MIXED SIGNAL OSZILLOSKOP

USB-PC BASED

TABLE OF CONTENTS

1. General information
2. Product overview
3. Hardware connections
4. Software installation
5. JoyScope user interface
   - 5.1. Organization of the user interface
   - 5.2. Changing the settings
   - 5.3. Further information
   - 5.4. Various displays
   - 5.5. Setting up devices
6. Oscilloscope
7. Recording mode
8. Analysis mode
9. Exporting data
10. Sampling frequency (analog and digital DAQ mode)
11. Use of multiple devices
12. X/Y Scope
13. Spectrum Analyzer
14. Recorder
15. Logic analyzer
    - 15.1. I2C decoder
    - 15.2. UART decoder
    - 15.3. SPI decoder
    - 15.4. 1-Wire decoder
    - 15.5. PWM decoder
16. Function generator
17. Further information
18. Support

1. GENERAL INFORMATION

Dear customer,

thank you for choosing our product. Following, we will show you what you need to bear in mind during commissioning and use.
Should you encounter any unexpected problems during use, please do not hesitate to contact us.

2. PRODUCT OVERVIEW

Thank you for choosing our products. We have prepared this practical manual for you so that you can get to know your new new device and software. If you have purchased a measuring device (JT-ScopeMega50), the package also contains:

- Two 60 MHz probes (Probe 60)
- One logic analyzer DB-25 probe (Probe L2)
- Standard USB cable

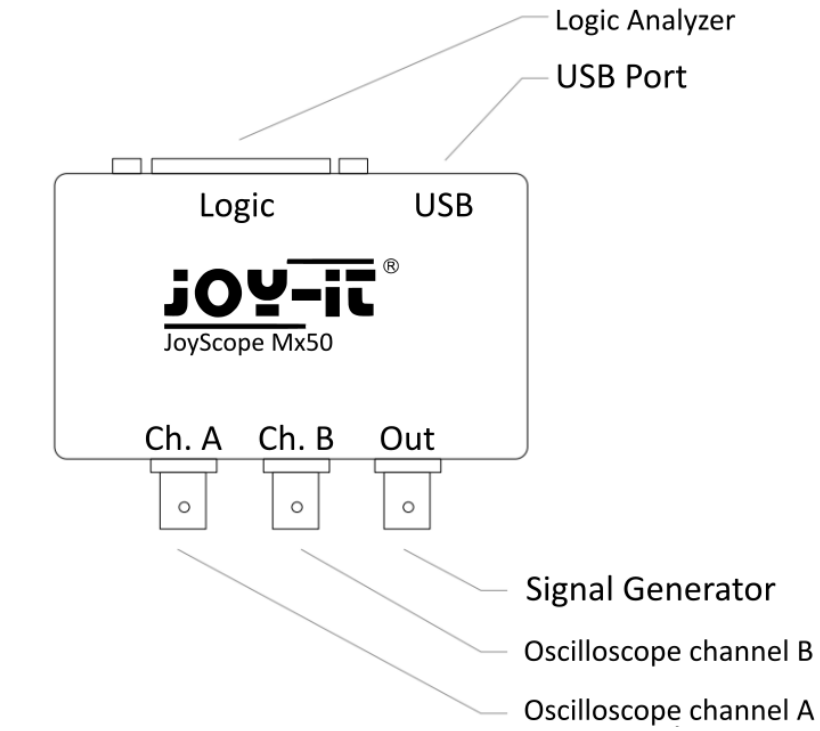
Please note that all devices are supplied with power via the USB port.

We provide free software, drivers and updates for this product, which you can download **here** at any time. You can simply install the software on your PC, connect the USB cable and get started.

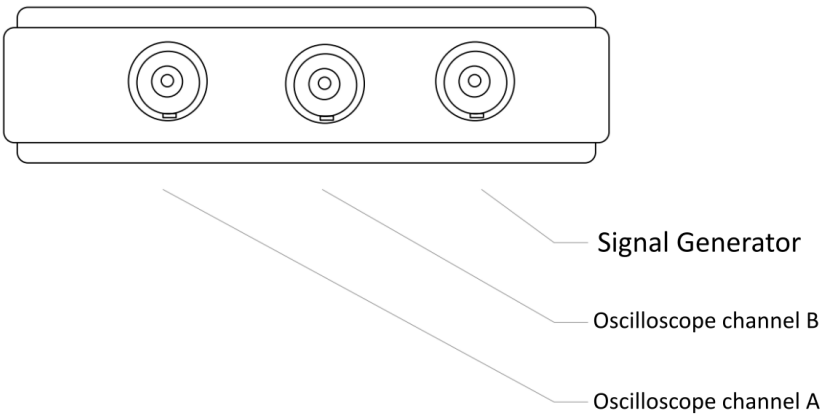
In addition to measuring electrical signals, your new device can also perform a number of other tasks. For example, you can record your measurements, use the device for spectrum analysis, decode most known digital communication protocols and even analyze digital circuits with the easy-to-use 16-channel logic analyzer. JT-ScopeMega50 includes a function generator that can output commonly used waveforms needed for various electronic design and troubleshooting tasks

3. HARDWARE CONNECTIONS

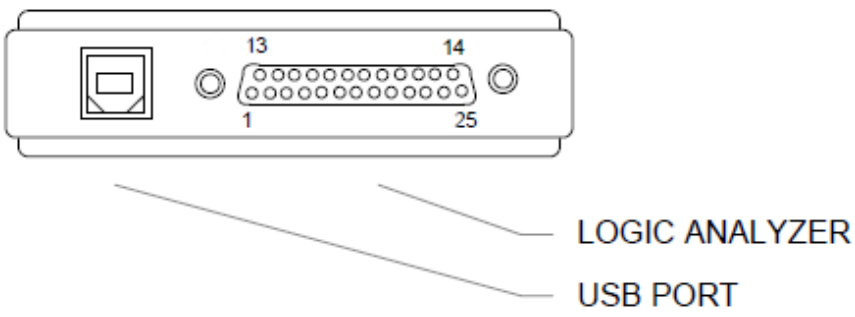
- View from above



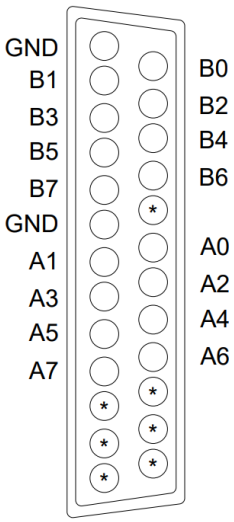
- View from the front



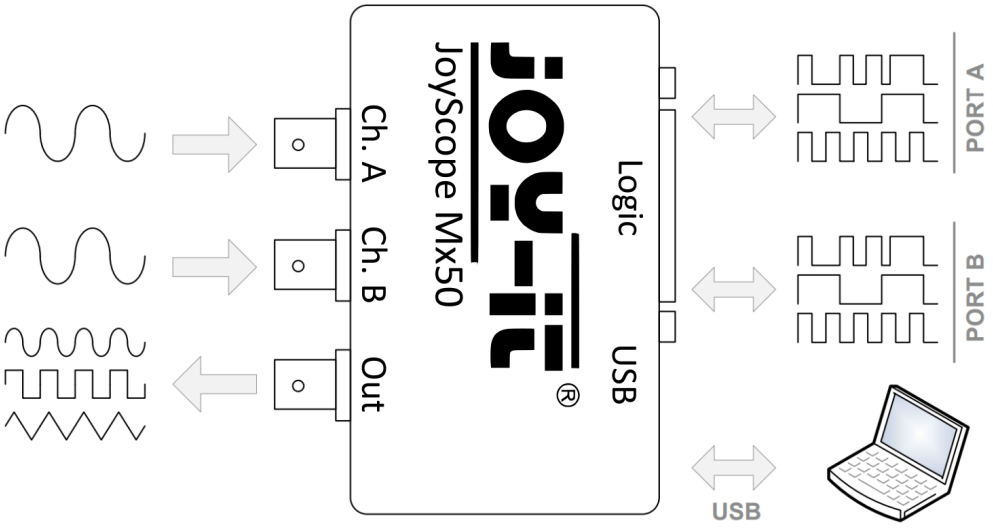
- View from behind



- Logic Analyzer Pinout

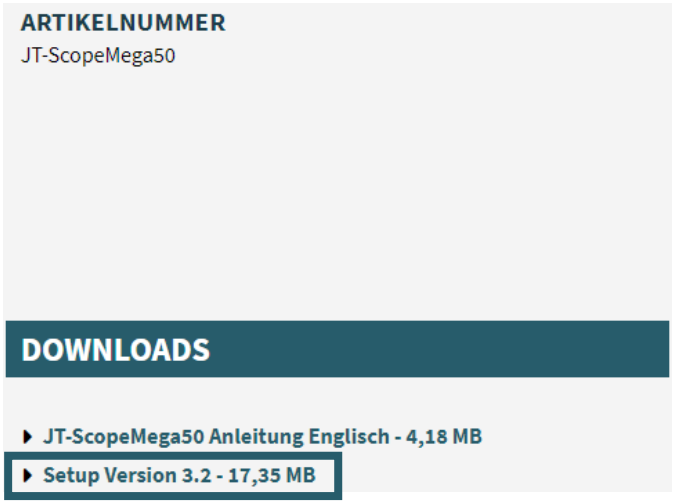


- Inputs and outputs

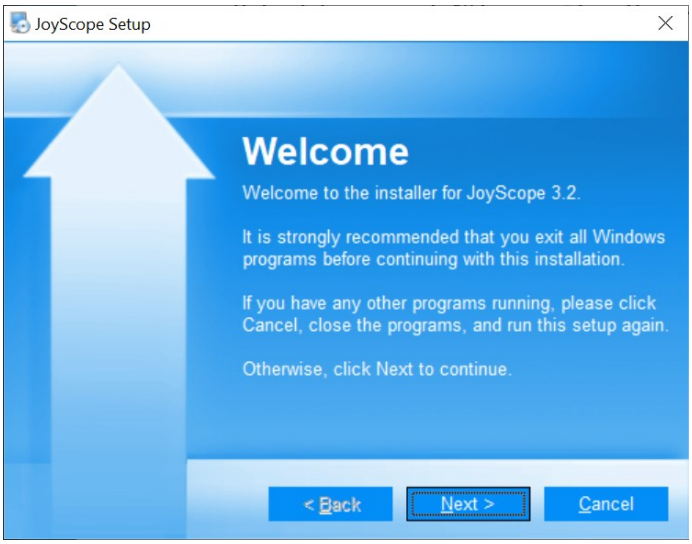


4. SOFTWARE INSTALLATION

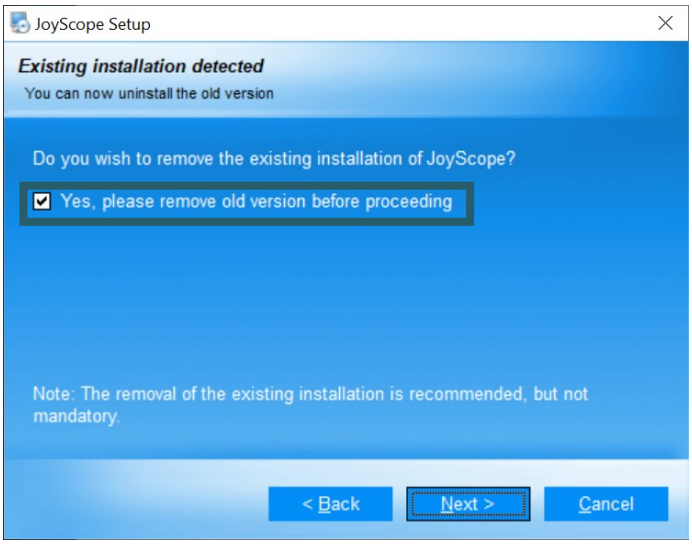
1.
You can find the installation file for the software on our JT-ScopeMega50 product page.



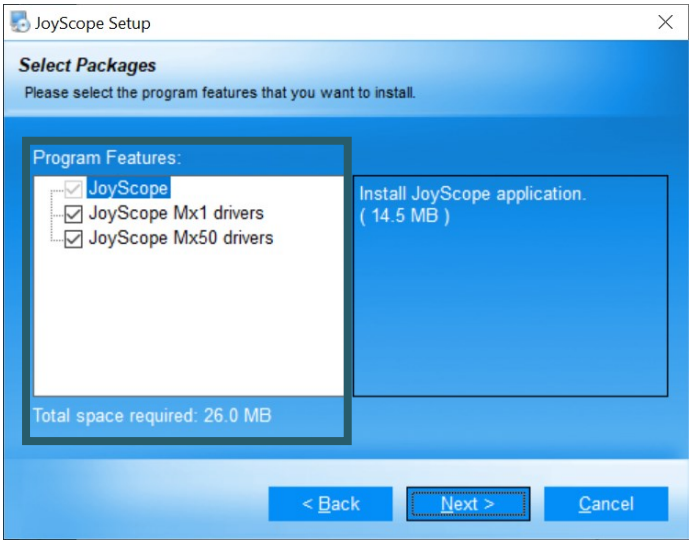
2.
Follow the instructions on the screen and install the JoyScope software.



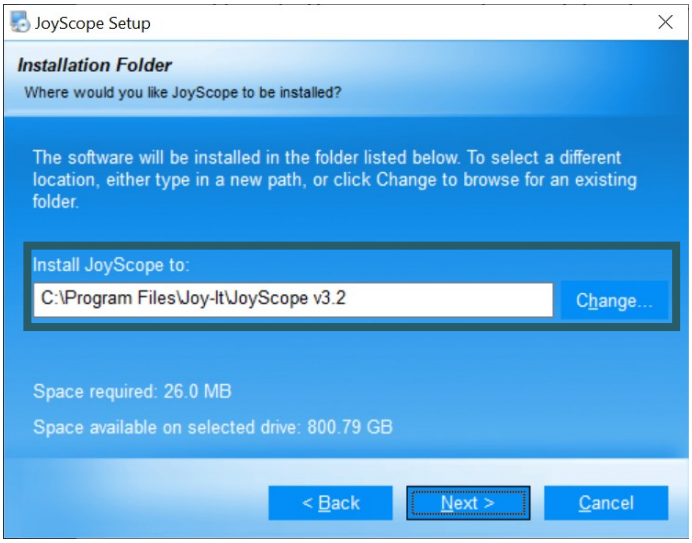
3.
It is recommended that you uninstall the old version before proceeding with the installation. If an older version is installed, please check the corresponding checkbox to remove it.



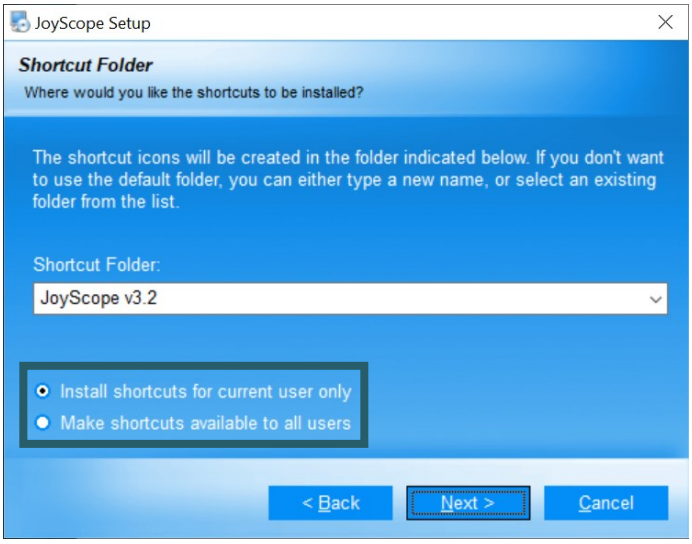
4.
In the package selection window, you can select which program functions you would like to install. Select all packages here.



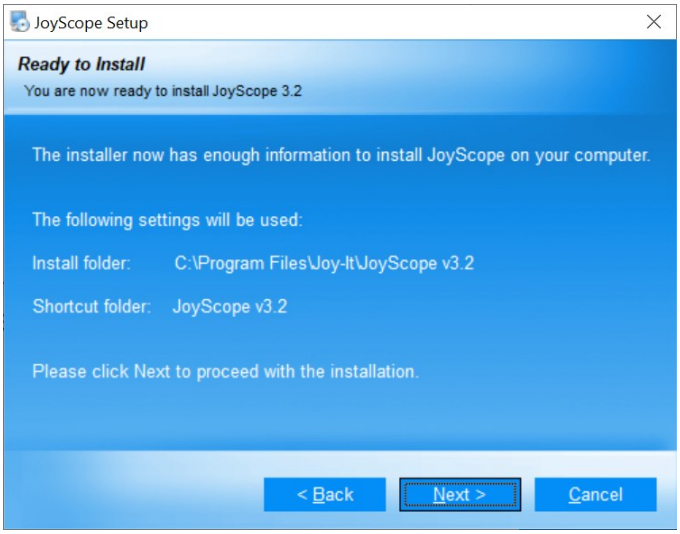
5.
You can now specify the storage location of the software.



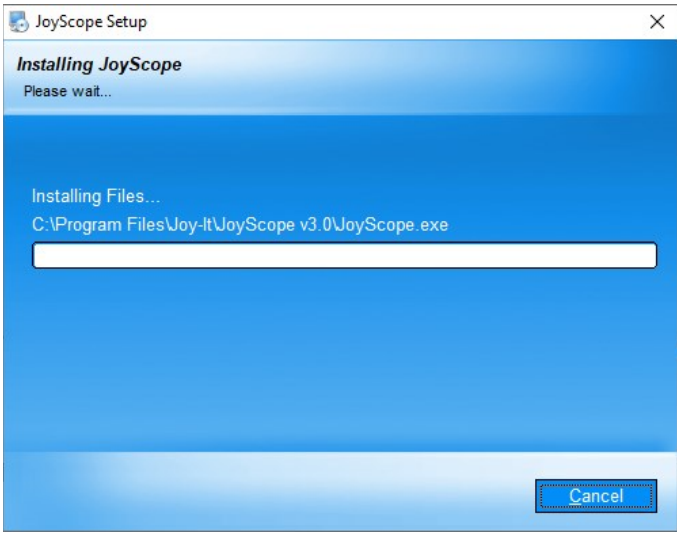
6.
You can install the software for all users or just the currently logged-in user.



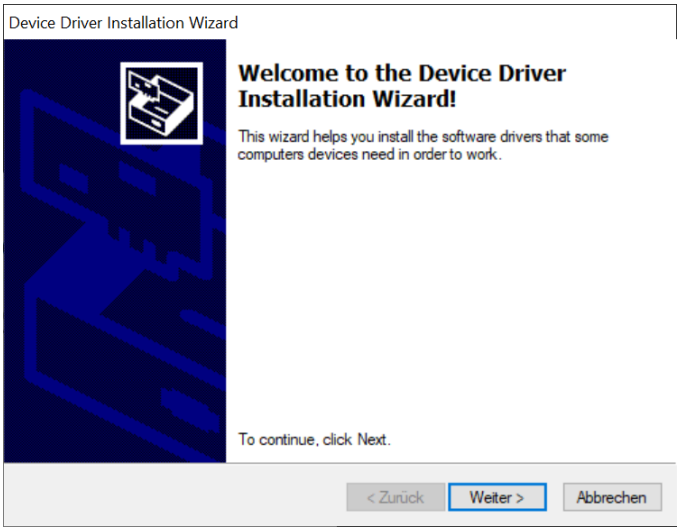
7.
Your installation overview will now be displayed.



