single cursor using the keys → and ←.

Delete the cursor

To delete cursors you have set, click  or  again.

7.2.4 Setting up the Presentation of Graphs

You set up the presentation of the graphs in the dialog Analyzer Properties. To activate the dialog, point at a diagram and double-click. You can also activate the analyzer (to do so, click it), select Edit > Properties and then display the page Data.

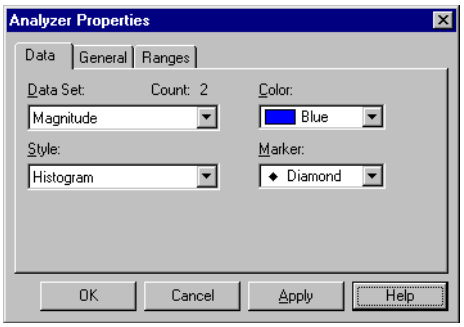


Figure 7.2: Page Data

Data Set: If the analyzer is displaying several diagrams, for example, magnitude and phase, then this is where you select the data set for which you want to set up the presentation.

Color: Here you select the color of the graph.

Style and **Marker:** Here you select the view style of the graph and, if applicable, the marker. In the view styles marker, histogram, thin bars and thick bars, only measured y-values are displayed. In the other view styles, the software interpolates between the measured values.

7.2.5 Setting up Diagrams

You set up the look of the diagrams in the dialog Analyzer Properties. The current settings in the dialog are also valid for all analyzers you now open. To open the dialog, double-click an axis name. You can also activate the analyzer (to do so, click it), select Edit > Properties and then display the page General.

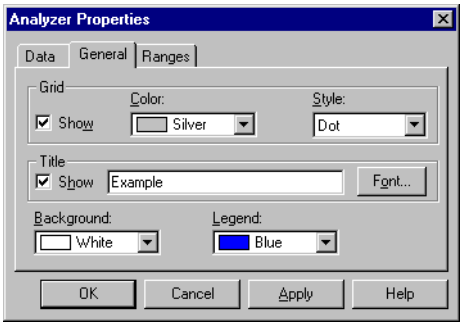


Figure 7.3: Page General

Grid

At the top you set up the grid.

Display: Tick the box to display a grid in the diagram.

Color and **Style:** Here you select the color and line style of the grid.

Title

In the middle you set up the diagram title.

Display: If you would like to display a title over the diagram, mark the box. You enter the title on the right.

Font: Click here to format the title. The dialog Font appears.

Colors

At the bottom you set up the background color of the diagram and the color of the data in the legend.

Background: Here you select the background color of the diagram.

Legend: Here you select the color of the data read in the legend.

7.3 Working in Presentation Windows

You analyze scans in presentation windows. You will find a description of the presentation window in section 1.4.4.

Depending on which functions you are working with in the presentation window, you display it in different views: See section 7.3.1 and section 7.3.2 on this. You will find more presentation options in the dialog Display Properties. You can scale there and set up the appearance of different views.

You select the data set to be displayed in the presentation window as described in section 7.4.

Apart from the display functions, in the presentation window you also have the possibility to evaluate the measurement data. See chapter 8 on this.

7.3.1 Setting up the View of the Presentation Window

Depending on which functions you are working with in the presentation window, you display it in one of four different views:

- Object
- Single Scan Point
- Average Spectrum
- Profile.

To set the view of the presentation window, click  and select the view required in the popup menu. You can also select Presentation > View. In the Object view you see the data superimposed on the video image. In the other three views you see the presentation window divided horizontally: at the top the Object view and at the bottom an analyzer. You work in the analyzer as described in section 7.2.

Object You see the data superimposed on the video image. You can present the data itself in five different ways, see section 7.3.2 on this. You can set up the look of the object (background color, title, transparency etc.) as described in section 7.3.4.

Single scan point In the upper part of the presentation window you see the object. At the bottom you see an analyzer. The analyzer displays the spectrum at the individual scan points. You select the scan point for which the spectrum is shown at the top. To do so, set a cursor at this scan point as described in section 7.3.3.

Average spectrum At the top you see the object. At the bottom the analyzer displays the average spectrum of all scan points.

Profile At the top you see the object. You can overlay profile sections on the object as described in section 8.3. You will see the profiles at the bottom in the analyzer.

7.3.2 Selecting the View Style of the Data

In the presentation window you see the data superimposed on the video image. You can present the data in five different ways:

- 2D
- 3D
- Isolines
- Scan Points
- Status

To select the view style of the data, click  and select the view style required in the popup menu. You can also select Presentation > View Style.

2D The data is color-coded. The software interpolates between the scan points.

3D You see the data in three dimensions. The software interpolates between the scan points. Outside the scan points, the video image is cut off. You can set up the 3D view style (perspective, colors, size) as described in section 7.3.6. You can also set the perspective directly in the presentation window using the arrow keys or the mouse. To do so, point at the object, press the mouse button and drag in the direction required.

Isolines You will see equal data values connected with color-coded isolines. You can set up the presentation of the isolines (number, thickness) as described in section 7.3.7.

Scan points You see each scan point as a color-coded square which corresponds to its data.

Status You see the scan points as squares in different colors. The color shows the status of the scan point.

Color-coding In some view styles you see the data color-coded. You can select the color palette for this as described in section 7.3.8. To the left of the object you can show (hide) a gauge for color-coding. To do so, select Presentation > Gauge.

7.3.3 Setting Cursors and Reading Data

If you would like to read data at individual scan points, you can set a cursor on the object. The cursor is shown as a flashing square. In the views Profile and Average Spectrum as well as in the view style 3D you can not set a cursor.

Set a cursor To set a cursor, proceed as follows:

1. Display the Object or Single Scan Point view. To do so, click  and select Object or Single Scan Point.
2. Display any view style you like apart from the 3D view style. To do so, click  and select the view style in the popup menu.
3. Point at a scan point and click. The scan point is highlighted.

Read data In the legend you can see the data of the scan point which the cursor is at. If the legend is not visible on the right of the object, select Presentation > Legend to display it.

In the Single Scan Point view at the bottom in the analyzer you can see the spectrum at this scan point.

Move the cursor You can move the cursor to other scan points using the mouse. The data in the legend is updated simultaneously. In the Object view you can move the cursor within a joined area using the arrow keys as well.

To do this in the Single Scan Point view as well hold the control key pressed.

Delete the cursor

To delete the cursor, in the tool bar of the presentation window, click  .

7.3.4 Setting up the Object

You set up the look of the object in the dialog Display Properties. To open the dialog, double-click the object. You can also activate the presentation window (to do so, click it) and select Edit > Properties. Then display the page General.