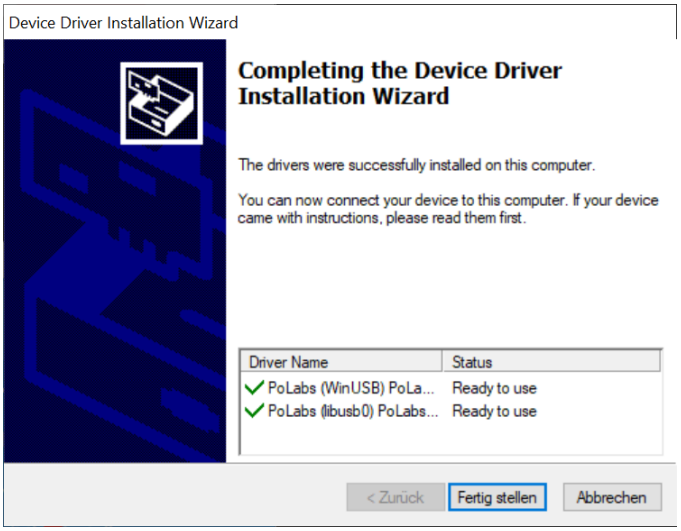
8.
Please wait until the setup program has installed all the necessary files on your computer.



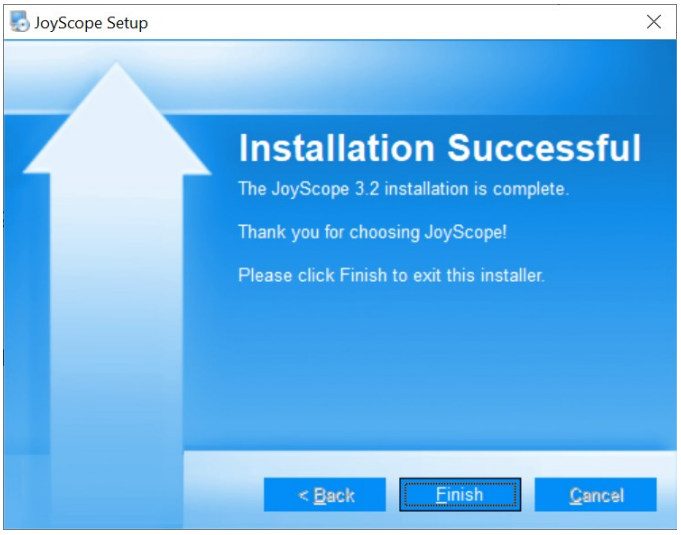
9.
The installation window keeps you informed about the installation of the device driver. up to date.



10.
A green tick means that the device drivers have been successfully installed, and the device is ready for use.



11.
Once the installation is complete, you will receive a notification that the JoyScope software and drivers have been successfully installed. You are now ready to continue and discover how to use the installed software.



5. JOYSCOPE USER INTERFACE

The JoyScope user interface consists of the following elements:

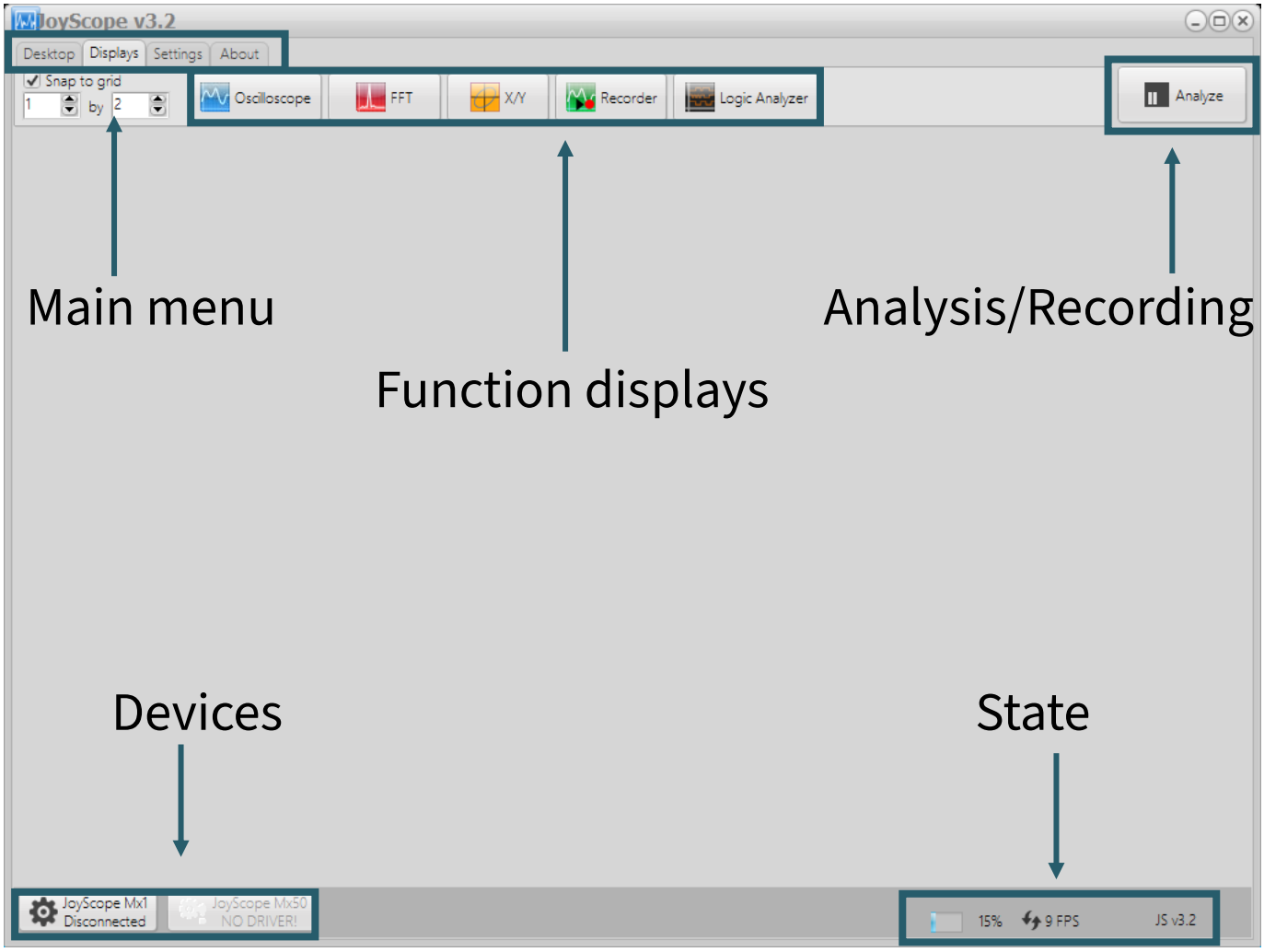
The **main menu** is divided into various tabs.

The **function displays** open separate windows in the software that can display different ScopeMega functions.

You can use the *Analyze* button to switch between **analysis** and **recording** modes. In recording mode, the signal curve is recorded for later analysis. In analysis mode, you can examine the incoming signals.

The **devices** are represented by buttons. Clicking on one of these buttons opens the device control of the selected device.

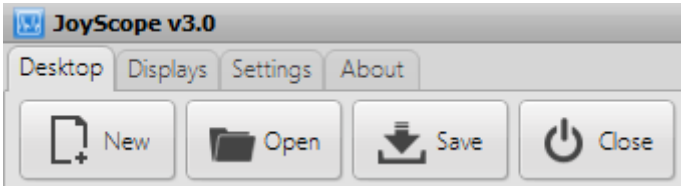
The **status toolbar** shows the current software version, the refresh rate of the displays and the current battery status in case the device has a battery.



5.1 Organization of the user interface

You can manage your workspace on the **Desktop** tab in the **main menu**.

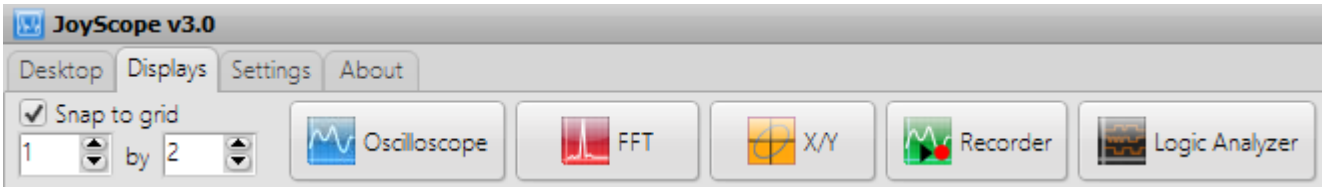
You have the option of arranging the windows opened in the workspace as you wish and save this arrangement. In this way, you do not have to rearrange the windows every time you restart the JoyScope application. The next time you open the JoyScope software, the last saved configuration is automatically loaded.



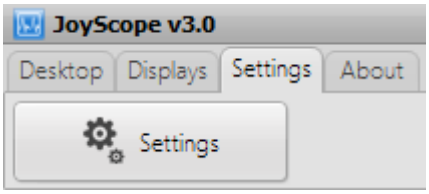
The **Displays** tab contains the following elements:

- Snap to grid
- Oscilloscope
- Frequency analyzer (FFT)
- X/Y Scope
- Recorder
- Logic analyzer

Some of these elements belong to the function displays.



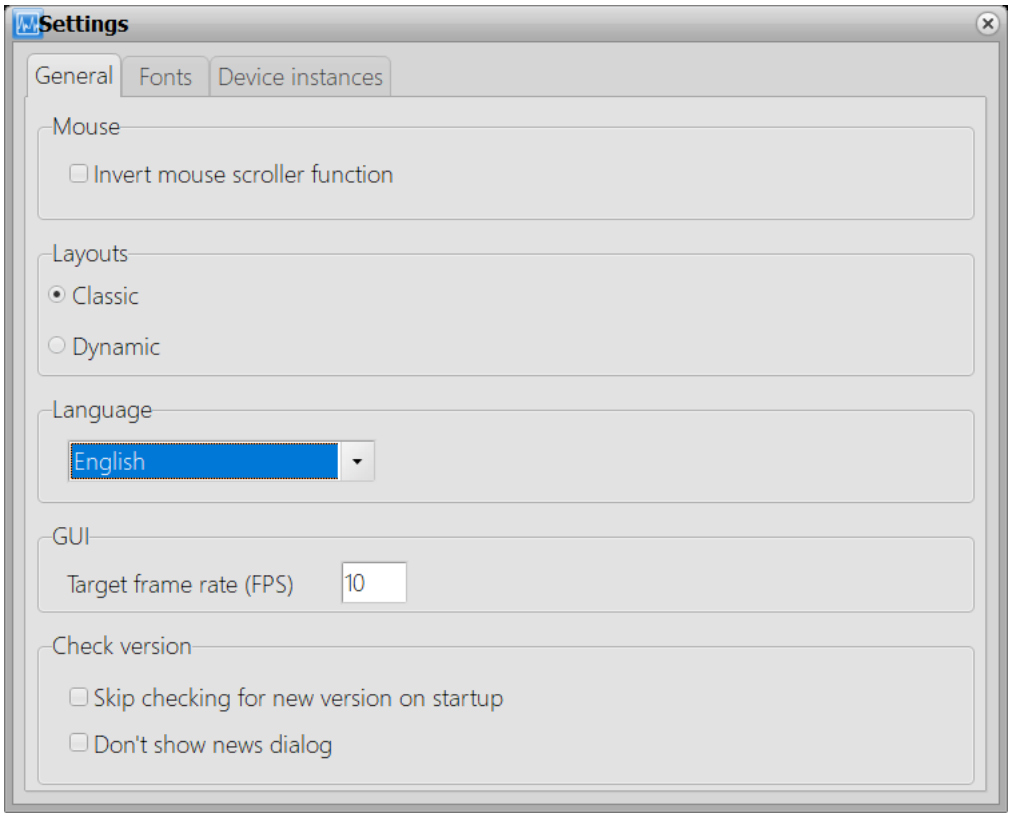
5.2 Changing the settings



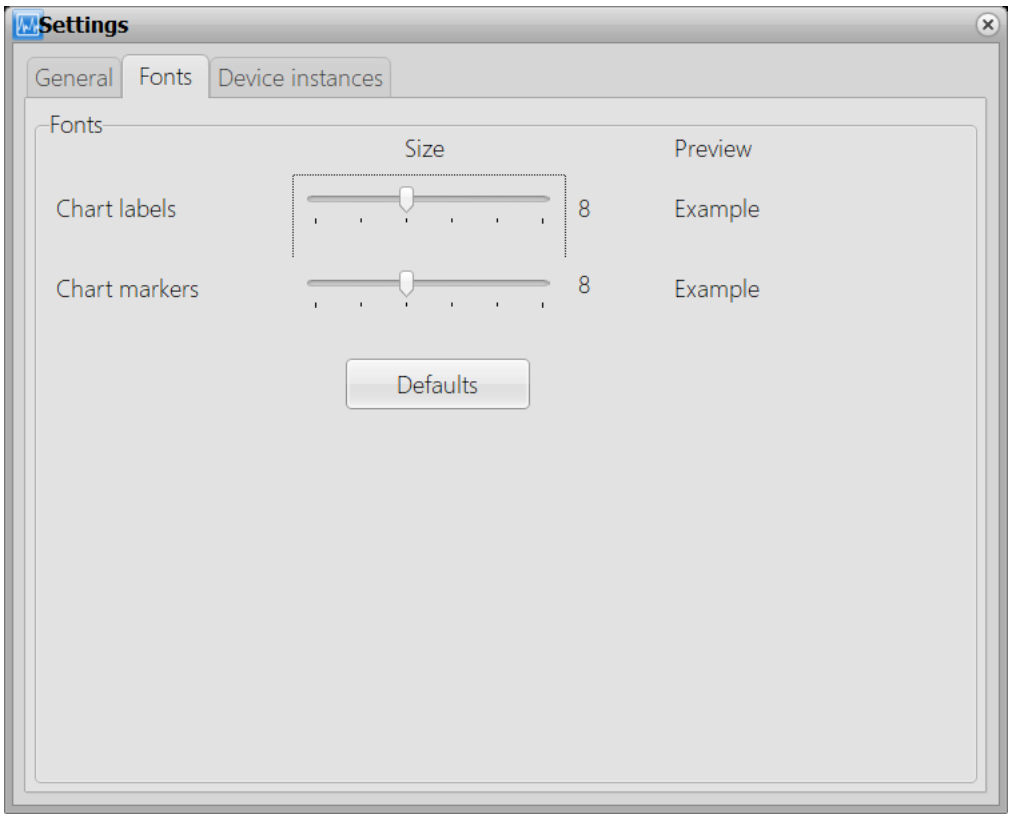
You can do the following in the **Settings** tab/button:

- **Mouse** - Invert mouse wheel function
  (zoom when scrolling forwards/backwards)
- **Layouts** - Choose between classic and dynamic layout
  (In the classic layout, the function displays remain within the application window, whereas in the dynamic layout you have the option of distributing and freely positioning the various displays independently of the application window.)
- **Language** - Changing the user interface language
  (You can use the language settings to select the language of the user interface. To change the language setting, you must then restart the application.)

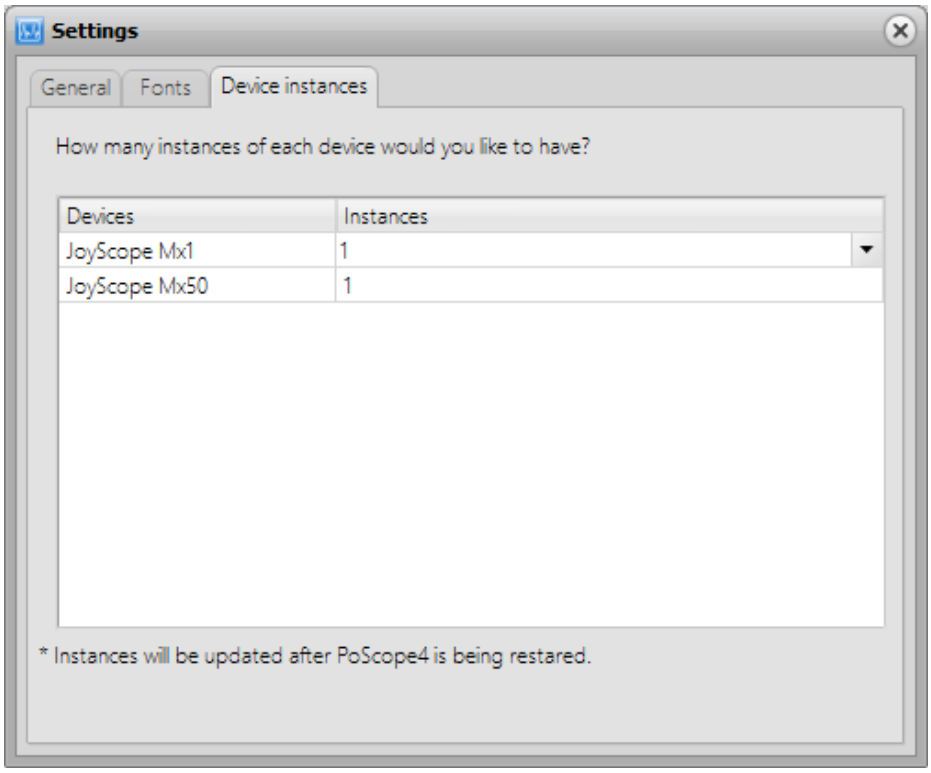
- **GUI** - Setting the frame rate
  (The setting of the target frame rate affects the refresh rate of the data on the screen. The higher the value, the faster the refresh rate (if possible).
- **Check version** - Check version
  (Skips the check for a new version on our website when starting the JoyScope software)



You will find the settings for the fonts on the **Settings** tab. You can configure the size of the fonts in the **Fonts** dialog box.

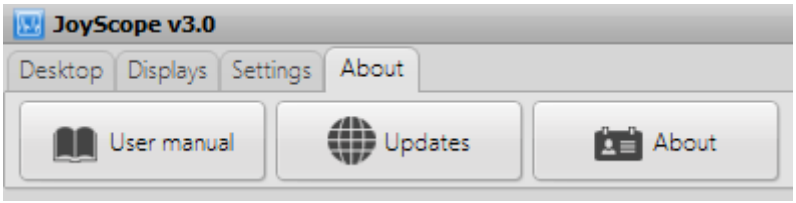


You can specify the number of connected devices on the **Device instances** tab.



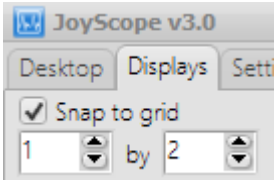
5.3 Further information

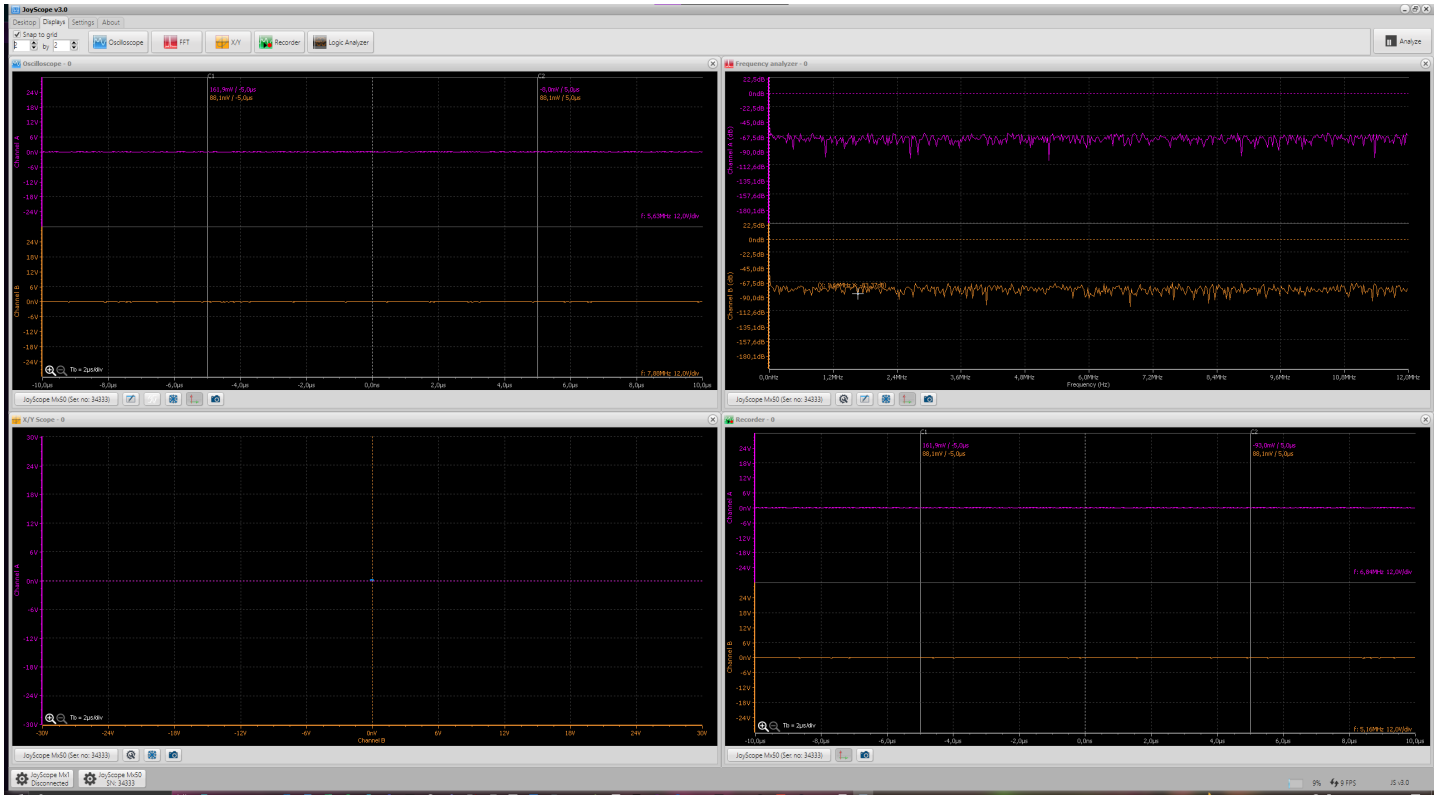
In the **About** tab, under **User manual**, you will find our website and under **About**, you will find the copyrights.