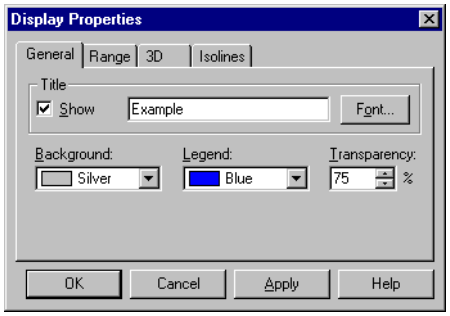


Figure 7.4: Page General

Title

At the top you set up the title of the object.

Display: If you would like to display a title over the object, mark the box. You enter the title on the right.

Font: Click here to format the title. The dialog Font appears.

Colors

At the bottom you select the background color of the object, the color of the data in the legend and the transparency of the data on the video image.

Background: Here you select the background color of the object.

Legend: Here you select the color of the data read in the legend.

Transparency: Here you enter how transparent the data is when superimposed on the video image. With 100% transparency you only see the video image.

7.3.5 Scaling the Z-axis

You scale the z-axis in the dialog Display Properties. To open the dialog, double-click the object. You can also activate the presentation window (to do so, click it) and select Edit > Properties. Then display the page Range.

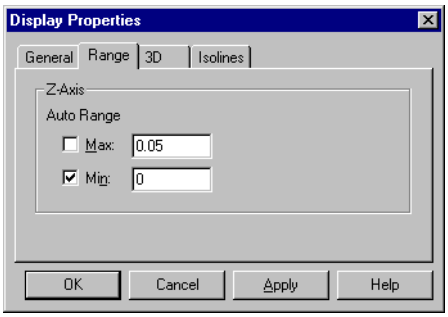


Figure 7.5: Page Range

Automatic

Auto Range: The software can autoscale maximum and minimum of the z-range individually. Tick the corresponding box to do so.

☞ Data which you have collected in time mode can also be scaled in the presentation window by clicking  .

Manual

Through entries in the fields Max: and Min: you can scale manually.

Max and **Min:** Enter the z-axis range here you want to display.

7.3.6 Setting up the 3D View Style

You set up the 3D view style in the dialog Display Properties. To open the dialog, double-click the object. You can also activate the presentation window (to do so, click it) and select Edit > Properties. Then display the page 3D. New settings on this page are immediately effective – i.e. without clicking Apply or closing the dialog.

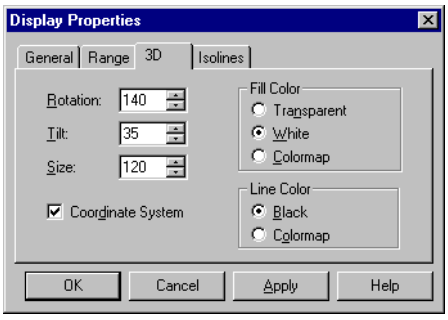


Figure 7.6: Page 3D

Perspective and size

On the left you scale and set the perspective from which you can see the object. Apart from that, you can show (hide) a small coordinate system which shows you the current perspective.

☞ You can also set the perspective directly in the presentation window using the arrow keys or the mouse, refer to section 7.3.2.

Rotation: You can rotate the measurement plane around the z-axis. Enter the angle of rotation in degrees here.

Tilt: You can tilt the measurement plane. Enter the tilt angle in degrees here.

Size: Enter the size of the object in percent here. The software scales all axes simultaneously.

Coordinate System: Tick the box to display a small coordinate system in the presentation window. The coordinate system is rotated and tilted with the object.

Colors

On the right you set the colors the software uses for the grid lines and the surfaces in between. The settings selected do not affect the 3D view style if you have set 100% transparency on the page General (refer to section 7.3.4). You can select the color palette for color-coding as described in section 7.3.8.

Fill Color: Here you select how the surface between the grid lines is to be filled.

Line Color: Here you select the color of the grid lines.

7.3.7 Setting up the Isolines

You set up the presentation of the isolines in the dialog Display Properties. To open the dialog, double-click the object. You can also activate the presentation window (to do so, click it) and select Edit > Properties. Then display the page Isolines. New settings on this page are immediately effective – i.e. without clicking Apply or closing the dialog.

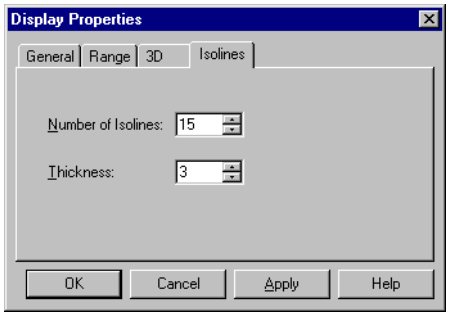


Figure 7.7: Page Isolines

Number of Isolines: Here you select how close the isolines are to each other.

Thickness: Here you select the thickness of the isolines.

7.3.8 Selecting the Color Palette

You can choose between different color palettes for color-coding in the presentation window. To do this, proceed as follows:

1. Click  or select Setup > Color Palette. The dialog Color Palette Selection appears.

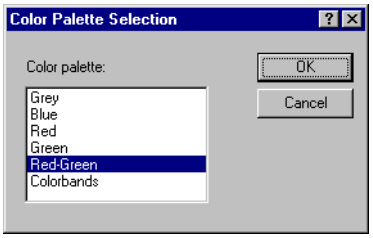


Figure 7.8: Dialog Color Palette Selection

2. Click the color palette required. Default is the color palette red-green.
3. Click OK.

7.4 Selecting the Data Set Displayed

To display a certain data set, select in the following order:

1. time signal or type of spectrum
2. channel or combination of channels
3. signal or function
4. display type in which the signal or function is shown
5. Only in presentation windows of data you have collected in FFT mode: frequency band.

The selection depends on

- the measurement mode
- averaging, trigger and reference signal; see your theory manual on this as well,
- which displays have already been selected – for example, you can display a frequency response just for frequency spectra and a combination of channels.

You first have to define frequency bands as described in section 8.2.

If an analyzer is part of a presentation window (refer to section 7.3.1), then you select the data set displayed in the presentation window. If possible, the analyzer displays the same data set.

To display a certain data set, proceed as follows:

Domain

1. Click  or select Analyzer > Domain or Presentation > Domain.
2. Select the time signal, a spectrum or the RMS signal in the popup menu.

☞ Measurement modes FFT, Zoom-FFT and MultiFrame: To calculate and display a frequency spectrum, the software needs to acquire a complete sampling period. Therefore, during a continuous measurement, the **spectrum** of the **last complete measurement** is displayed at the same time as the **time signal** of the **current measurement**.