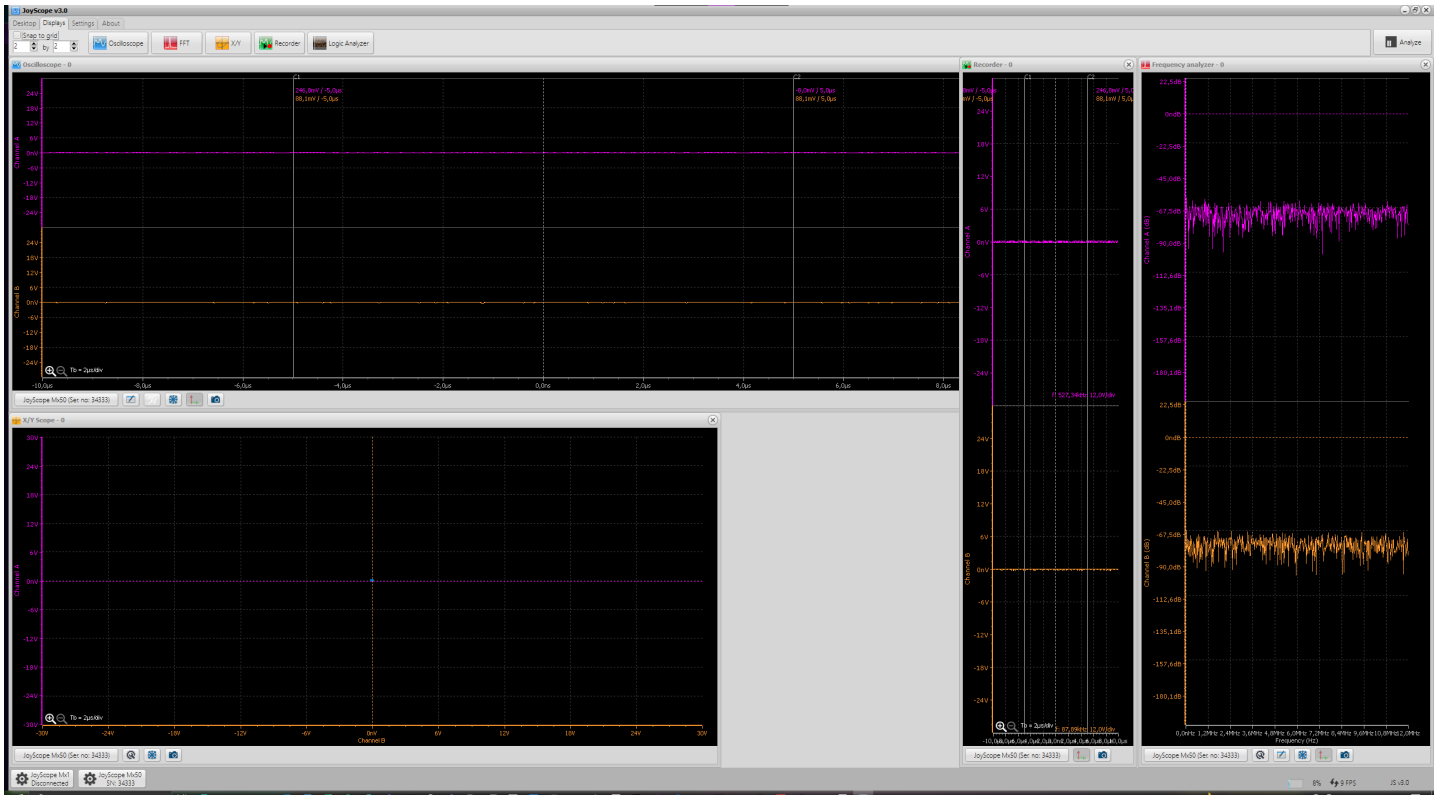
5.4 Various displays

With the **Snap to Grid** function, the displays in the application window can be automatically placed in different arrangements. The first number stands for the number of rows and the second for the columns. If **Snap to Grid** is activated, all displays are positioned in a grid with the specified number of rows and columns.



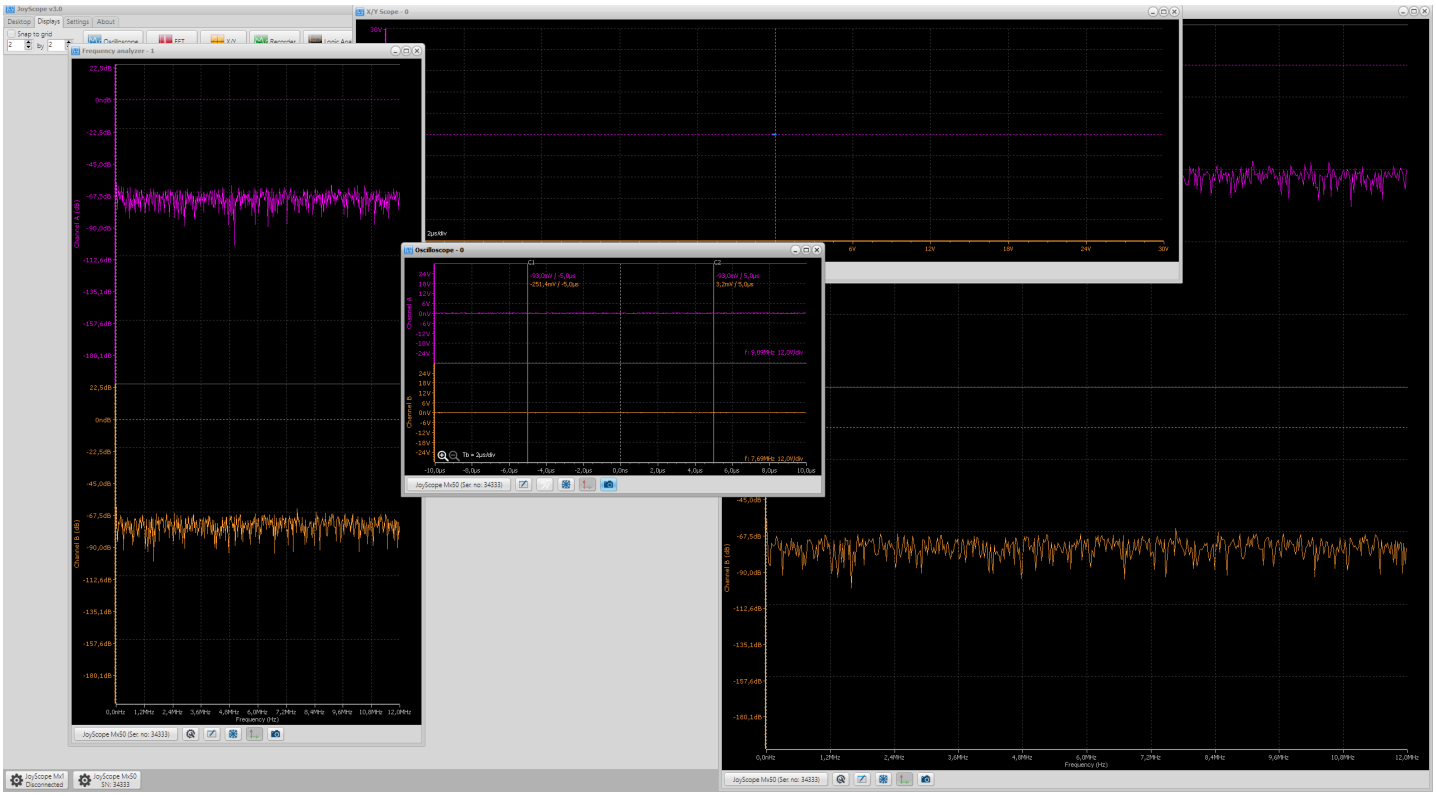


You can arrange the displays according to your own ideas at any time. Both the position and the size of the displays in the work area can be customized. The device functions are also shown in the **Displays** tab.



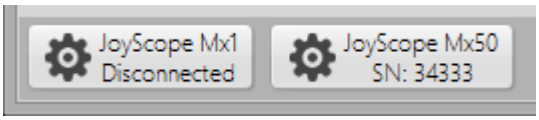
A **dynamic layout** is also available and can be activated in the settings menu. This layout allows you to position displays outside the application.

It should be noted that windows that are opened from the application automatically come to the foreground and therefore overlay all other content on every monitor.



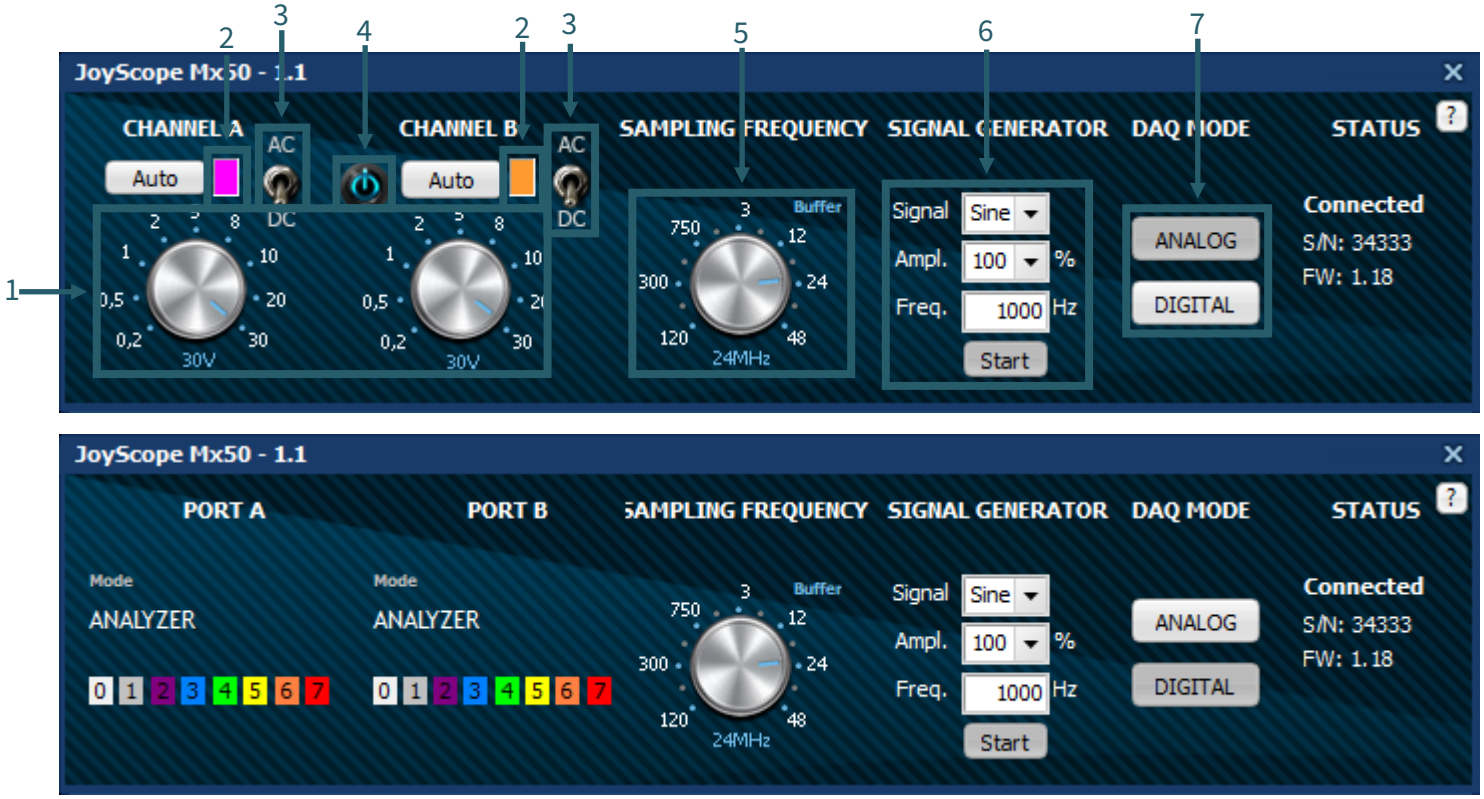
5.5 Setting up devices

Each supported physical device is represented in the JoyScope software by a visual interface called the **Device Control GUI**. This interface allows configuration and control of the connected devices. The controls to adjust the visibility of this interface are located in the lower left corner of the main window of the JoyScope software.

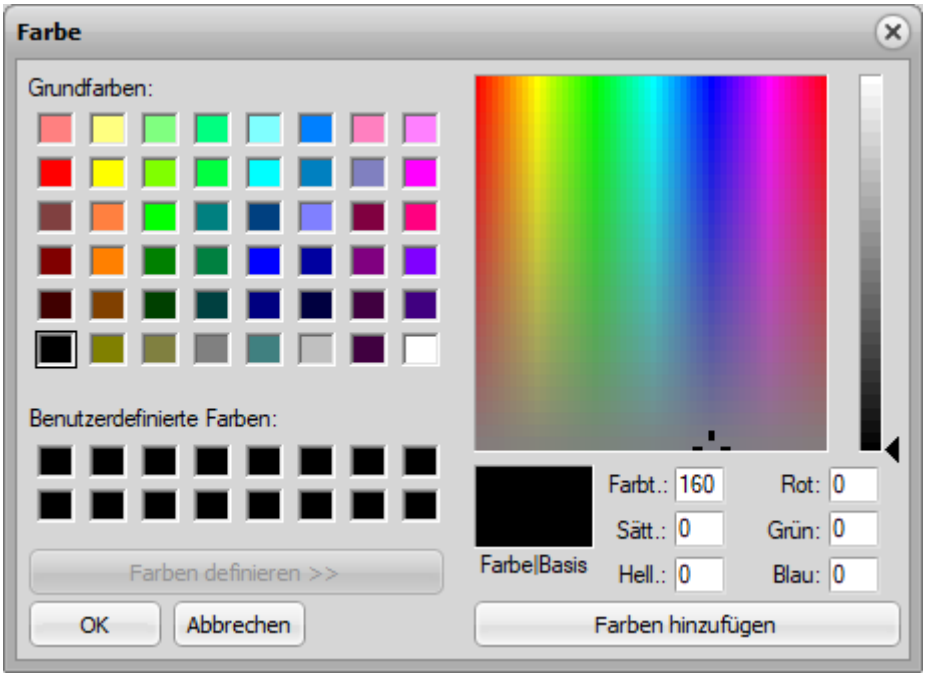


The following adjustments can be made in the device settings:

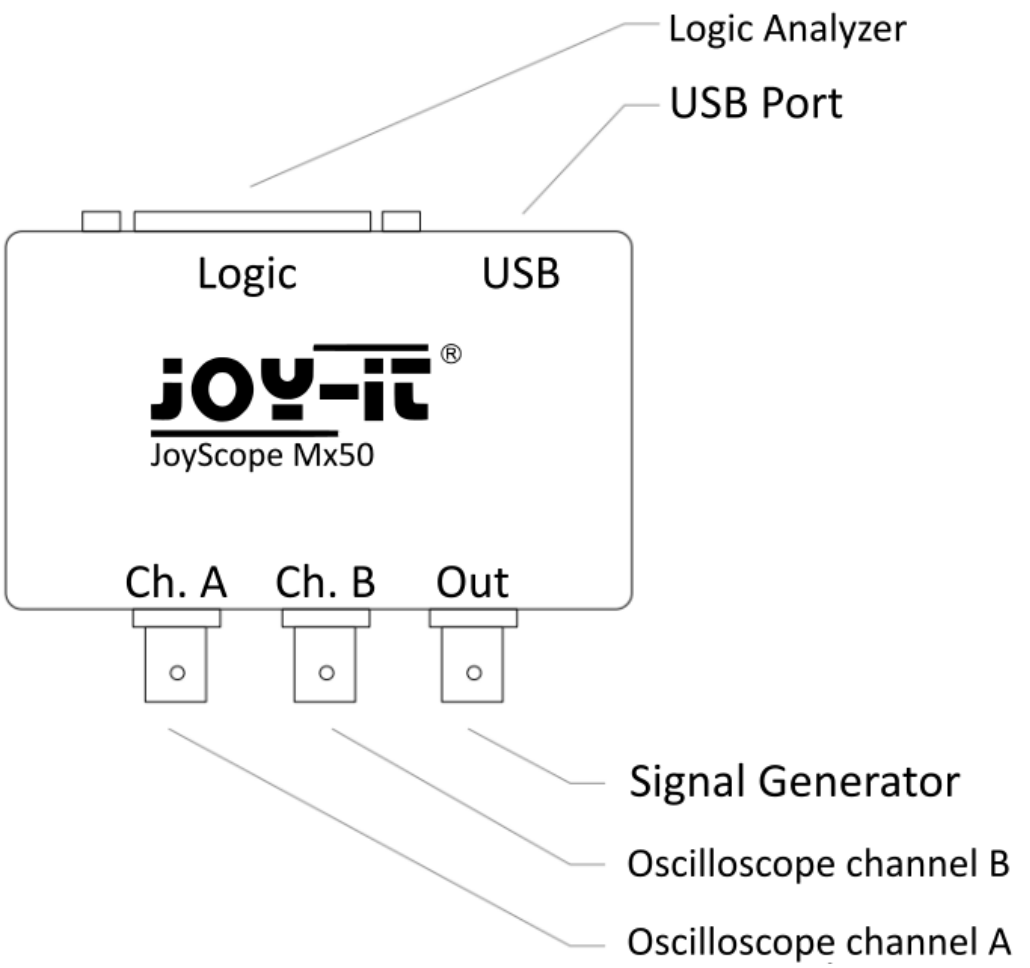
1. Setting the channel input range for each channel (A, B)
2. Selection of the channel color
3. Switching between AC and DC (to suppress the DC component)
4. Activation or deactivation of channel B
5. Definition of the sampling frequency
6. Configuration of the signal generator (waveform, amplitude, frequency)
7. Switching between analog and digital DAQ mode



The color of the graphical signal display of each channel can also be adjusted by clicking on the color field under the channel name.