Channel

3. Click  or select Analyzer > Channel or Presentation > Channel.
4. Select a channel or a combination of channels in the popup menu.

Signal

5. Click  or select Analyzer > Signal or Presentation > Signal.
6. Select a signal or a function in the popup menu. The abbreviations used are explained in table 7.1.

Table 7.1: Abbreviations for functions

Abbreviation	Function
AP	Autopower
CP	Crosspower
FRF	Frequency Response Function
H1	Frequency Response Function
H2	Frequency Response Function

Display type

7. Click  or select Analyzer > Display Type or Presentation > Display Type.
8. In the popup menu, select the way in which the vibration is shown.

☞ In the display type Magnitude [dB (A)], the magnitude measured is frequency-weighted according to the standard EN 60651 (IEC 651) (A-weighting). The A-weighting approximately describes the acoustic sensitivity of a human at low magnitudes. The frequency-weighting curve is shown in figure 7.9. Outside the range shown, the software frequency-weights as follows: Below 10.0 Hz at -70.4 dB and above 20,000 Hz at -9.3 dB.

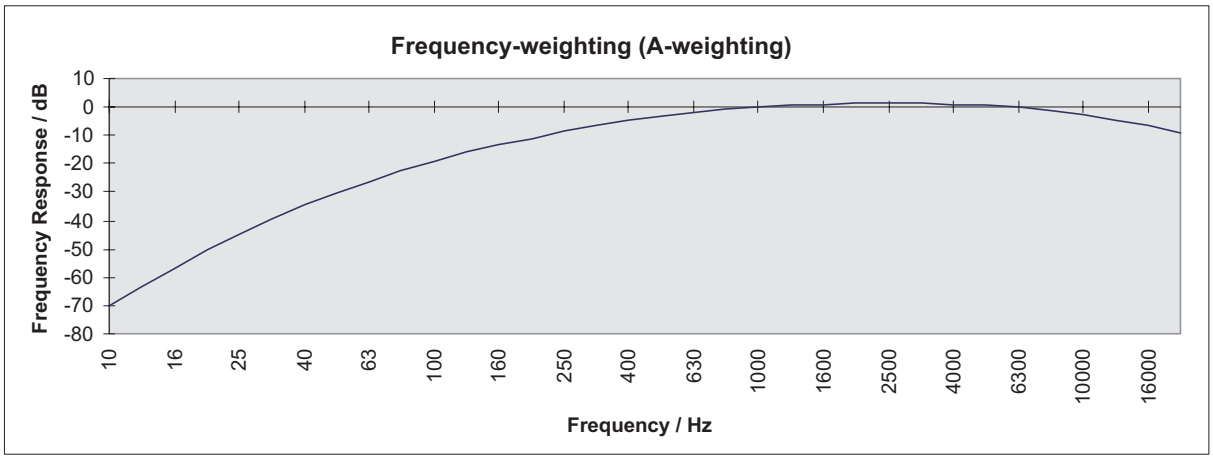
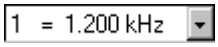


Figure 7.9: Frequency-weighting curve for the display type Magnitude [dB(A)]

☞ In the display type Nyquist, the real part of the magnitude is plotted over the x-axis and the imaginary part over the y-axis.

Frequency band (only in presentation windows)

You have three different ways of selecting the frequency band displayed:

9. Select the frequency band in the tool bar of the presentation window in the list  . If no frequency bands are defined, then the only entry in the list is Entire Bandwidth.

or

10. Press the key PgDn ↓ or PgUp ↑. You will see the next or previous animation frame.

or

11. Either display the Single Point view or the average spectrum. To do so, click  and select Single Scan Point or Average Spectrum.

12. In the tool bar of the analyzer, click  . You see the frequency bands in the analyzer in color.

13. Click the required frequency band in the analyzer. The software shows this frequency band at the top.

In table 7.2 you will find a summary of how to select domain, channel, signal and display type.

Table 7.2: Selecting the data set displayed

Click	Or select Analyzer> or Presentation >	Display of	Example
1. 	Domain	Time signal, spectrum or root mean square	FFT, 1/3 Octave RMS Time
2. 	Channel	Channel or combination of channels	Vib Ref 1 Vib & Ref 1
3. 	Signal	Signal or function	Velocity Acceleration Voltage FRF Displacement / Voltage AP Displacement^2
4. 	Display type	Way in which the signal or function is shown	Amplitude Magnitude [dB] Phase Real & Imag. Nyquist

8 Evaluating Scans

The software offers numerous possibilities to evaluate scans. To do this, first open the scan as described in section 9.1.1. Depending on which measurement mode you used to acquire the data, you will see the results of your measurement in the time or frequency range.

You can see which mode a measurement was carried out in by opening the additional information in the dialog at the bottom of a file.

If you have opened a scan, you will see in the status bar which mode the measurement was carried out in.

If you open a scan generated in FFT mode for the first time in presentation mode, you will see the root mean square of the magnitude over the whole bandwidth. You can apply certain evaluation functions directly to the root mean square. See section 8.1 and section 8.3 on this. You can also evaluate sections of the measured spectrum – for example, in the proximity of resonance or 1/3 octave bands. To do this, you first have to define these frequency bands as described in section 8.2. At the peak frequencies of the frequency bands you can also animate the vibration, see section 8.4 on this.

In FastScan mode you only measure a certain individual frequency. Correspondingly you will only see a surface-like presentation of the data of this frequency if you open the scan in presentation mode.

If you open a scan generated in time mode for the first time in presentation mode, you will see the root mean square of the magnitude over the whole sample time. With the time data animation you can present the momentary magnitude of the sample time for selected ranges.

8.1 Processing Data

You can prepare a scan for display with the following commands in the menu Presentation:

- Invalidate Selected Point
- Invalidate Not Optimal Points
- Interpolate Data
- Filter Data.

You can undo all commands.

Invalidate selected point

To single scan points – for example those with a bad signal/to/noise ratio – you can allocate the status Invalidate Selected Point. The software does not show any measurement data for these scan points. The averaged spectrum over all sample points is corrected correspondingly.

To allocate the status Invalidate Selected Point to a point, proceed as follows:

1. Set a cursor at the scan point. See section 7.3.3 on this.

2. Press the key combination Ctrl+space bar or select Presentation > Invalidate Selected Point.

> **Caution!**
>
> When saving the average spectrum the uncorrected original data is saved (refer also to section 9.2.2). The same applies to access via Polytec File Access.

3. To return to the initial status of the scan point, press Ctrl+space bar again or select Presentation > Invalidate Selected Point.

Invalidate not optimal Points

You can allocate the status Invalidate Selected Point to all scan points which do not have the status Optimal. This command is only active if

- Signal Enhancement was active while scanning (refer to section 5.5.1)
- not all scan points have got the status Optimal.

To do this, proceed as follows:

1. Select Presentation > Invalidate Not Optimal Points.

2. To return to the initial status of the scan points, select Presentation > Invalidate Not Optimal Points again.

☞ If you had changed the status of single scan points with the command Invalidate Selected Point (see above), this scan point is also allocated its original status.

Interpolate data

For scan points with the status Overrange or Invalidated, the software does not display any measurement data. You can interpolate and display data from neighboring scan points for these as well as for not optimal scan points.

☞ The software allocates the status Overrange while scanning. You can allocate the status Invalidated manually (see above).

To interpolate, proceed as follows:

1. Select Presentation > Interpolate Data. If you would like to interpolate not optimal points as well, then in addition to that, in the menu Analysis > select Invalidate Not Optimal Points.

2. To undo the command, select Presentation > Interpolate Data again.

> **Caution!**
>
> If interpolate is active, then in ASCII export only the interpolated data is saved. On exporting as a Universal File or on accessing via Polytec File Access you will obtain the original data.

Filter data

You can smooth the display of scans with a filter. Filtering is done on two levels. First of all median averaging and then center value filtering of neighboring scan points is carried out.

To smooth the display, proceed as follows:

1. Select Presentation > Filter Data.

2. To undo the command, select Presentation > Filter Data again.

Caution!

If Filter Data is active, then in ASCII export only the smoothened data is saved. On exporting as a Universal File or on accessing via Polytec File Access you will obtain the original data.

☞ The commands Interpolate Data and Filter Data represent measures taken to rid measurements of interference signals and noise levels. When defining the quantity of scan points, already ensure that there is a sufficient scan point density, depending on the maximum vibration frequency in the object to be measured.
Careful scan point definition also helps avoiding interference signals caused by unsuitable selection of scan points, e.g. glancing incidence at the edge of an object or undesirable measurements in object openings.

8.2 Defining Frequency Bands (not in Time Mode)

You can evaluate sections of the measured spectrum, so-called frequency bands. To do this, in the frequency band definition, you first enter the start and end frequencies of the frequency bands. The software determines the peak frequency and the bandwidth of the frequency bands.

Requirements

You can define frequency bands for scans

- whose file is not write-protected and
- which contain a complete spectrum.

Scans without a complete spectrum are those

- in which only frequency bands are saved (refer to section 9.2.1)
- which were measured in the measurement mode FastScan
- which were measured in the measurement mode Time.

Save

The software automatically saves the frequency band definition in the scan and loads it when you reopen the scan. You can also save frequency band definitions separately as an ASCII file and load them into several scans. See section 9.2.4 and section 9.1.2 on this.

Define

To define frequency bands, proceed as follows:

1. Either display the Single Scan Point view or the Average Spectrum. To do so, click  and select Single Scan Point or Average Spectrum.

2. Open the frequency band definition. To do so, click  or select Setup > Frequency Bands. The dialog Frequency Band Definition appears.

3. Enter the start and end frequencies of the frequency bands as described below. You have two possibilities to do this: directly in the analyzer or in the dialog Frequency Band Definition. The software determines the peak frequencies of the frequency bands for the data set displayed in the view set.

4. You can change between different views or display other data sets without closing the frequency band definition. The software does not determine the peak frequencies again at that point. To determine the peak frequencies again, in the dialog Frequency Band Definition click  .

5. If the software is to calculate 1/3 octave bands, then in the dialog Frequency Band Definition click  .

6. When you have defined all frequency bands, close the frequency band definition. To do so, click  again or select Setup > Frequency bands. The software calculates the root mean square of the frequency bands and saves them in the scan.

Analyzer

To enter start and end frequency of a frequency band directly in the analyzer, proceed as follows:

1. Click  in the analyzer.

2. Point at the diagram (not at the edge of the diagram).

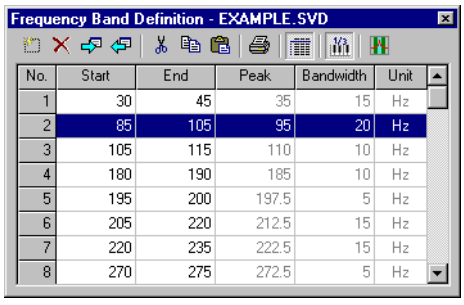
3. Point at the start point of the required frequency band.

4. Press the mouse button and then drag right or left to the end point. The software displays the frequency band in color and enters it in the dialog Frequency Band Definition (see below).

Dialog

To enter start and end frequency of a frequency band in the dialog Frequency Band Definition, proceed as follows:

1. The dialog Frequency Band Definition consists of a tool bar and a table, the frequency band editor. If you can not see the table, display it. To do so, in the tool bar, click  .



No.	Start	End	Peak	Bandwidth	Unit
1	30	45	35	15	Hz
2	85	105	95	20	Hz
3	105	115	110	10	Hz
4	180	190	185	10	Hz
5	195	200	197.5	5	Hz
6	205	220	212.5	15	Hz
7	220	235	222.5	15	Hz
8	270	275	272.5	5	Hz

Figure 8.1: Dialog Frequency Band Definition

2. In the tool bar, click  . A new line is added to the bottom of the table.

3. Enter the start frequency in hertz. The start and end frequency must be on FFT lines. The software replaces the value entered by the next possible one.

4. Press Tab or click the corresponding cell in the column End.
5. Enter the end frequency in hertz.
6. Press Enter. The software enters the peak frequency and the bandwidth in the frequency band editor and displays the frequency band in the analyzer in color.

Select

To delete or copy frequency bands from the definition, you have to select them first. To do this, proceed as follows:

1. Click the frequency band in the analyzer or its line number in the frequency band editor.
2. To select further frequency bands, press the Control key and click the number of the line in the frequency band editor.
3. To select all frequency bands or to undo a selection, click the top left of the table.

Delete

To delete frequency bands from a definition, proceed as follows:

1. Select the frequency bands as described above.
2. Press the Del key or in the frequency band editor, click  .

Copy

You can copy a frequency band definition via the clipboard into another scan. To do this, proceed as follows:

1. Select the frequency bands as described above.
2. In the dialog Frequency Band Definition, click  .
3. Close the current frequency band definition. To do so, in the presentation window click  or select Setup > Frequency bands.
4. Open the scan into which you want to copy the frequency bands.
5. Open the frequency band definition again. To do so, click  or select Setup > Frequency Bands.
6. In the dialog Frequency Band Definition, click  .