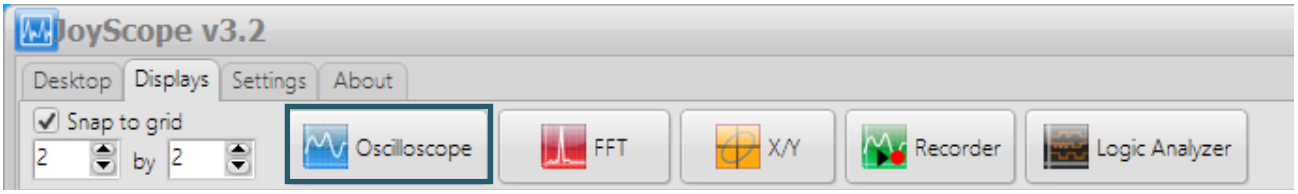


6. OSZILLOSCOPE

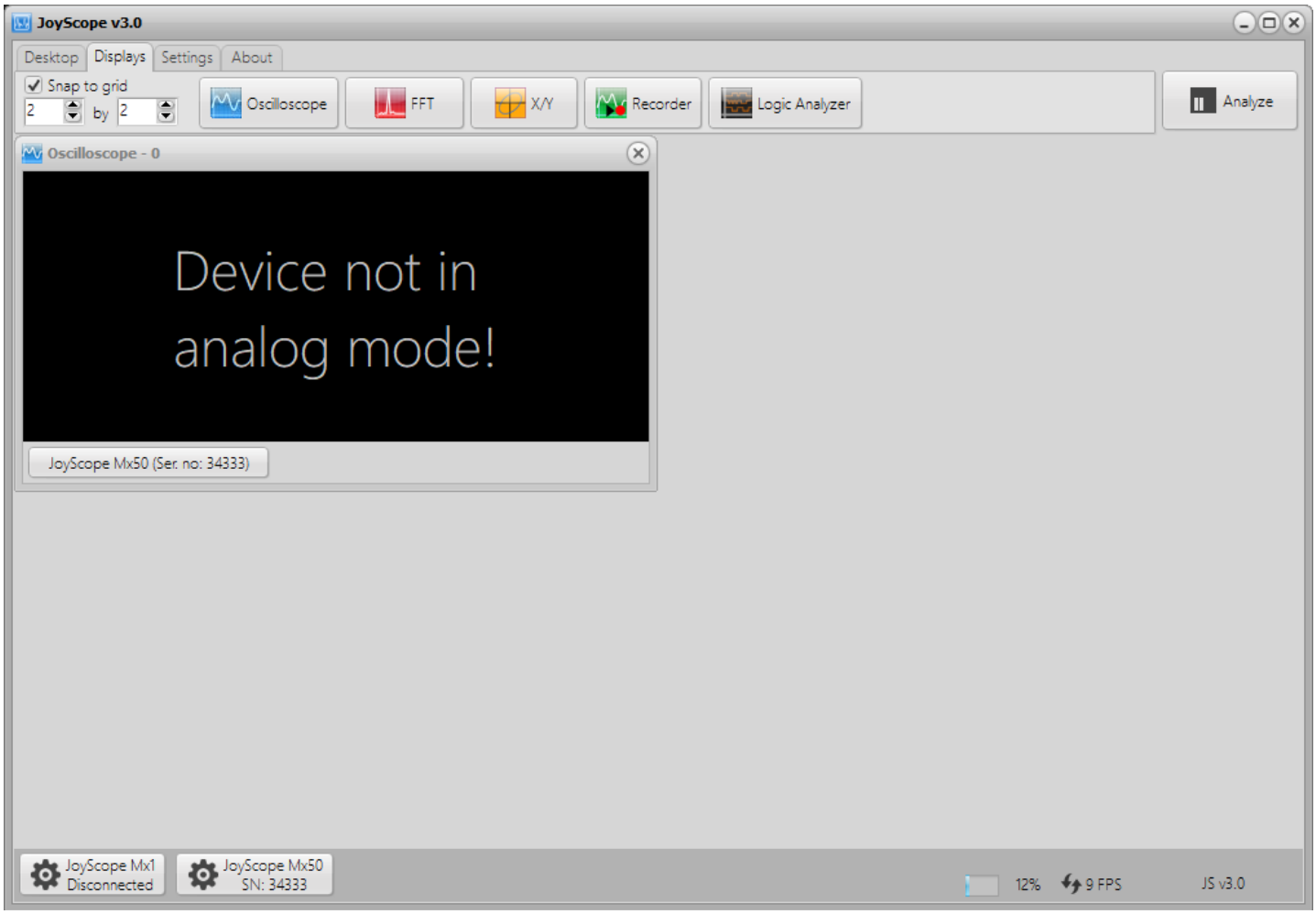


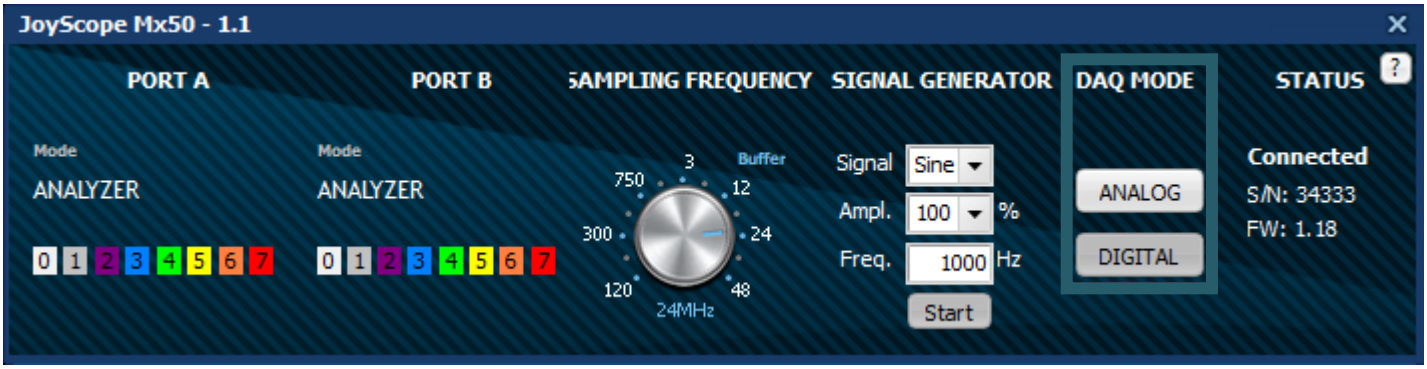
An oscilloscope is an electronic measuring device that visually displays electrical signals, both alternating and direct current (AC or DC). It displays voltage fluctuations graphically and is ideal for measuring and analyzing rapidly changing electrical signals. In this example, we have connected the output of the analog generator to oscilloscope channel A of the same ScopeMega50 device. You can use two probes connected to CH.A and OUT. Connect the tips of the probes and the grounds of the probes to each other.

You can open the oscilloscope by clicking on the Oscilloscope symbol in the **Function displays** menu.



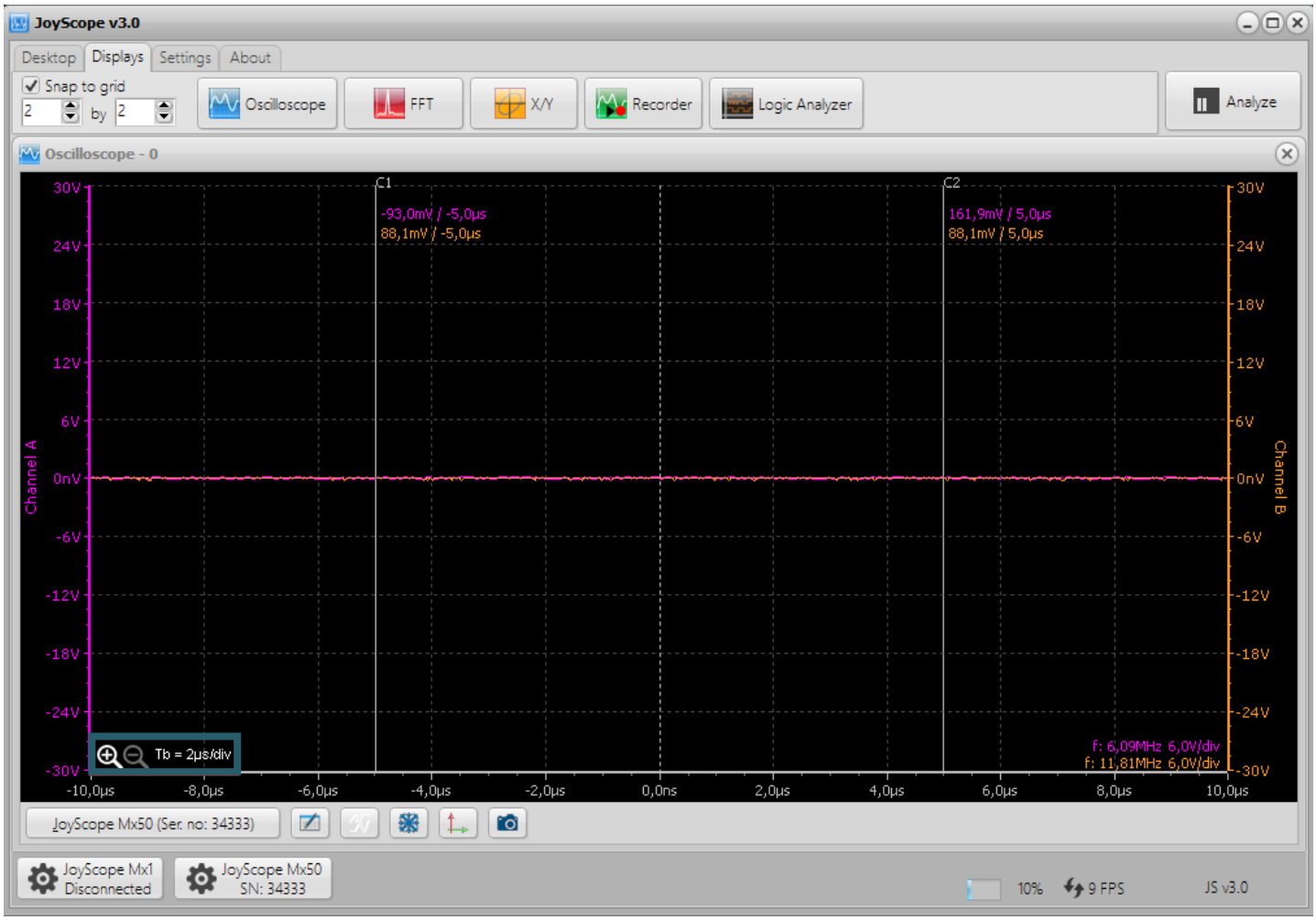
It may happen that you receive the message “Device not in analog mode”. In this case, you must open the JoyScope Mx50 control unit and switch the device to analog mode in **DAQ mode**.



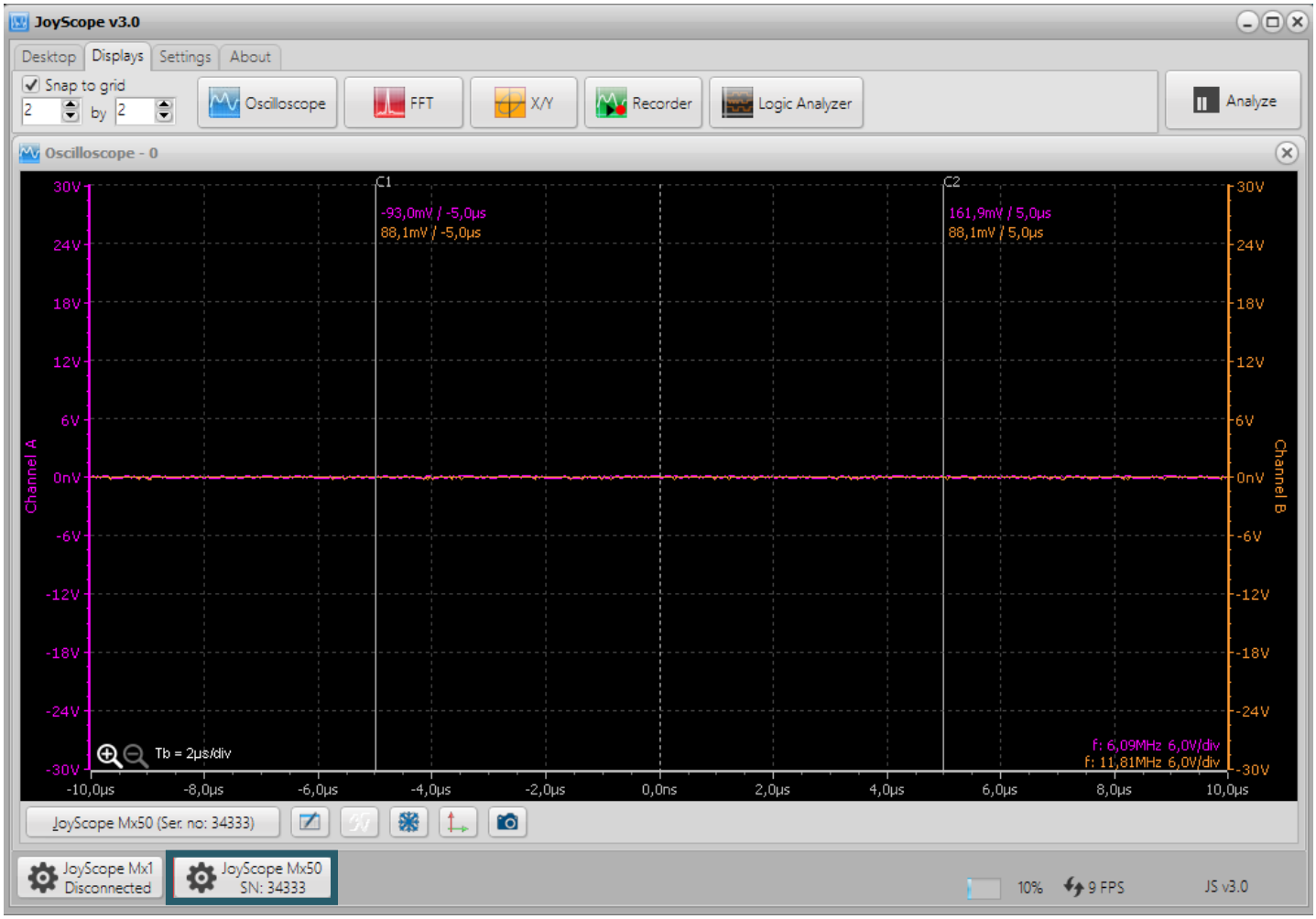


7. RECORDING MODE

A window with the graphical representation of the measured signal opens. To enlarge the time base, you can use the **magnifying glass symbol** in the bottom left-hand corner of the oscilloscope display.



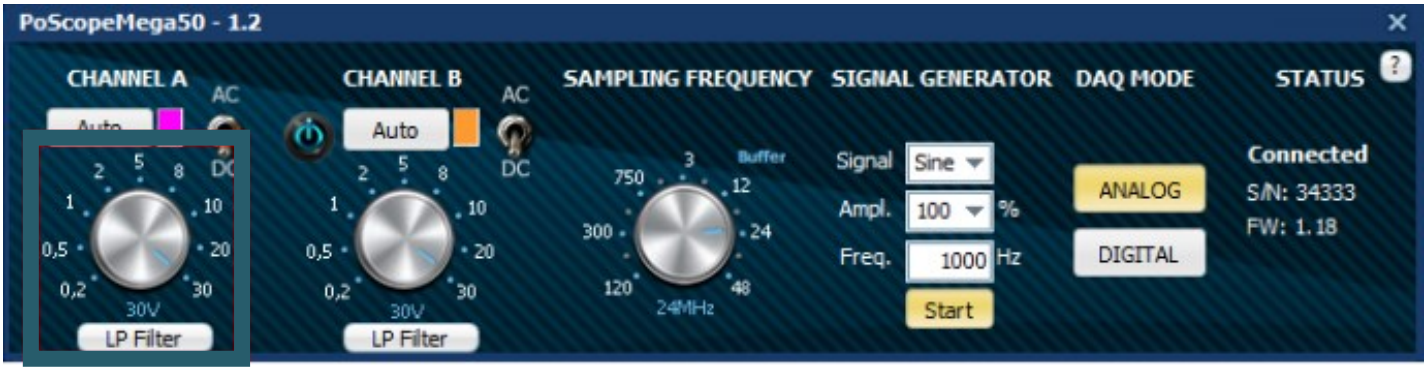
If several devices are connected, they will be listed in the device settings. Please note that you must increase the number of device instances in the settings menu to be able to manage multiple devices.



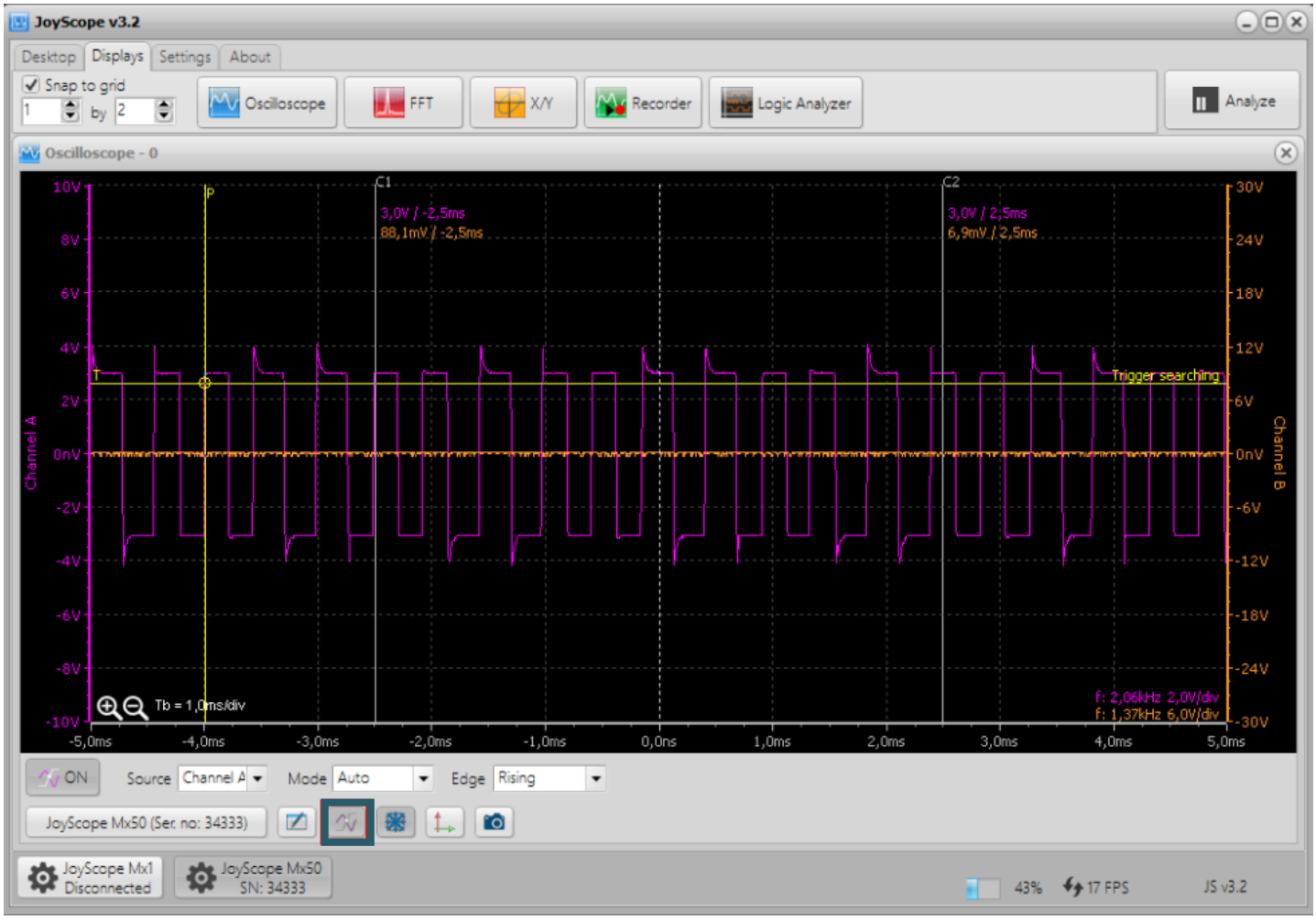
There are two ways to change the voltage range of the input:

1. Left-click on the channel slider in the graphical user interface and drag it, or use the mouse wheel while the mouse is over the slider.
2. Move the mouse over the axis in the diagram and use the mouse wheel.