8.3 Displaying Profiles

You can display profiles of the data. To do this, you first draw profile sections on the object. You will see the profiles at the bottom in the analyzer. The software automatically saves the profile sections in the scan and loads them when you reopen the scan.

Drawing profile section

To draw a profile section on the object, proceed as follows:

1. Display the Profile view. To do so, click  and select Profile in the popup menu.
2. Either display the view style 2D, Isolines or Scan Points. To do so, click  and select the view style in the popup menu.
3. Click  .
4. Point at the object and click the start point of the profile section. The cursor becomes a  .
5. Click as many other vertex points as you like. You can see the profile at the bottom in the analyzer.
6. To define the end point of the profile section, double-click.

Select profile sections

To edit profile sections, you have to select them first. To do this, proceed as follows:

1. Click  .
2. Point at a profile section. The cursor becomes a  .
3. Click. The software marks the vertex points of the profile section with yellow handles.
4. To select further profile sections, click them while holding the shift key pressed.
5. To select all profile sections, press the key combination Ctrl+A or select Edit > Select All.
6. To remove a profile section from the selection, click it while holding the shift key pressed. The yellow handles disappear.

Move profile sections

To move profile sections, proceed as follows:

1. Select the profile sections as described above.
2. Point at one of the lines (not at a vertex point). The cursor becomes a  .
3. Press the mouse button and drag in the direction required.

Move vertex points

You can change the shape of profile sections by moving the individual vertex points. To do this, proceed as follows:

1. Select a profile section as described above.
2. Point at a vertex point. The cursor becomes a  .
3. Press the mouse button and drag in the direction required.

Duplicate profile sections

To duplicate profile sections, proceed as follows:

1. Select the profile sections as described above.
2. Click  or select Edit > Copy. The profile sections are copied onto the clipboard.
3. Click  or select Edit > Paste.

Delete profile sections

To delete profile sections, proceed as follows:

1. Select the profile sections as described above.
2. Press the Del key or select Edit > Delete.

Undo and redo

You can undo up to 20 operations and then redo them. To do so, click  and  or select Edit > Undo and Edit > Redo.

Animate

You can animate the profiles' vibrations. See section 8.4 on this.

8.4 Animating Vibrations

If you have measured phases while scanning, then you can animate the vibration.

8.4.1 Data Acquisition Mode FFT, Zoom FFT, Fast Scan, MultiFrame

If you have carried out a measurement in one of the above mentioned modes, you can set how many images per animation cycle are shown.

☞ You measure phases if you are using a reference signal or are triggering.

To animate a vibration, proceed as follows:

Preparation

1. Define frequency bands for the scan. See section 8.2 on this.
2. Display the frequency band whose peak frequency you want to animate.
3. Display the FFT spectrum. To do so, click  and select FFT in the popup menu.

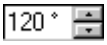
4. Display a complex signal.

☞ For complex signals, the display type Phase is available.

5. It is useful to display the 3D view style. To do so, click  and select 3D in the popup menu. You can also animate in other view styles, however not in the Status view style.

Start

There are two ways of starting animations:

6. To start the animation in the active presentation window, click  or select Animation > Start. The software shows the signal in the display type Instant Value and animates the vibration. You can see the instantaneous angle in the legend in the field Angle  . If the legend is not visible on the right of the object, select Presentation > Legend to display it.

or

To start animations simultaneously in all open presentation windows, select Animation > Start All.

Velocity

7. To increase or decrease the speed of the animations, select Animation > Increase Speed or Animation > Decrease Speed.

Stop

There are two ways of stopping animations:

8. To stop the animation in the active presentation window, click  again or select Animation > Stop.

or

To stop all current animations, select Animation > Stop All.

Step forward, step backward

You can manually make animations run forwards and backwards. To do so, click  and  or select Animation > Step Forward and Animation > Step Backward. You will see the next or previous animation frame respectively.

Frames per animation cycle

If you have carried out the measurement in FFT mode, you can set the number of frames per animation cycle in the dialog Preferences. To open the dialog, select Setup > Preferences. Then show the page Display.

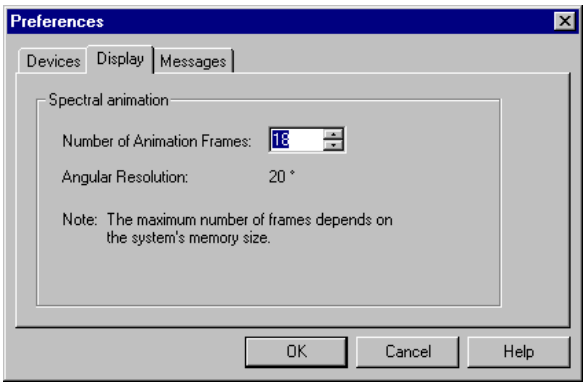


Figure 8.2: Page Display in the dialog Preferences

Number of Animation Frames: Here you enter how many frames per animation cycle you want to see. The minimum number is 8, the maximum number depends on the capacity of your computer's memory. The default setting is 18.

Angular Resolution: The software calculates the Phase angles between the data of two frames and displays it here.

Save

You can save animation in multimedia format *.avi. See section 9.2.5 on this.

8.4.2 Data Acquisition Mode Time

If you have carried out the measurement in Time mode, you will see the animation in the time domain. To animate a vibration, proceed as follows:

Preparation

1. Display the time domain. To do so, click  and select Time in the popup menu.

   Depending on the size of the scan file and capacity of your computer's memory, the following dialog may appear:

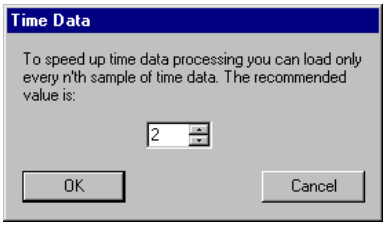


Figure 8.3: Dialog for reducing the time data step size

   It is recommended that you use the suggested value or enter a higher one. Click OK. Subsequently the time data will be resorted and you can start the animation in the time domain.

☞ If you set a smaller step size that the one suggested, the subsequent resorting of the data can take quite some time!

To return to RMS mode, click Abort.

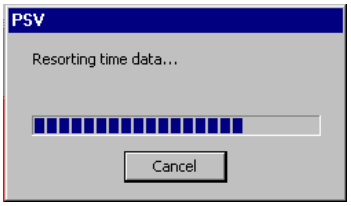


Figure 8.4: Resorting of time data

☞ If you click Abort during resorting, the dialog to reduce the time data step size opens (refer to section 8.3).

2. It is useful to display the 3D view style. To do so, click and select 3D in the popup menu. You can also animate in other view styles, however not in the Status view style.