Both methods have the same effect. You can invert the scrolling direction of the mouse in the settings menu.

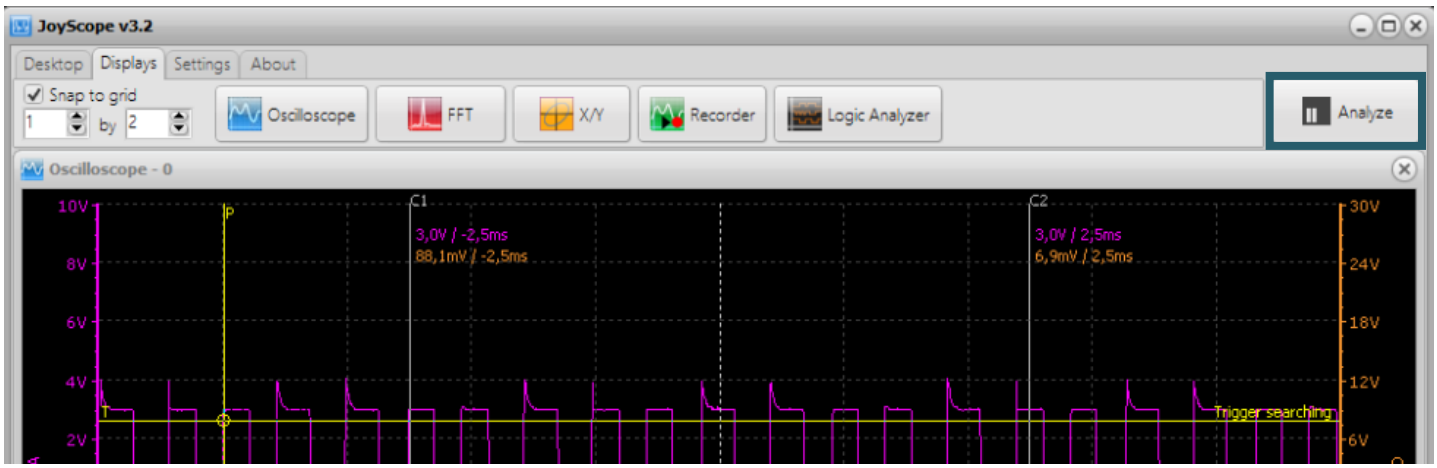


As the analog generator outputs a maximum voltage amplitude of 2 V, you can set the controller for channel A to 2 V. This results in the graphical representation shown below.



Some signals change very quickly. To capture the desired event precisely, you can use the trigger function. This standard function of an oscilloscope allows you to intercept the live signal at a specific point in order to analyze it precisely. Simply select the appropriate channel, the trigger mode (automatic loop or single trigger) and the edge of the trigger (rising, falling, or arbitrary).

Once you have captured the signal, you can analyze it in detail. To do this, click *Analyze* in the top right-hand corner of the JoyScope software to freeze the waveform.

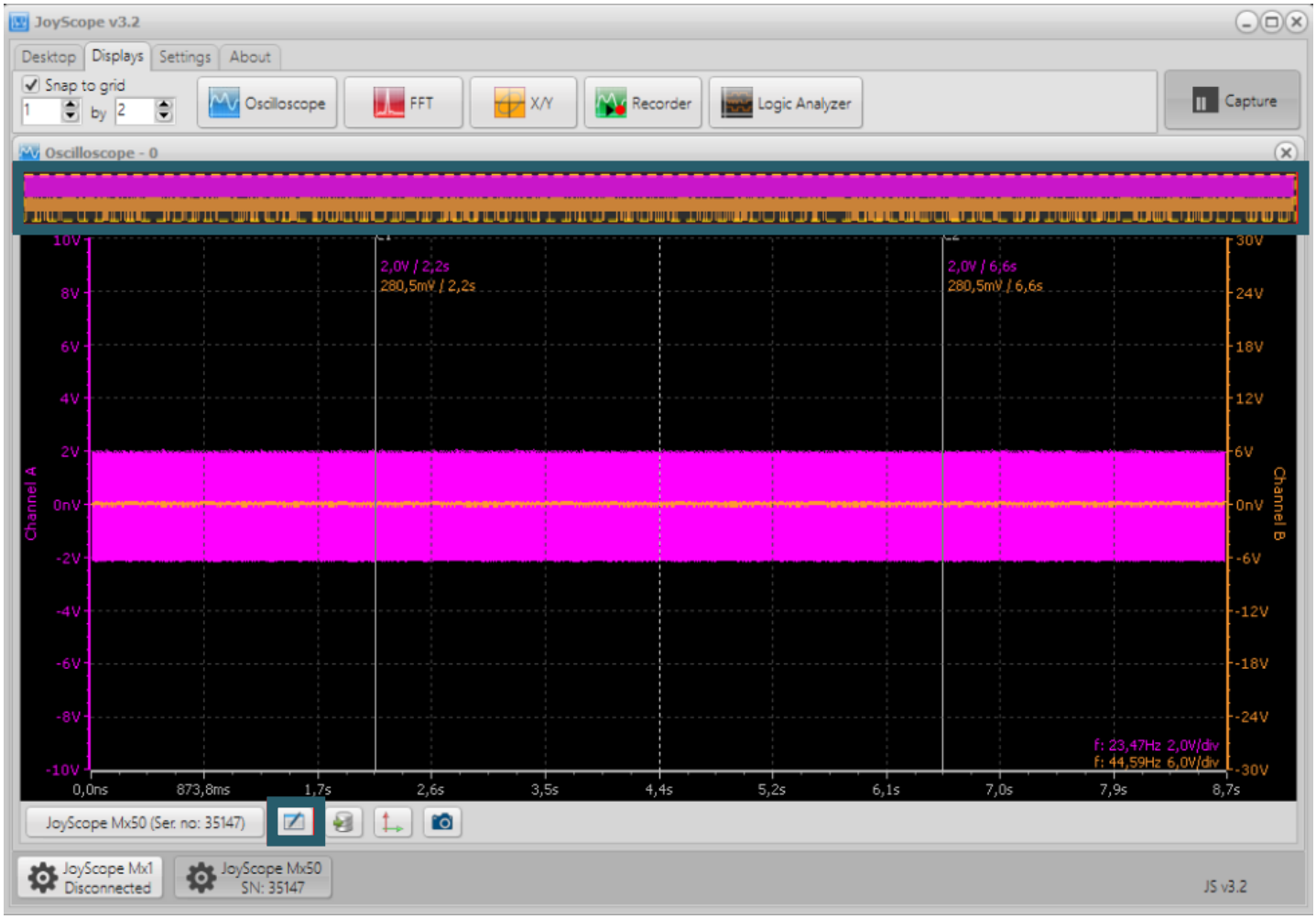


8. ANALYSIS MODE

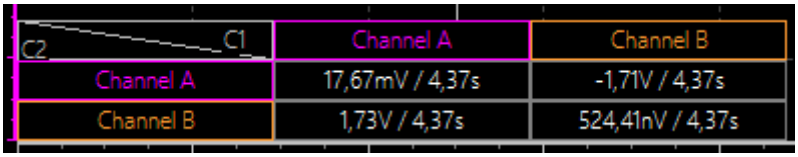
When you click *Analyze*, another control element appears above the waveform with a yellow selection area, known as the analysis overview. This function provides a quick overview of the entire data range in the buffers to facilitate detailed analysis. The data is sorted chronologically, with the oldest information displayed on the left and the most recent on the right.

The yellow selection rectangle in the overview marks the data section that is displayed in the main chart for analysis. The size and position of this selection area can be adjusted simply by moving the mouse.

To change the size of the yellow selection area, move the cursor over the main chart or the overview and use the mouse wheel. To move the waveform, hold down the left mouse button and drag across the diagram or move the selection rectangle directly in the overview.

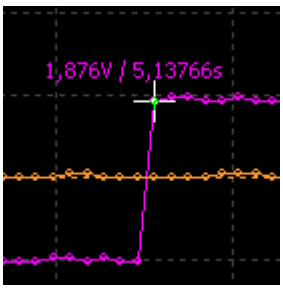


You do not have to calculate the amplitude of the signal manually each time by multiplying the Volts/Division formula by the number of graduation marks. Instead, you can use two vertical cursors, C1 and C2. These are located in the oscilloscope window and can be moved to the desired position for your measurements by clicking and dragging. At the bottom of the window, there is an icon that represents a data table. If you activate this, a small table will appear showing you the current values for the voltage amplitude and the time difference between C1 and C2.

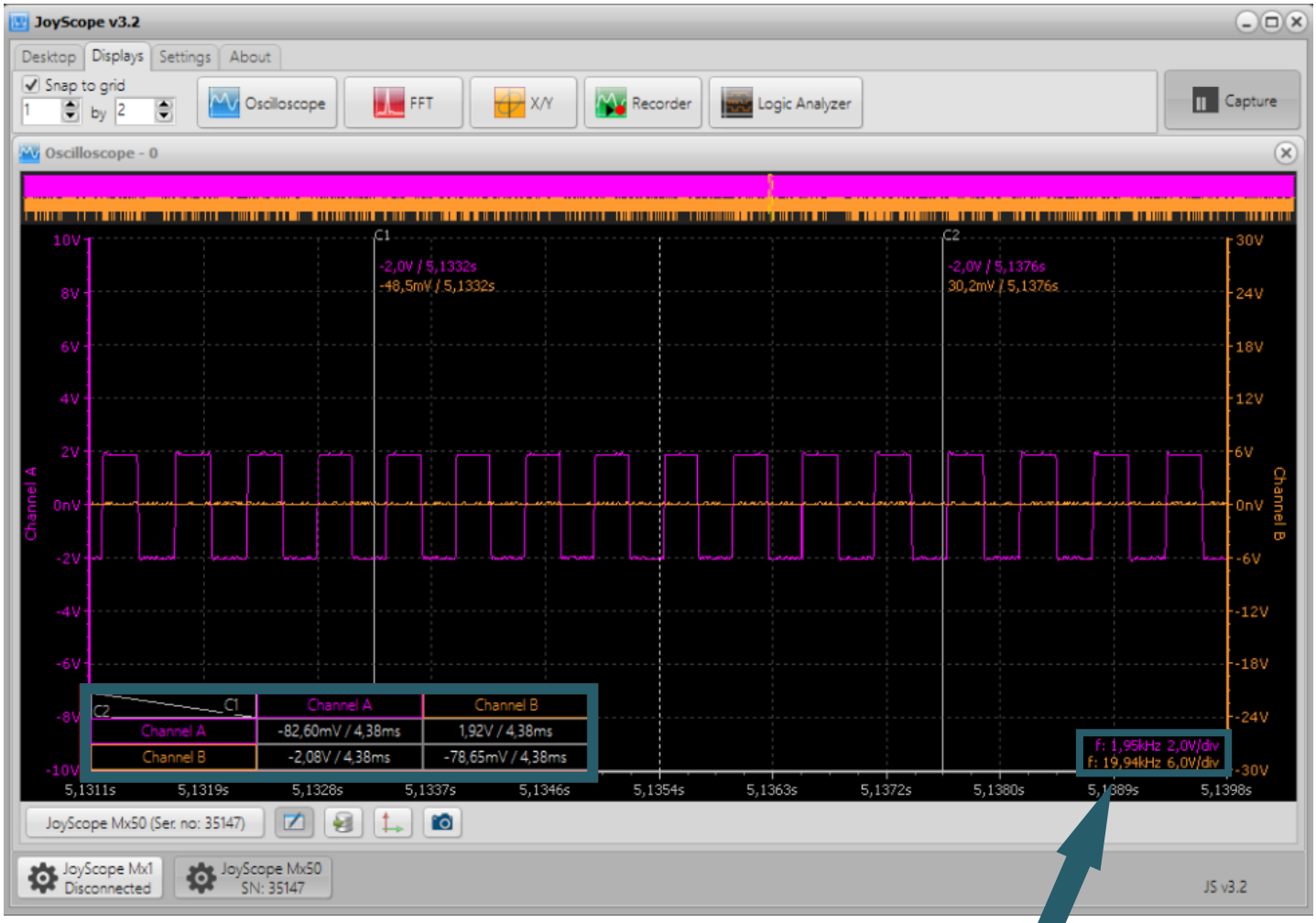


The data in the data table shows the voltage and time differences between the points of the signals under the cursors. An example of this would be the value 1.73 V / 4.37 s. The voltage difference is calculated as: Cursor C1 on channel A minus Cursor C2 on channel B. The time difference, which is simply calculated as C2 - C1, remains the same for each channel combination.

C1@ChA - C2@ChA (V) / C2 - C1	C1@ChB - C2@ChA (V) / C2 - C1
C1@ChA - C2@ChB (V) / C2 - C1	C1@ChB - C2@ChB (V) / C2 - C1



You can also simply move the mouse over the signal, with the crosshair cursor showing the current value of the signal, which is located directly under the mouse pointer.



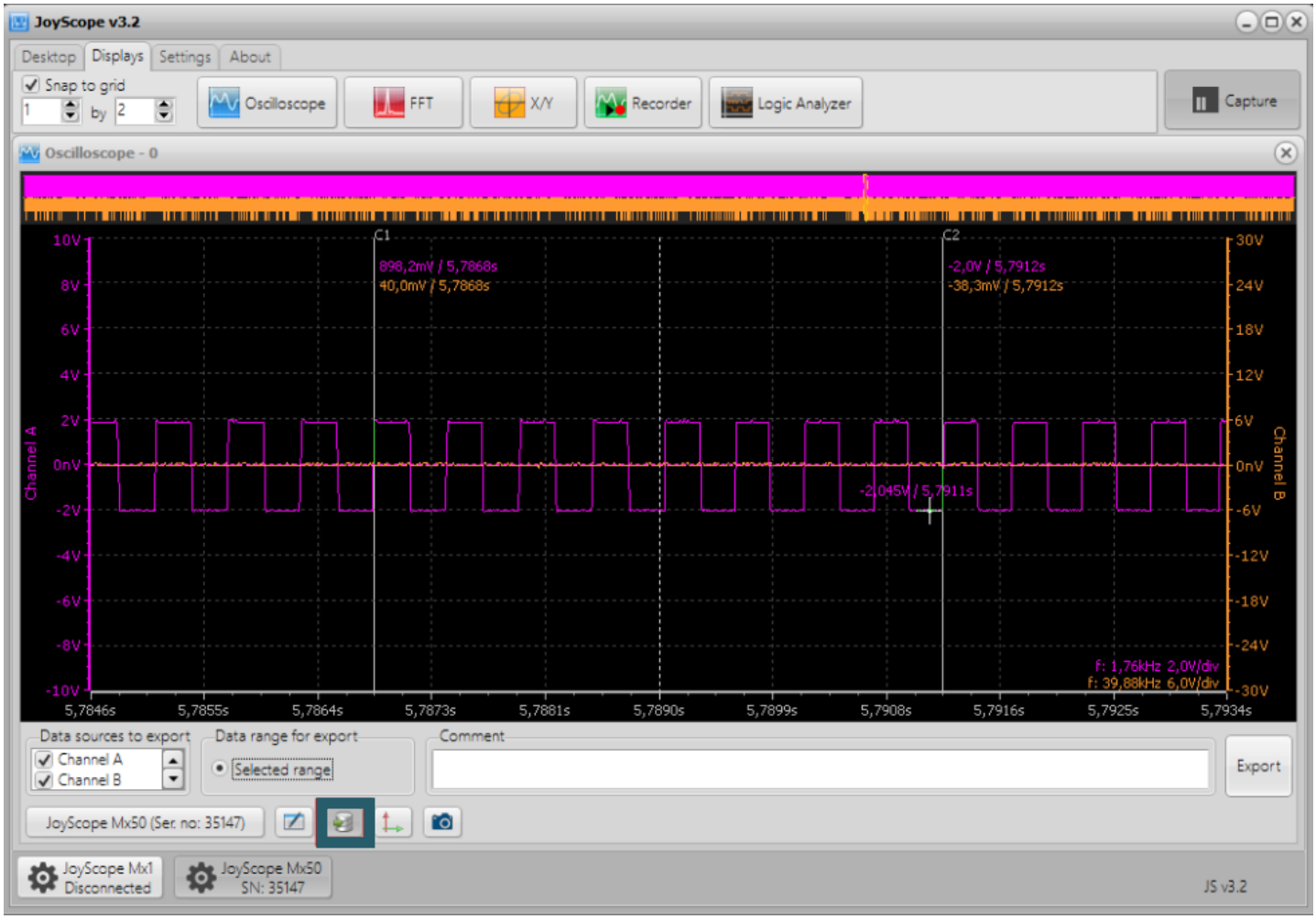
Tip: If you click with the left mouse button in the data table, you can switch between the calculation of the time difference between two cursors and the frequency calculation.

In the bottom right-hand corner, you will see the estimated frequency calculation. This estimate depends on the set sampling frequency and the signals at the inputs. Please note that if the sampling frequency is too high and the signal frequency is too low, the estimate may be inaccurate.

9. EXPORTING DATA

If required, you can export the measurement data to an external file. Select the data of interest (channel A, B) and the current diagram or buffer area and then click on the export icon. Supported file formats are:

- *.csv*
- *.xls*
- *.html*
- *.pcm* - when exporting to PCM, an additional *.txt*-header file with information is created (only available for oscilloscope and recorder)



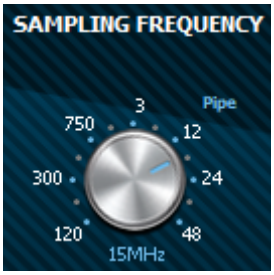
10. SAMPLING FREQUENCY (ANALOG AND DIGITAL DAQ MODE)

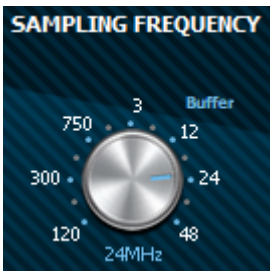
If you change the rotary knob for the sampling frequency, you will notice that the label above the rotary knob changes from Pipe to Buffer. Pipe and buffer are designations for two different data transfer modes.

The data is transferred via USB 2.0 at full speed. However, the JoyScope can create samples faster than the data can be transferred via USB.

Pipe mode: Sampling is slower than the maximum transfer speed of USB 2.0, allowing the device to transfer the sampled data continuously and without interruptions via USB 2.0.

Buffer mode: Scanning is faster than the maximum transfer speed of USB 2.0. In this case, the device only transfers a limited amount of data (data images) and there are interruptions between data transfers.