3. Select the section within the sample time in which the data is to be animated. There are two ways to do this:

   Enter the start and end time directly into the legend to the right of the presentation window. If the legend is not visible, select Presentation > Legend to display it.

or

   Open an analyzer. To do so, click and choose Single Scan Point.

   Click and with a further mouse click, set the starting point directly in the diagram. (As initially the starting point is the same as the current point in time, it is marked by a red line.)

   Hold the mouse button pressed and drag it to the right. A second black line will appear which marks the end point. If the line is in the right place, release the mouse button.

Start

There are two ways of starting animations:

4. To start the animation in the active presentation window, click or select Animation > Start. You can see the defined time frame and the current status of the animation and also the step size in the legend.

or

   To start animations simultaneously in all open presentation windows, select Animation > Start All.

You can let the animation run forwards or backwards and stop it at any time, change its speed or save it as described in the following:

Velocity

5. To increase or decrease the speed of the animations, select Animation > Increase Speed or Animation > Decrease Speed.

Stop

There are two ways of stopping animations:

6. To stop the animation in the active presentation window, click  again or select Animation > Stop.

or

To stop all current animations, select Animation > Stop All.

Step forward, step backward

You can manually make animations run forwards and backwards. To do so, click  and  or select Animation > Step Forward and Animation > Step Backward. You will see the next or previous animation frame respectively.

Save

You can save animation in multimedia format *.avi. See section 9.2.5 on this.

9 Managing Data

9.1 Opening Files

The software can open the following files:

- scans in native binary format Scan Data *.svd
- single point measurements in native binary format Single Point Data *.svd
- frequency band definitions in ASCII format
- settings in native binary format *.set.

9.1.1 Opening Scans and Single Point Measurements

In presentation mode you can open scans and single point measurements in the native binary formats Scan Data *.svd and Single Point Data *.pvd. There are two ways of doing this:

- the dialog Open
- the list of most recent files in the menu File.

You can also open the last scan measured if you change to presentation mode. To do this, follow the instructions in the software.

Dialog

To open a scan or a single point measurement, proceed as follows:

1. Click  or select File > Open. The dialog Open appears.

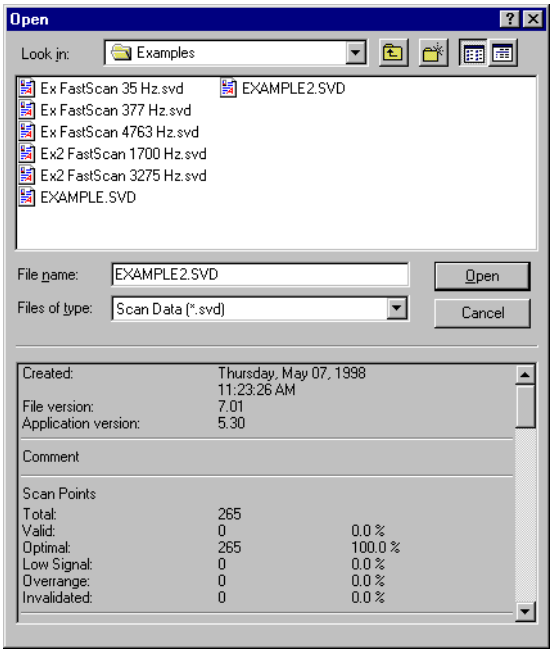


Figure 9.1: Dialog Open

2. Navigate to the saving location and select the file type and name. To make it easier to identify the files, look at meta-information at the bottom in the dialog – for example, the data acquisition board and the settings with which the data was measured.

3. Click Open. If you open a scan, a presentation window appears. If you open a single point measurement, an analyzer appears.

File list In presentation mode, in the menu File at the bottom there is a list with up to nine files which you last opened. To open one of these files again, click it in the list.

9.1.2 Loading Frequency Band Definitions

In presentation mode you can load frequency band definitions in ASCII format.

☞ When you load a frequency band definition, the existing one is overwritten.

To open a measurement, proceed as follows:

1. Open the scan you want to load a frequency band definition for. See section 9.1.1 on this.

2. Open the frequency band definition. To do so, click  . The dialog Frequency Band Definition appears.

3. In the dialog Frequency Band Definition, click  . The dialog Open appears.

4. Navigate to the saving location and select the file.

5. Click Open.

6. Close the frequency band definition. To do so, click  again.

9.1.3 Loading and Importing Settings

In acquisition mode you can load settings from files in native binary format *.set and import them from scans. These are:

- the settings for data acquisition in the dialog Acquisition Settings  (only possible for identical data acquisition boards)
- the scan point definition 
- the settings of the optics  (alignment, only PSV 300: zoom and focus of the video camera)
- the window configuration  (not possible from scans).

You can also import scan point definitions from files in the formats *.apss, *.apsp and *.apsa.

Loading

To load settings, proceed as follows:

1. Click  or select Setup > Settings Manager. The dialog Settings Manager appears. Every folder you can see in the dialog corresponds to a setting file in the subdirectory Settings of the software's installation directory.

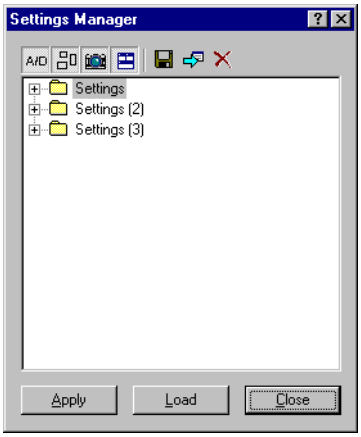


Figure 9.2: Dialog Settings Manager

2. At the top left, activate the icons for the settings you want to load (see above). To (de)activate an icon, click it.
3. Below, click the file you want to load the settings from.
4. Click Load. The settings are active immediately.

Import

You can import settings from scans as of file format 7.0. Scan point definitions can also be imported from files in the format *.apss, *.apsp and *.apsa, which you saved with previous versions of the software. During import, the software generates a setting file in native binary format *.set.

To import settings, proceed as follows:

1. Click  or select Setup > Settings Manager. The dialog Settings Manager appears.
2. At the top left, activate the icons for the settings you want to import (see above). To (de)activate an icon, click it.
3. Click  . The dialog Open appears.
4. Navigate to the saving location and select the file type and name.
5. Click Open. The software creates a settings file in the subdirectory Settings of the software's installation directory. You can see this file as a new folder in the dialog Settings Manager.
6. If you want to load the imported settings immediately, click Apply or Load.

9.2 Saving and Exporting

The software provides the following ways of saving and exchanging data:

- save scans in native binary format Scan Data *.svd
- save single point measurements and average spectrum in the native binary format Single Point Data *.pvd
- export measurements in Universal File Format *.uff (as an option)
- save frequency band definitions in ASCII format
- save animations in multimedia format *.avi
- save settings in native binary format *.set
- export measurement data in ASCII format
- save windows in various graphics formats
- copy windows as graphics onto the clipboard.

9.2.1 Saving Scans

Complete scan

When you start a scan, the software automatically creates a file for the scan and prompts you to save. See section 5.2 on this. When scanning, the software saves the following data in native binary format Scan Data *.svd:

- the settings (refer also to section 9.2.6)
- the spectra at all completely measured scan points (single point spectra)
- the average spectrum of all scan points
- the root mean square of the magnitude over the entire bandwidth.

Even if you abort a scan, the software has saved all scan points which had already been measured.

In presentation mode you can define frequency bands for a scan (refer to section 8.2) and draw profile sections (refer to section 8.3). The software automatically saves the frequency band definitions and profile sections in the scan.

Frequency bands

In presentation mode often only certain sections of the frequency range measured are of interest. For these sections you usually define frequency bands and then evaluate only those frequency bands. You can then save this scan without the single point spectra, as the frequency bands contain all relevant information. This means you can save a lot of disc space. The scan is then subject to certain limitations:

- You can not define any other frequency bands.
- The View Single Scan Point is not available.

To save a scan without single point spectra, proceed as follows:

1. Open the scan you want to save. See section 9.1.1 on this.
2. Define the frequency bands which you want to evaluate. See section 8.2 on this.
3. Select File > Save Bands. The dialog Save As appears.

4. Navigate to the saving location and enter the file name. The software suggests the file name *scanname_b*. However, you can also select another file name.

5. Click OK.

9.2.2 Saving Single Point Measurement and Average Spectra

In acquisition mode you can save the measurement data in the active analyzer in the native binary format Single Point Data *.pvd. In the analysis you can save the single scan point or the average spectrum displayed from a scan. Please note that the Average Spectrum saved corresponds to the original data and not to the spectrum created by removing invalidated single points (refer also to section 8.1). In doing so, the software saves the following data:

- time signals
- spectra
- parameters for data acquisition and evaluation.

To save, proceed as follows:

1. Select File > Save Analyzer. The dialog Save As appears.

Figure 9.3: Dialog Save As

2. Navigate to the saving location and enter the file name.

3. At the bottom in the dialog you can enter meta-information to give the file more precise properties. This meta-information is part of the file properties which you can also view and edit in Explorer.

4. Click Save.

9.2.3 Exporting Universal Files (as an Option)

In presentation mode you can export scans as a Universal File which is compatible with the modal analysis software from the companies SDRC (MTS), LMS and STAR and Vibrant (ME'Scope). To do this, you need the option Universal File Format. Scan points with the status Not Measured, Overrange or Invalidated are not exported. Data evaluation with Presentation > Filter Data and Presentation > Interpolate Data does not have any effect on export, the measured data is exported.

You can export signals which have been acquired through integration and differentiation from the original signals and also time data from order analysis files.

To export a Universal File, proceed as follows:

1. Open the scan you want to export. See section 9.1.1 on this.
2. Select File > Export Universal File. The dialog Save As appears.
3. Navigate to the saving location and enter the file name.
4. Click Save. The dialog Universal File Export appears.

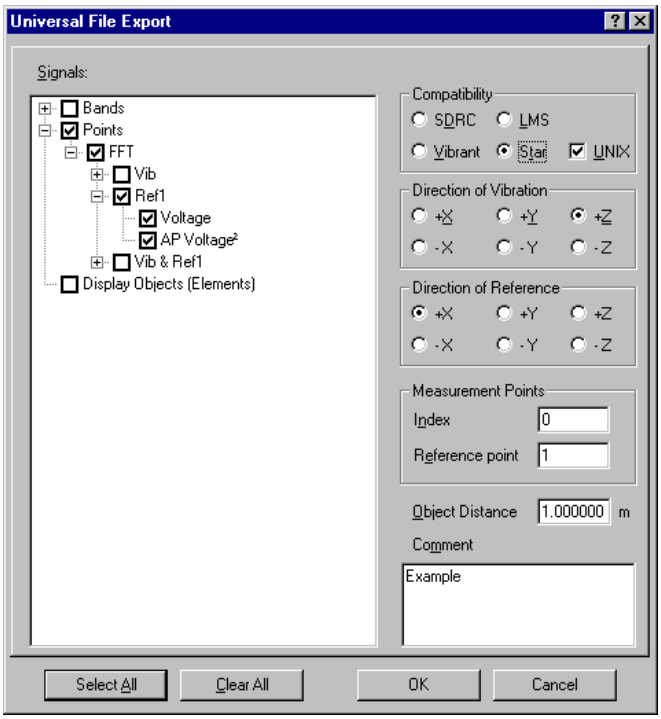


Figure 9.4: Dialog Universal File Export

5. Mark the signals which you want to export and set the parameters for export. You will find information on this in the following.
6. Click OK.

In the dialog Universal File Export you mark the signals which you want to export and set the parameters for export.

Signals On the left you mark the signals which you want to export. The selection available depends on which signals you have measured and whether frequency bands have been defined. If Display Objects (Elements) is ticked, then a graphic representation of the scan points and their connections is exported.

Parameters On the right you set the parameters for export.

Compatibility: Here you select the modal analysis software which the Universal File is to be compatible with. You will find a short description of the data sets used in table 9.1. For detailed information, see the manual for your modal analysis software.

Table 9.1: Data sets which are exported in the Universal File

Data Set	Description	SDRC (MTS)	LMS	STAR	Vibrant
Header	General information, e.g. file name, date, comment	151	151	151	151
Units	Units of the signals and coordinates	164 Double precision	164 Double precision	164 Double precision	164 Double precision
Nodes (Geometry)	Coordinates of the scan points	2411 Double precision	2411 Double precision	15 Single precision	2411 Double precision
Elements (Trace Lines)	Graphic representation of the scan points and their connections	2412 Integer	82 Integer	82 Integer	2412 Integer
Frequency bands	A data set for every signal and every frequency band	55 Single precision	55 Single precision	55 Single precision	55 Single precision
Scan points	A data set for every signal and every scan point	58 Single precision	58 Single precision	58 Single precision	58 Single precision

UNIX: Tick the box if you carry out modal analysis under UNIX. Text lines will then be ended with a line feed to be UNIX-compatible. Special characters in the comment – for example, German umlauts – are not adapted to UNIX.

Direction of Vibration: Here you can select the direction of the vibration in your local coordinate system.

Direction of Reference: Here you can state the direction of the reference excitation at the reference point.