The limit value for the pipe mode is set dynamically, depending on the available resources. This allows the data transfer rate to be adjusted to ensure optimum performance based on the current system conditions:

- f_s - Sampling frequency [samples / second]
- N - Number of channels used [n] (is channel B switched on?)
- S - Size of a sample in bytes [bytes] (8-bit / 12-bit)
- V_{usb} - USB 2.0 maximum transfer rate (1.5 MB/s in theory)

$$f_s \times N \times S < V_{usb}$$

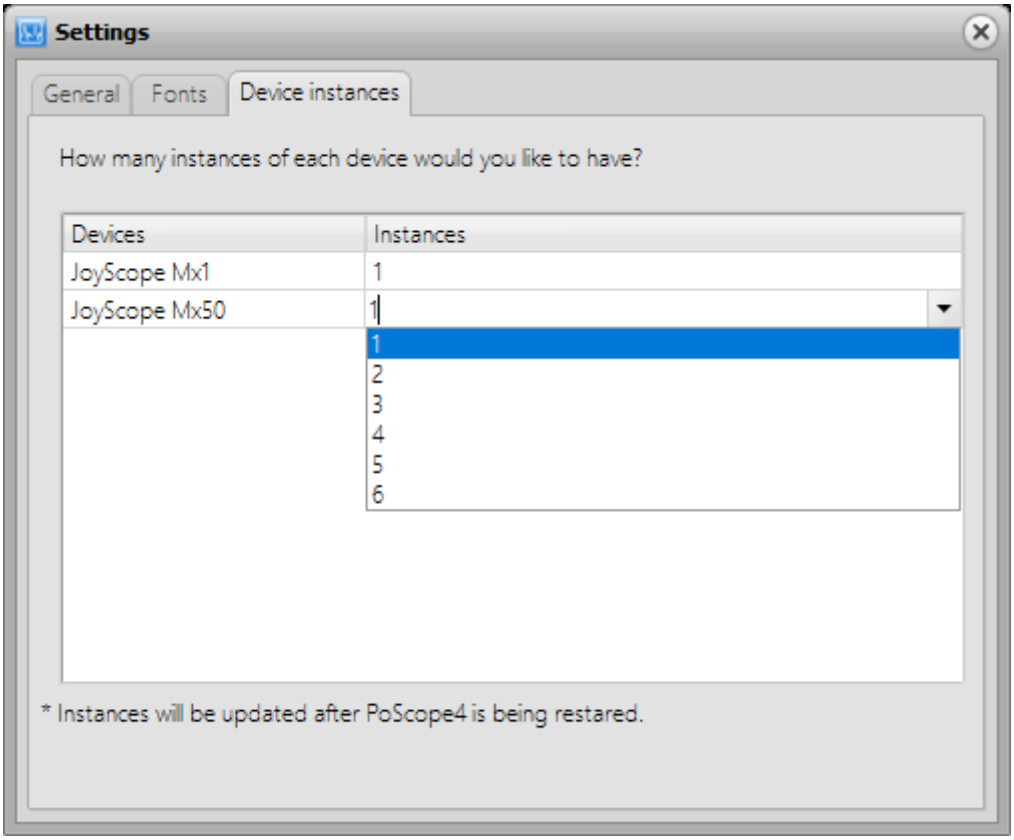
11. USE OF MULTIPLE DEVICES

Occasionally it is necessary to measure several analog signals simultaneously. JoyScope supports the operation of multiple devices and the use of multiple instances of one device type. Each device is provided with its own user interface.

If you have several JoyScopes, you can configure them so that they can manage multiple instances. This is done via the **Settings** button in the **Settings** tab.

In the window that appears, navigate to the **Device Instances** tab and specify how many instances are required.

After selecting the desired number of instances, restart the software. New buttons for the devices are added to the lower toolbar. Each device control instance automatically detects the connection of the device to the computer and establishes a connection.



Synchronization problem

It is possible to connect two or more devices to the computer, but they cannot be displayed on the same graph because the devices do not have a common synchronization clock for sampling. Each device works with its own clock and does not provide an external clock synchronization function, so the data between different devices cannot be synchronized correctly. Nevertheless, the devices can be used on separate displays without any problems.

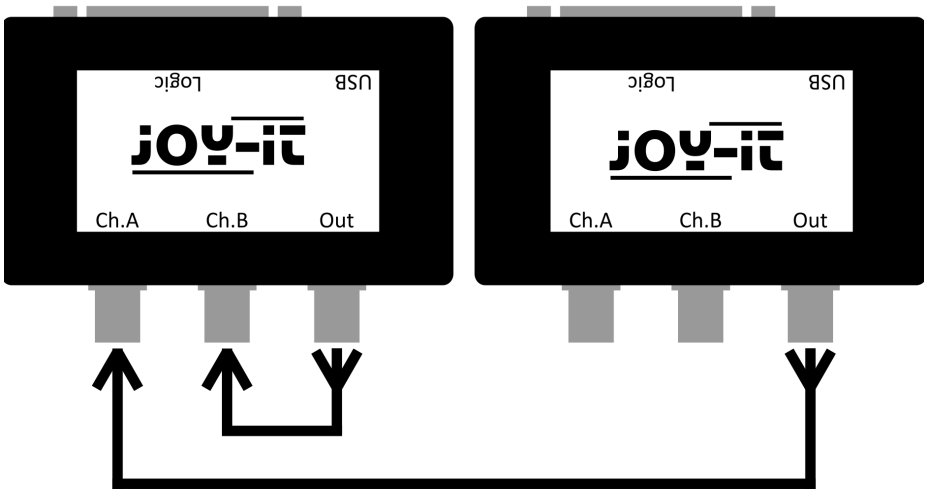
12. X/Y SCOPE

There are applications where it is important to know the phase difference between two or more different voltages. An oscilloscope can be used to display the phase differences between several input signals. The X/Y scope visualizes one voltage against another. If the input signals are periodic, so-called Lissajous figures are created. This tool can be used in various areas, including:

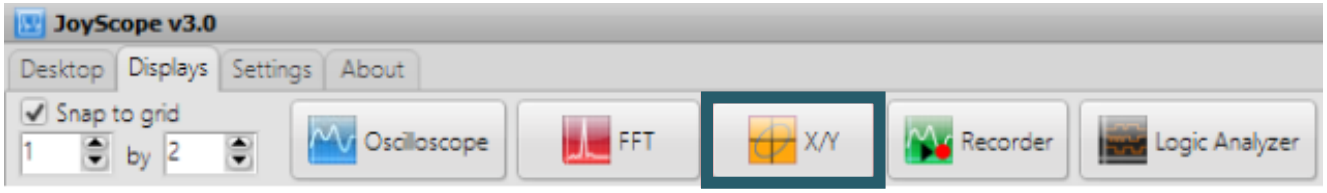
- Component characteristics (I-V curves)
- Radio technology (stereo - left/right)
- Audio amplifiers (distortion between input and output)
- Guitar tuning
- And much more

For this example, we need an oscilloscope (JT-ScopeMega50) and two voltage sources. The analog generator is a standard function of the JoyScope oscilloscope. However, since the JoyScope is only a single-channel device, we require two JoyScope devices to measure both voltage sources simultaneously.

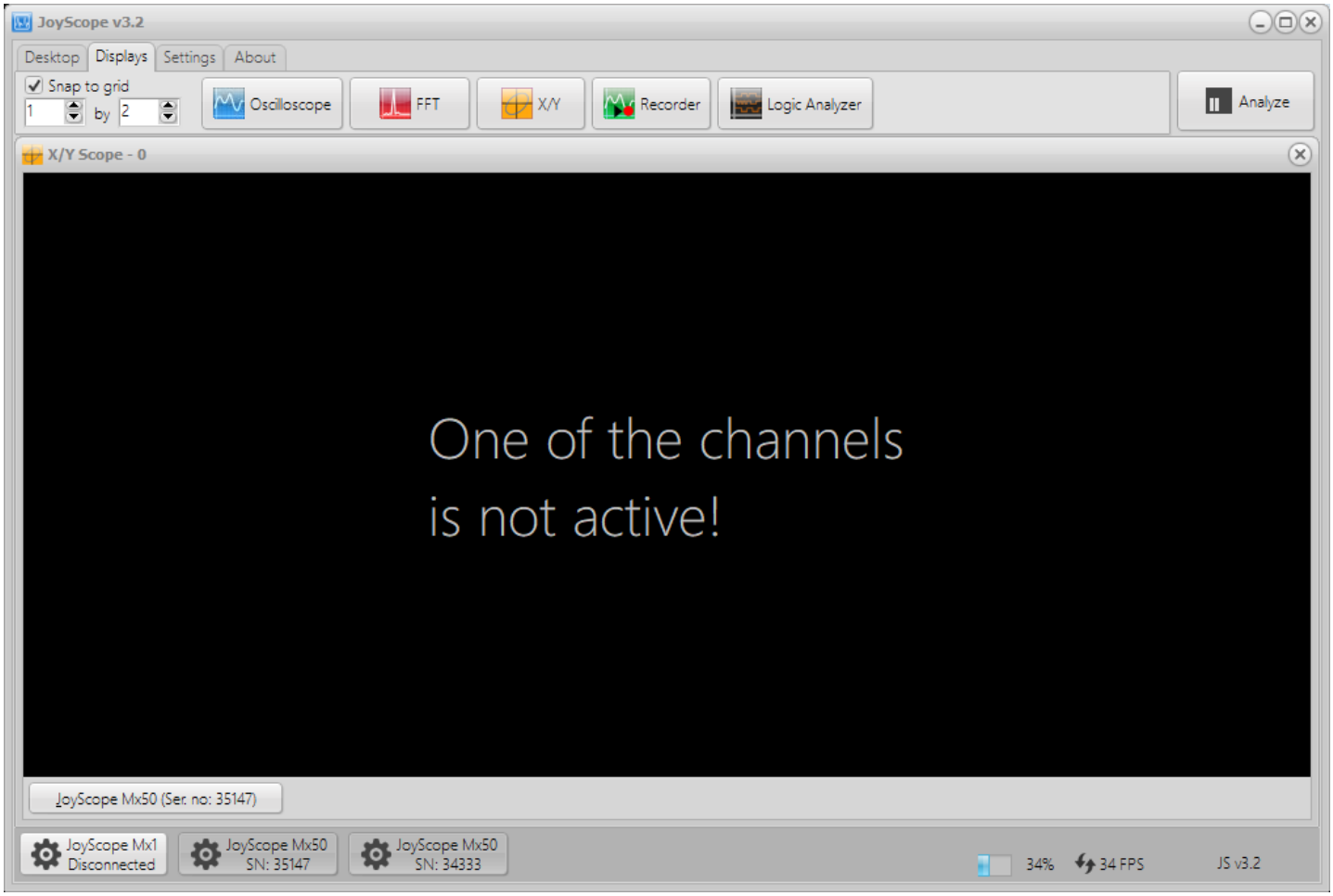
To use multiple devices in the software, navigate to the settings and increase the number of device instances. This allows you to operate both devices simultaneously and measure the voltages on each device separately.

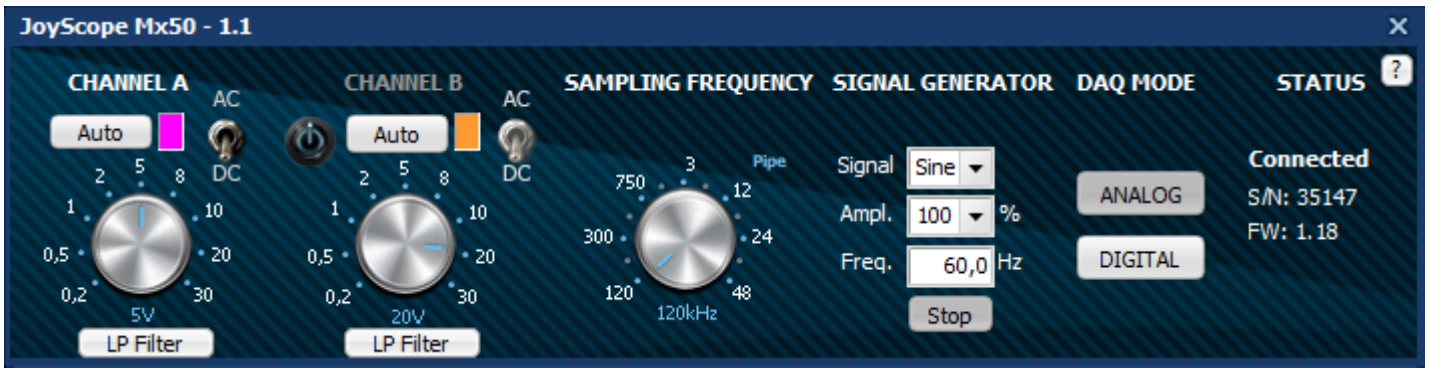


You can open the X/Y scope by clicking on **X/Y** in the **Displays** tab of the JoyScope software.

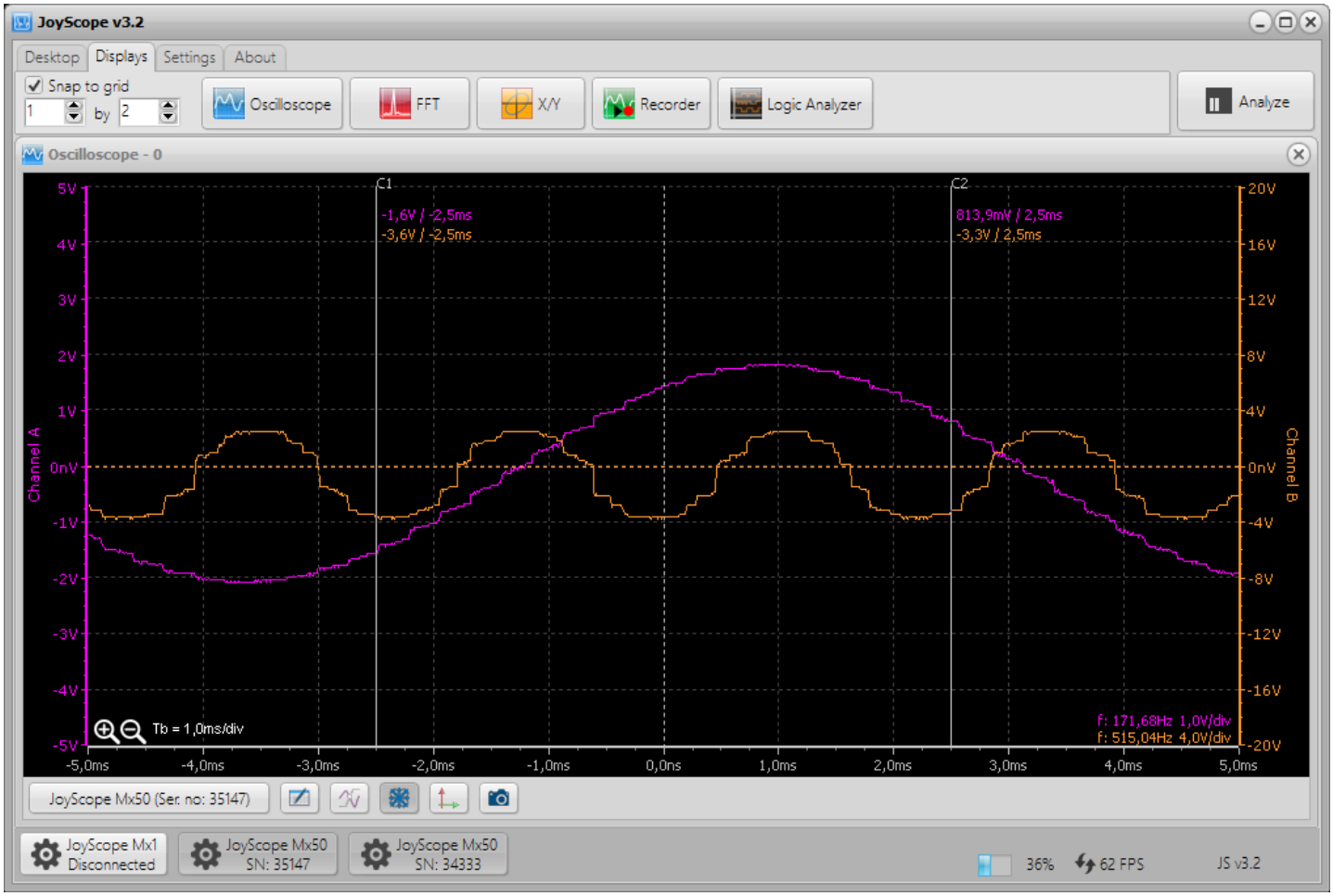


To be able to track phase differences, both channels must be activated. Switch the signal to channel B in the device settings.

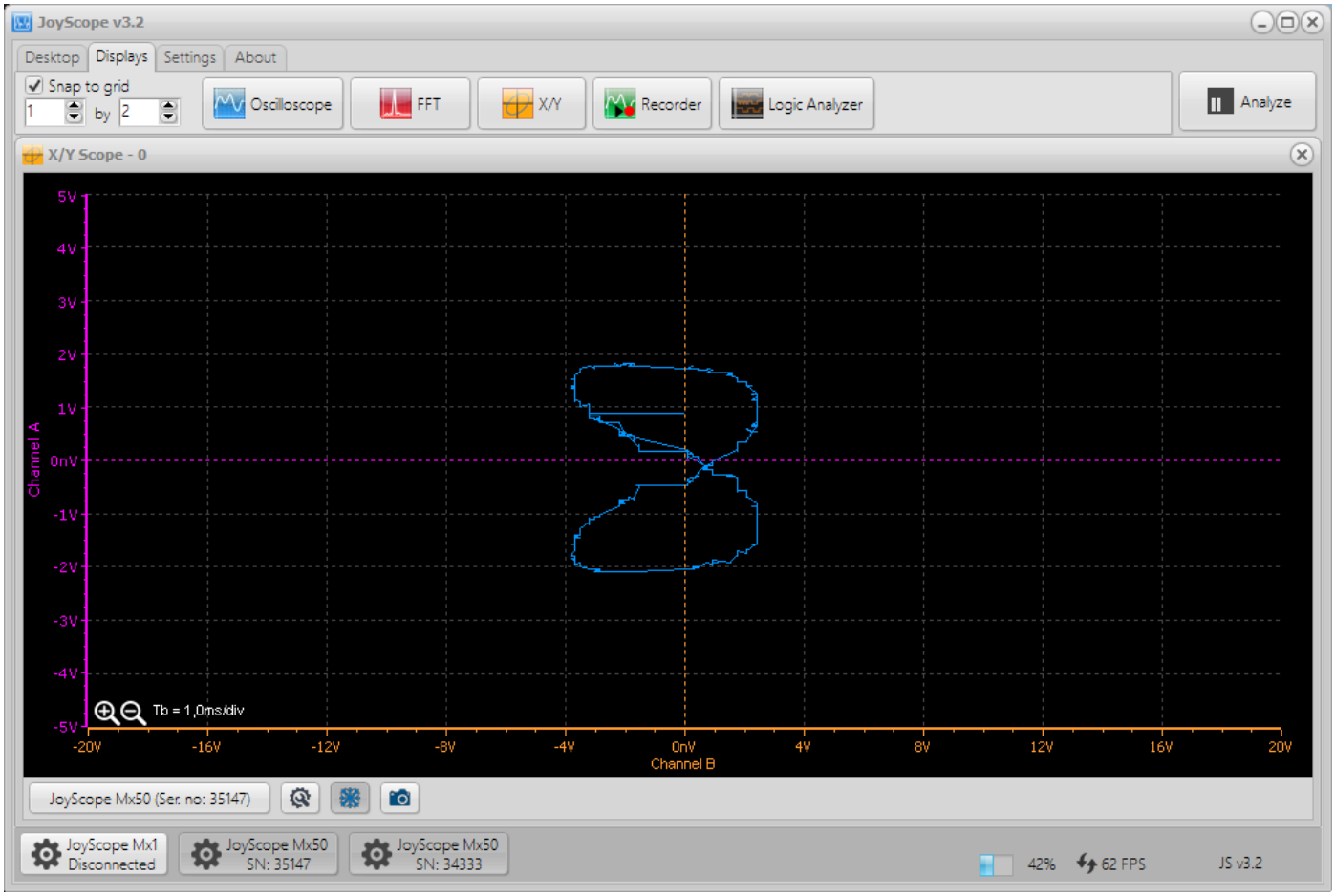




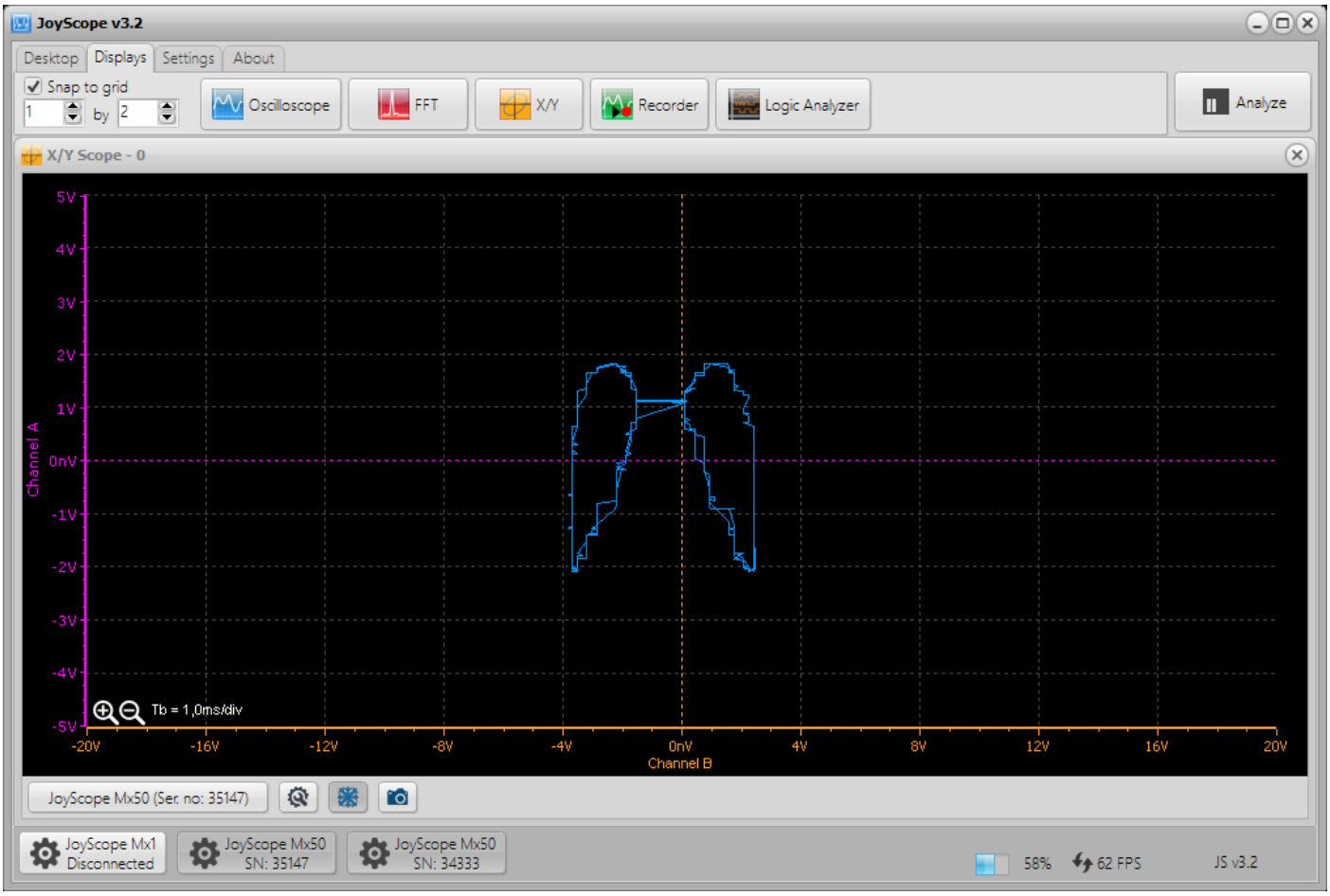
To make the phase differences visible, you can now open the oscilloscope display and activate the trigger function for one of the channels. One of the signals is frozen while the other moves across the display in relation to the first signal.