Index Offset: Here you can enter an offset which the software adds to the indices of the scan points. If you want to evaluate several scans together, then you can use this to avoid an index being allocated several times.

Reference Point: If you have measured the reference signal at one of the scan points, then you can enter the index of this reference point here.

Object Distance: Here you enter the stand-off distance in meters, measured from the front panel of the scanning head. In native file format, the coordinates of the scan points are saved as angles, in the Universal File however in Cartesian coordinates. Thus the stand-off distance is required for the conversion. For measurements with the close-up attachment OFV-056-C you have to add 43 mm to the stand-off distance. This corresponds to the optical path length within the close-up attachment.

Comment: Here you can enter comments up to 80 characters long.

9.2.4 Saving Frequency Band Definitions

In presentation mode you can save the current frequency band definition in ASCII format. To do this, proceed as follows:

1. Open the frequency band definition. To do so, click  . The dialog Frequency Band Definition appears.
2. In the dialog Frequency Band Definition, click  . The dialog Save As appears.
3. Navigate to the saving location, enter the file name and select the file extension.
4. Click Save.
5. Close the frequency band definition. To do so, click  again.

9.2.5 Saving Animations

In presentation mode you can save animations in multimedia format *.avi. You will find information on animation in section 8.4.

To save an animation, proceed as follows:

1. Open the scan you want to save an animation for. See section 9.1.1 on this.
2. Set your animation up as described in section 8.4.1 or section 8.4.2.
3. Select File > Save Animation. The dialog Save As appears.

☞ If the presentation window contains an analyzer, you can select whether you would like to save an animation from top, bottom or both diagrams.

4. Navigate to the saving location and enter the file name.
5. Click Save.

9.2.6 Saving Settings

In acquisition mode you can save the current settings in native binary format *.set. These are:

- the settings for data acquisition in the dialog Acquisition Settings
- the scan point definition
- the settings of the optics (alignment, only PSV 300: zoom and focus of the video camera)
- the window configuration .

To save settings, proceed as follows:

1. Click or select Setup > Settings Manager. The dialog Settings Manager appears.

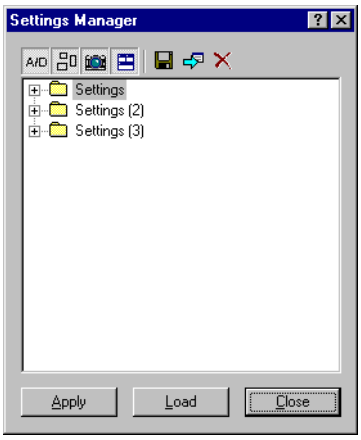


Figure 9.5: Dialog Settings Manager

2. At the top left, activate the icons for the settings you want to save (see above). To (de)activate an icon, click it.

3. In the dialog at the top right, click . The software suggests a file name and saves the file in the subdirectory Settings in the software's installation directory.

4. Click Close.

9.2.7 Export Measurement Data in ASCII Format

You can export the data in the active analyzer or presentation window as an ASCII file. To do this, proceed as follows:

1. Select File > Export ASCII. The dialog Save As appears.
2. Navigate to the saving location, enter the file name and select the file extension.
3. If you are exporting from the presentation window in the view Single Scan Point, Average Spectrum or Profile, then at the bottom of the dialog you see the field Select Data. Select whether you want to export the data sets from the upper or the lower diagram.
4. Click Save.

9.2.8 Saving Graphics

You can save the active window in different graphics formats. The title bar and the frame are not saved. The live video image is saved without scan points.

To save, proceed as follows:

1. Select File > Save Graphics. The dialog Save As appears.
2. Navigate to the saving location, enter the file name and select the file type.
3. If you are saving the graphics from the presentation window in the view Single Scan Point, Average Spectrum or Profile, then at the bottom of the dialog you see the field Select Graphics. Choose whether you want to save the upper or the lower diagram.
4. Click Save.

9.2.9 Copying Graphics onto the Clipboard

You can copy the active window as graphics onto the clipboard. The title bar and the frame are not copied. The live video image is copied without scan points.

To copy, proceed as follows:

1. Click  or select Edit > Copy.
2. If you are copying diagrams from the presentation window in the view Single Scan Point, Average Spectrum or Profile, then the dialog Select Graphics appears. Choose whether you want to copy the upper or the lower diagram.
3. Click OK.

You can then paste the graphics from the clipboard into other applications.

Index

S

T

U

V

W

Z

Polytec Deutschland

POLYTEC GmbH

Polytec-Platz 1-7
D-76337 Waldbronn

Tel.: (07243) 604-0
Fax: (07243) 69944
E-mail: info@polytec.de
Internet: http://www.polytec.de

POLYTEC Physik Instrumente
Vertriebs- und Beratungsbüro Berlin
Schwarzschildstraße 1
D-12489 Berlin

Tel.: (030) 6392-5140
Fax: (030) 6392-5141
E-mail: polytecberlin@polytec.de
Internet: http://www.polytec.de

Polytec International

Germany (D)
POLYTEC GmbH
Polytec-Platz 1-7
D-76337 Waldbronn

Tel.: (+49) 7243 604-0
Fax: (+49) 7243 69944
E-mail: info-vi@polytec.de
Internet: http://www.polytec.com

France (F)
Polytec PI/RMP S.A.
32, rue Délizy
F-93694 Pantin Cédex

Tel.: (+33) 1-48103930
Fax: (+33) 1-48100803
E-mail: info@polytec-pi.fr
Internet: http://www.polytec-pi.fr

Great Britain (GB)
Lambda Photometrics Ltd.
Lambda House, Batford Mill
GB-Harpenden,
Hertfordshire AL5 5BZ

Tel.: (+44) 1582-764334
Fax: (+44) 1582-712084
E-mail: info@lambdaphoto.co.uk
Internet: http://www.lambdaphoto.co.uk

Japan (J)
PI-Polytec KK
Akebono-cho 2-38-5
Tachikawa-shi
J-Tokyo 190

Tel.: (+81) 425-267300
Fax: (+81) 425-267301
E-mail: info@pi-polytec.co.jp

USA East
Polytec PI, Inc.
16 Albert Street
Auburn, MA 01501

Tel.: (+1) 508-832356
Fax: (+1) 508-832050 6
E-mail: info@polytecpi.com
Internet: http://www.polytecpi.com

USA West
Polytec PI, Inc.
1342 Bell Avenue, Suite 3A
Tustin, CA 92780

Tel.: (+1) 714-850 1835
Fax: (+1) 714-850 1831
E-mail: info@polytecpi.com
Internet: http://www.polytecpi.com