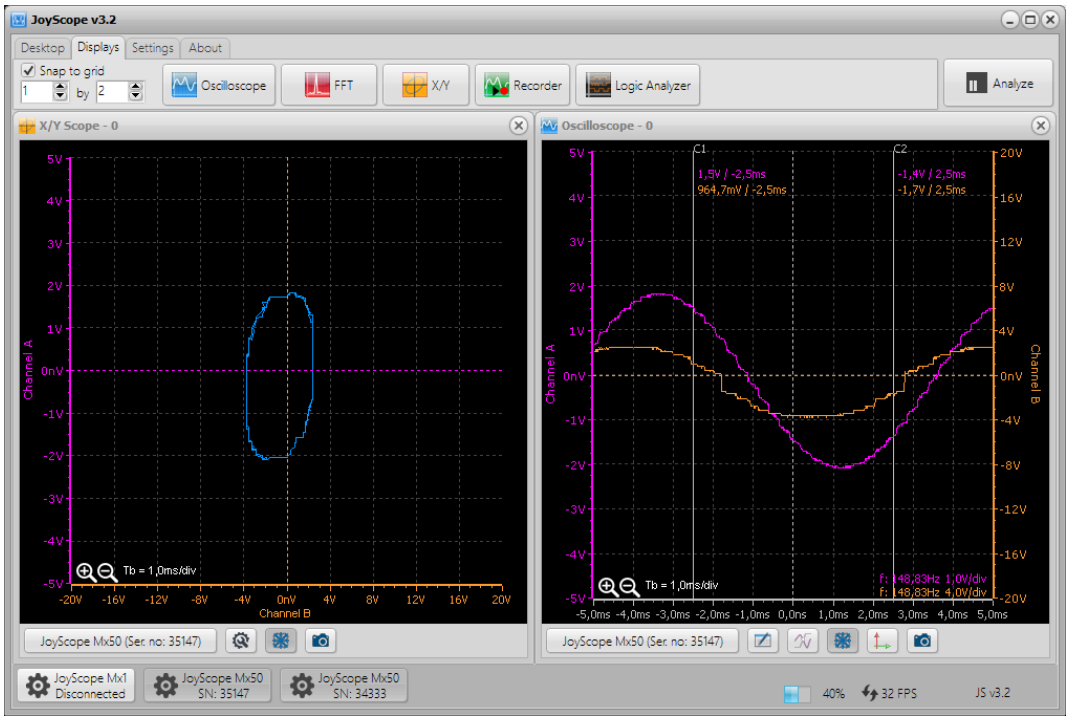
Let us now draw some basic Lissajous curves. If the ratio of the frequencies is set to 2:1, for example 60 Hz on channel A and 120 Hz on channel B, the following characteristic Lissajous curve is obtained, which takes on an elliptical shape. This curve shows the ratio and the phase difference between the two periodic signals.



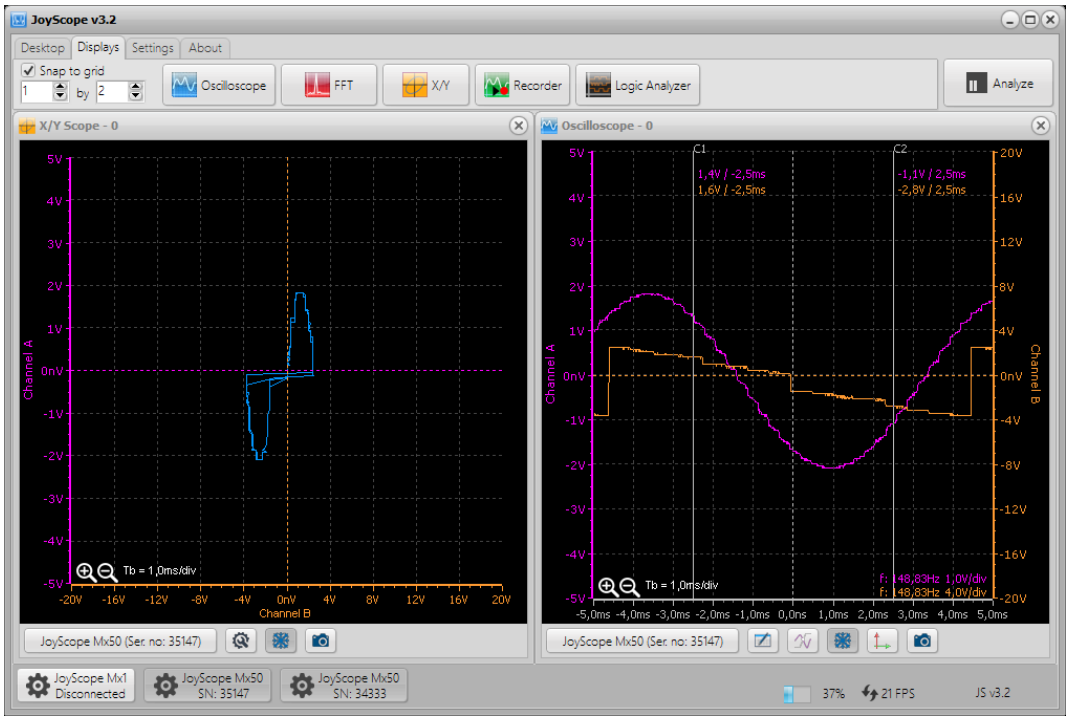
If we change the frequency ratio to 1:2, i.e. 120 Hz on channel A and 60 Hz on channel B, the Lissajous curve rotates by 90°. With larger ratios, we get more twists and more complex shapes, which is due to the changed phase relationship between the two signals.



If there is a phase shift of 90° between the two signals, the X/Y display will draw a circle. This occurs when both signals have the same frequency and are exactly 90° out of phase, representing a harmonic relationship between the two signals.



There are many variations of Lissajous curves that are influenced by the way the signals are shaped. If we change the shape of a signal to a saw-tooth wave, we get a completely different Lissajous curve. The sawtooth wave is not sinusoidal, but has a sharp rising and falling edge, resulting in a more complex and less symmetrical Lissajous figure. These curves can be particularly useful for visualizing non-linear phase relationships and frequency relationships.



13. SPECTRUM ANALYZER

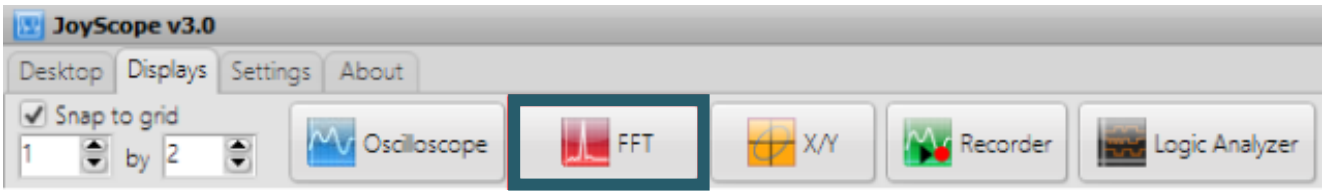
The spectrum analyzer measures the signal amplitude over a wide frequency range and plots a waveform of the signal amplitude (Y-axis) against the frequency range (X-axis). You can use the spectrum analyzer to determine the dominant frequency of a known signal, as well as the amplitude and frequency of unknown or unwanted signals. It also analyzes other spectral components such as power, distortion, harmonics, and bandwidth.

JoyScope uses the **Fast Fourier Transformation (FFT)** to analyze the frequency of an electrical signal. This mathematical process converts a waveform into its frequency components, allowing a detailed analysis of the signal structure.

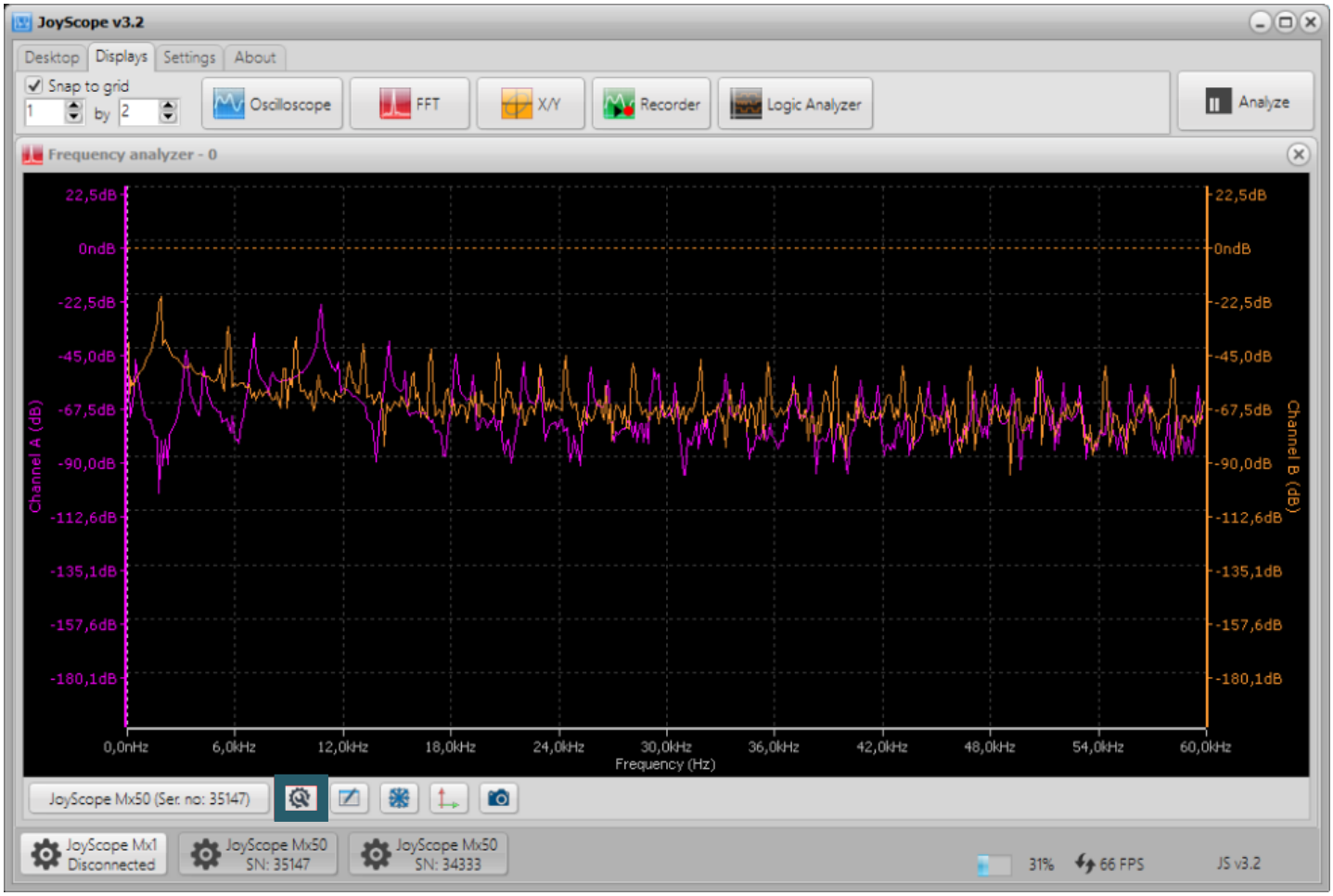
The Spectrum Analyzer can be used for a variety of tasks, including:

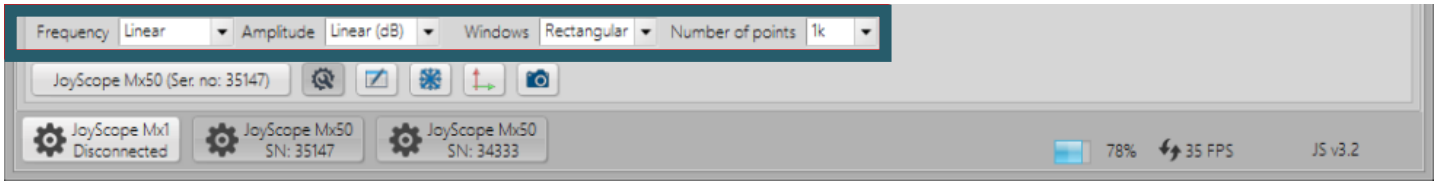
- Finding the dominant frequency
- Searching for signal interference
- Characterization of electronic devices
- Benchmark tests
- Detection of unknown signals
- EMC tests

You can open the Spectrum Analyzer by clicking on the **FFT** icon in the **Displays** tab:



The waveform below shows the frequency spectrum of a periodic signal coming from the analog generator. The frequency of the signal is set to approximately 10 kHz. The waveform is plotted as signal amplitude (Y-axis) against frequency (X-axis). If you click on **Settings** in the **Settings** tab, you can make additional settings to examine the frequencies of the input signal.

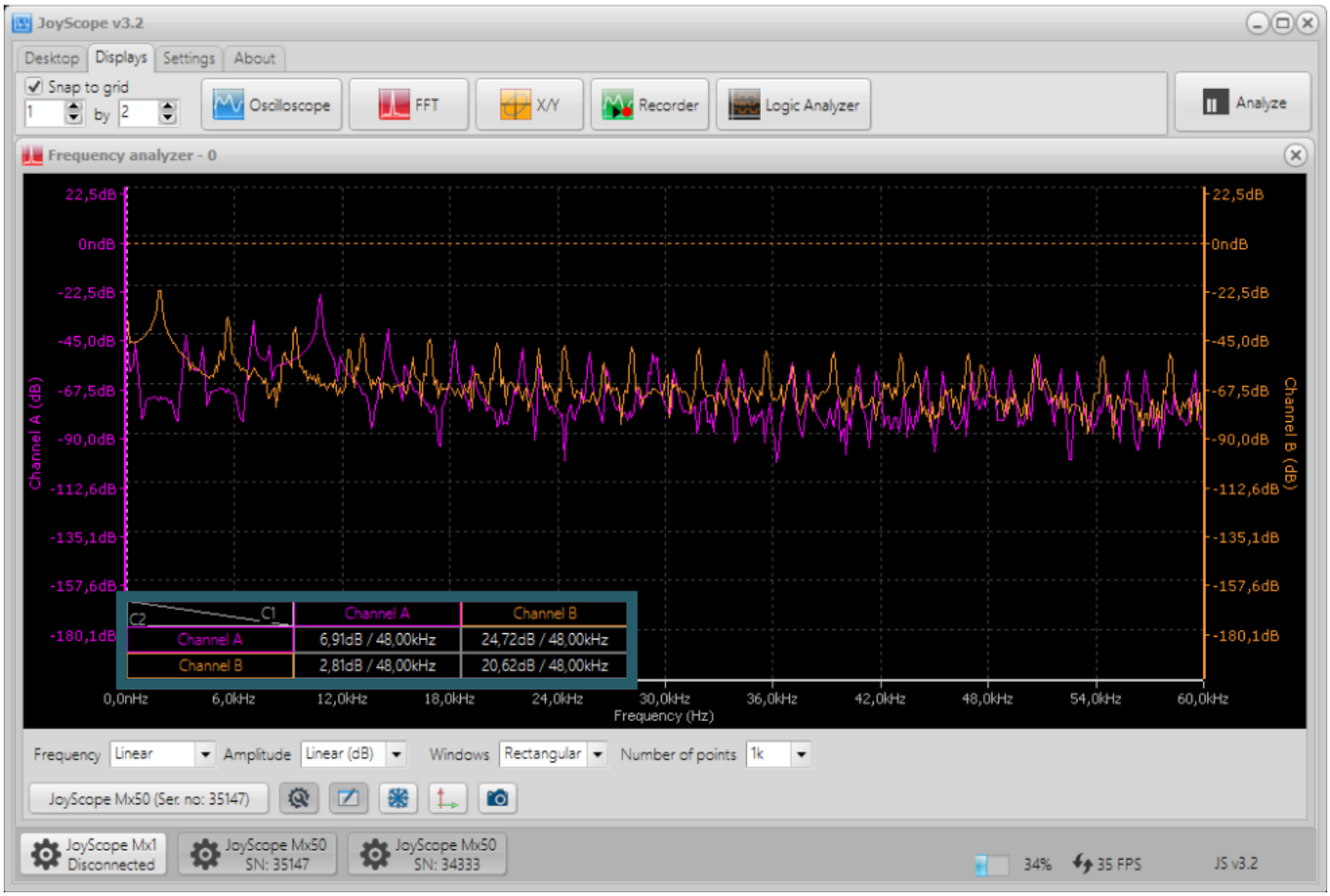




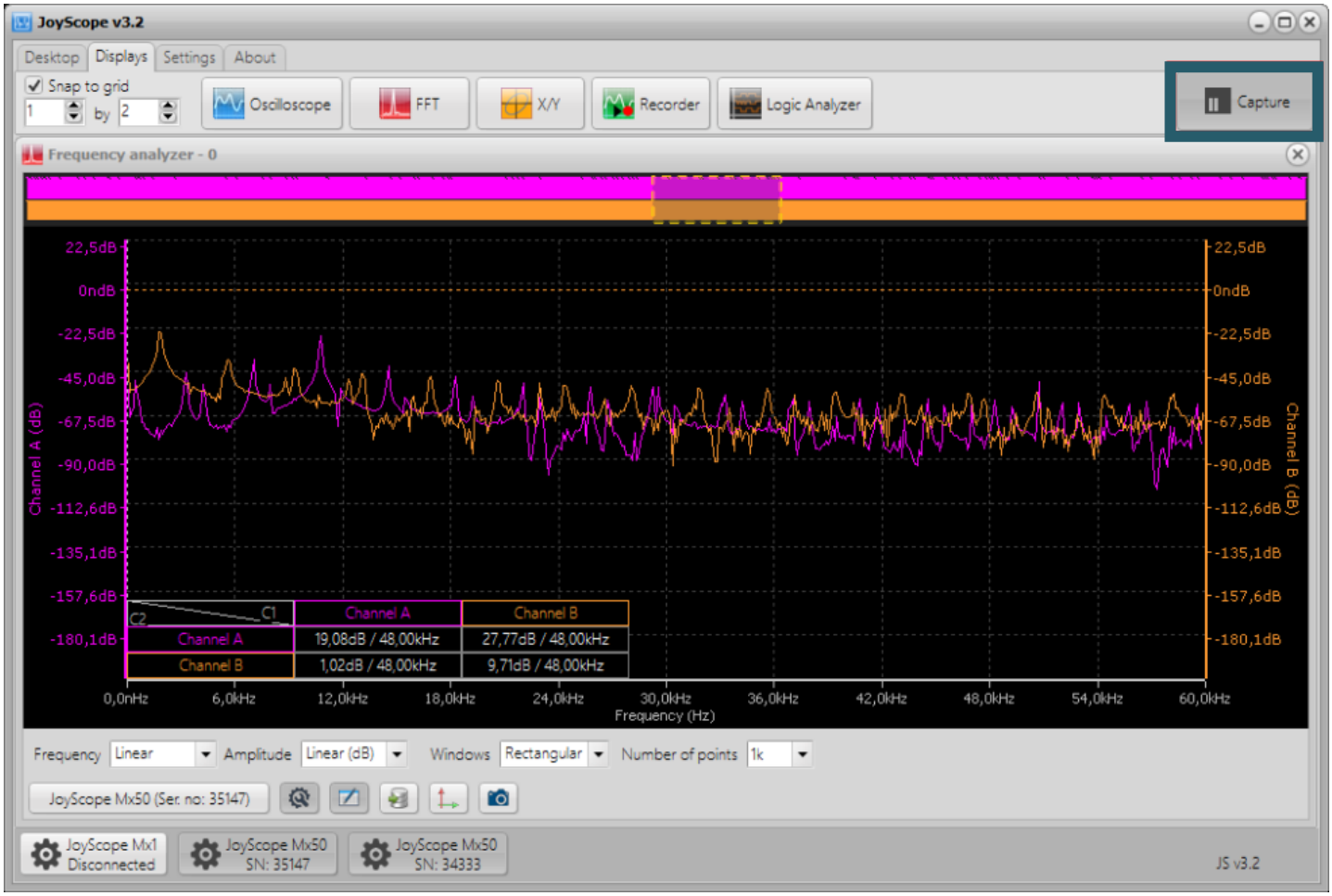
Here is a table showing all the possible combinations for visualizing the signals:

Frequency	Amplitude	Window function	Number of points
Linear	Linear	Rectangle	32
Logarithmic	Linear (0 db)	Hanning	64
Normalized	Linear (0 db max)	Hamming	128
	Normalized	Flat Top	256
		Triangle	512
		Blackman	1k
		Blackman Harris	2k
		Exponential descending	4k
			8k
			16k

Cursors are available in the frequency analyzer, which you can drag with the mouse to identify the most dominant frequencies. To analyze the exact relationships between cursors C1 and C2, click on the data table. This shows you important information such as the frequency difference and the amplitude values between the two cursors.



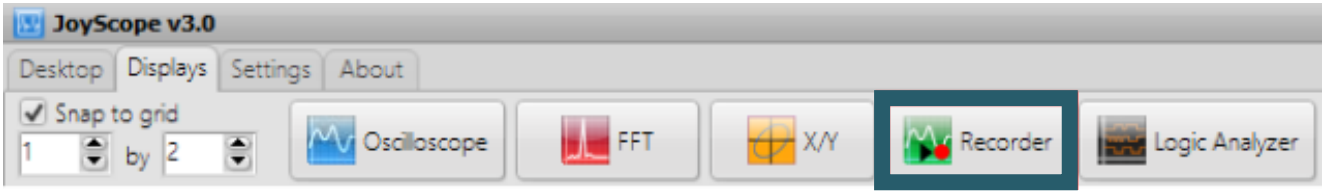
Clicking on the **Analyze** button takes you to analysis mode. The overview control is opened in analysis mode. You can adjust the size of the yellow selector by moving the mouse over it and reducing the size of the yellow quadrant using the mouse wheel. You can then move the waveform to the left and right to perform an FFT analysis on the selected part of the plot. In analysis mode, you also have the option of exporting the data to various file formats. To return to the live signal, simply click on the **Capture** button.



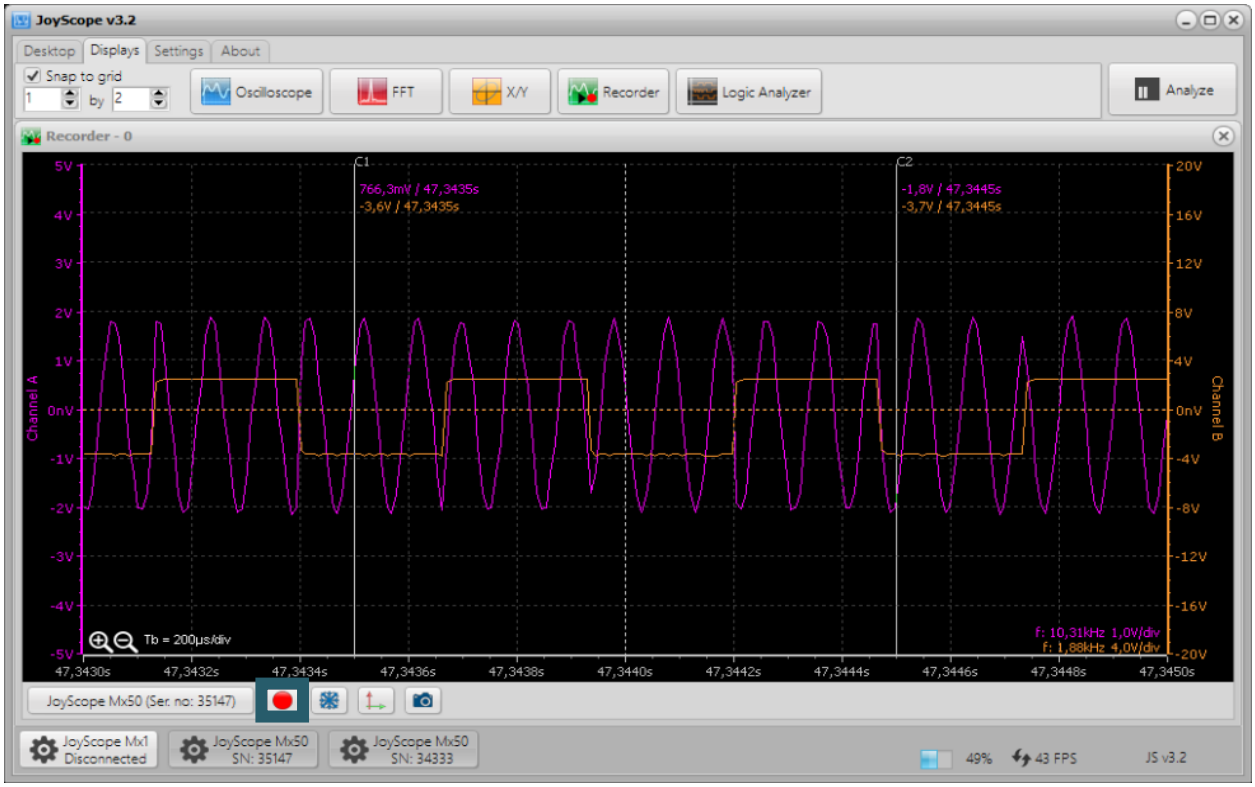
14. RECORDER

The **Analyze** button is limited by the size of the RAM memory installed in your PC on which the measurements are performed. To bypass these RAM limits, you can save the measurements in an external file on your hard disk. The total amount of measurement samples is then only limited by the capacity of your hard disk, SSD or USB drive.

To start the Recorder module of the JoyScope software, click on the **Recorder** icon in the **Displays** tab.



The standard oscilloscope window opens and offers you the **Recorder** button, by which you can use to start recording the measurement data.

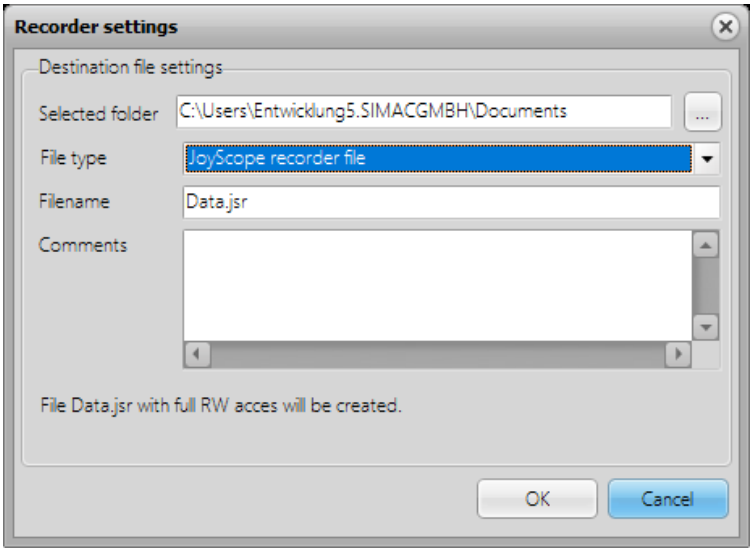


NOTE: The record button is only visible when the source device is in pipe mode. Recording is not possible in buffer mode.

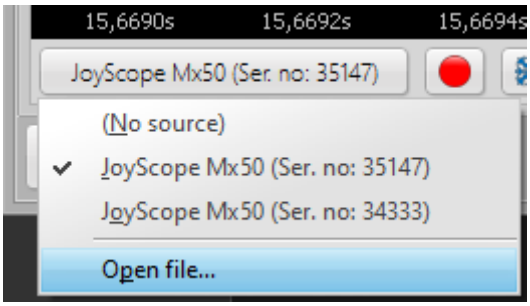
Captured samples can be saved either as *a JoyScope recorder file* or as a *PCM* file (pulse code modulated data). *PCM* files are intended for use in third-party software and cannot be opened in the JoyScope application. When saving to *PCM*, a *.txt* header file is also created containing the following information:

- Number of channels,
- Data size per channel,
- Bit order,
- Sampling frequency etc.

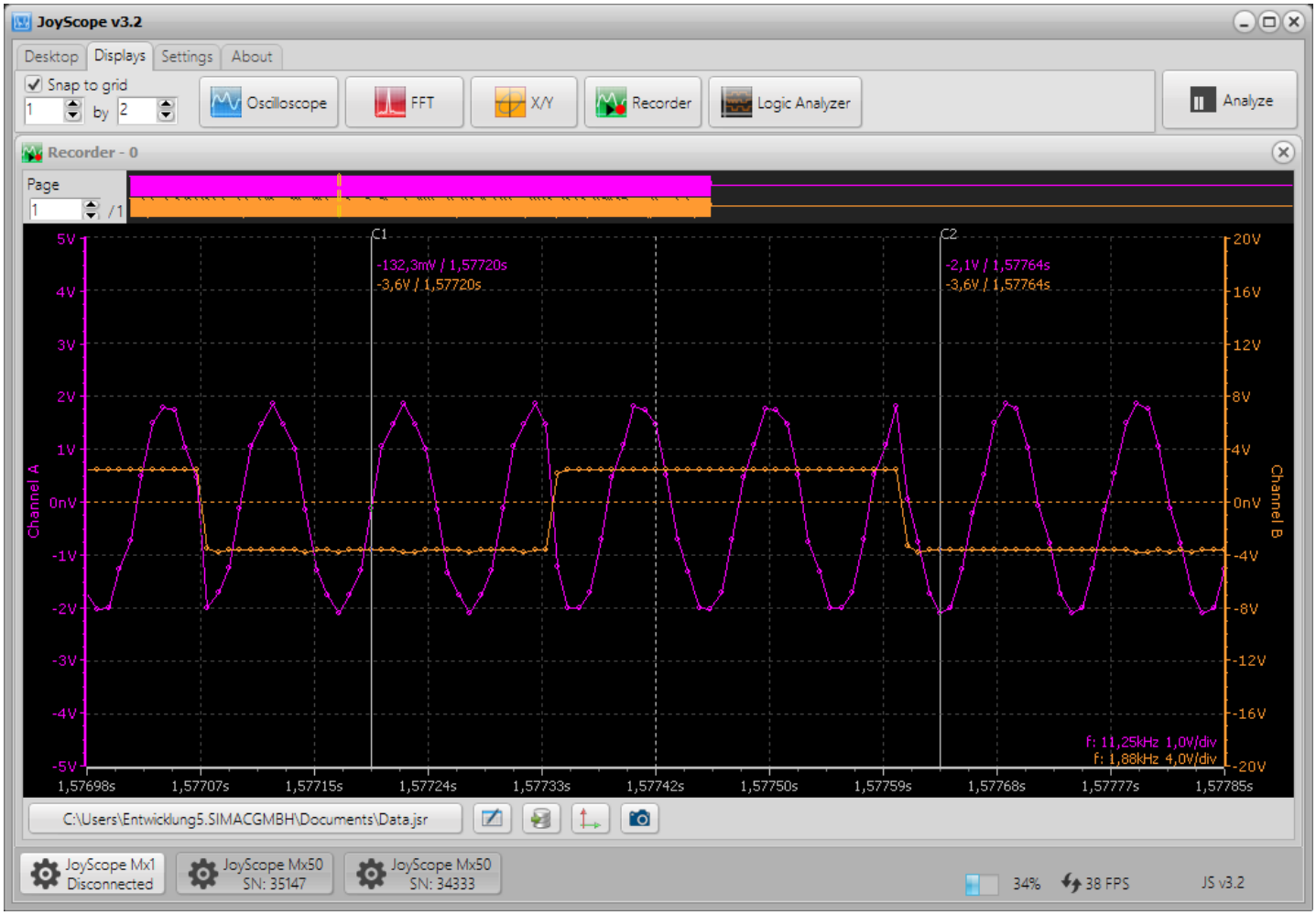
This information helps third-party software to interpret the *PCM* file correctly.



After recording, you can end it by pressing the Record button again. Only the *JoyScope recorder file* can be opened in the JoyScope software. To load the file, click on the source and select **Open file...**. Then locate the *JoyScope recorder file* on your hard disk and open it for further analysis in the software.



After opening the file, you can switch between the pages and adjust the size of the overview window. If you move the mouse pointer over the waveform, you can change the time base with the mouse wheel and move the waveform to the left and right by dragging it with the left mouse button. The data table and export functions are available to you in the same way as in the oscilloscope display.



15. LOGIC ANALYZER

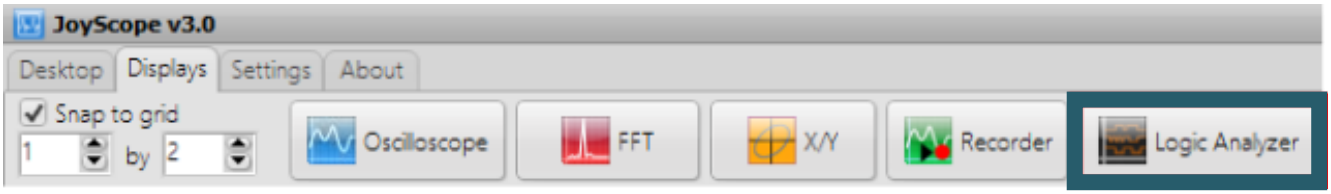
The logic analyzer is an electronic measuring instrument that makes it possible to record and display several signals from a digital circuit simultaneously. With this powerful tool, you can monitor many digital signals in parallel and carry out precise time measurements. time measurements. It is particularly useful for various tasks in electronics development, testing and repair, such as:

- Analyzing timing errors
- Checking timing diagrams
- Observing timing relationships between many signals
- Decoding information on data buses
- Displaying traces in state machines

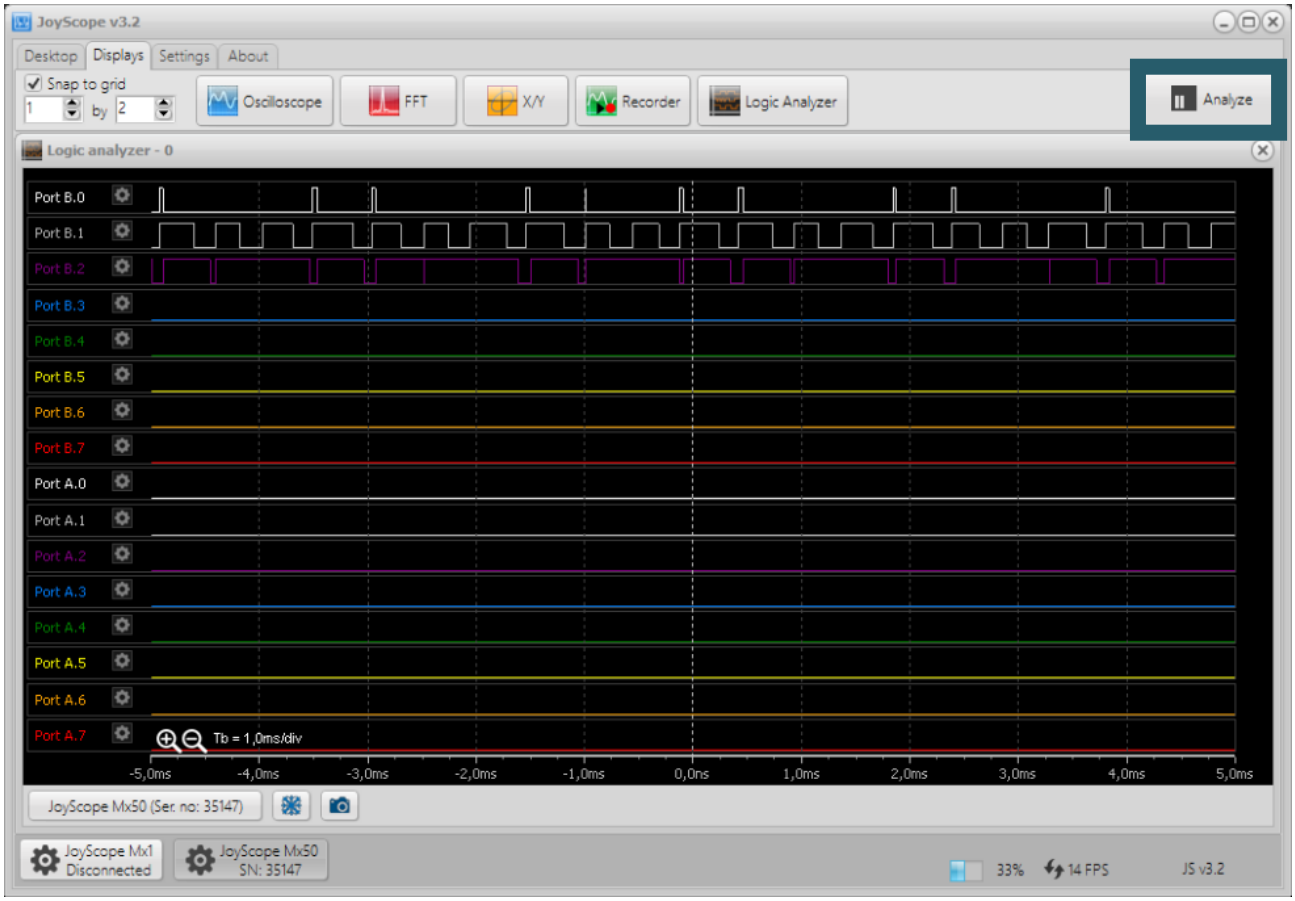
With JoyScope you can display, capture and analyze up to 16 digital channels simultaneously in the range of 0 to +5V.

For this example, we used a Raspberry Pi Pico that can be programmed with the user-friendly **ThonnyIDE**. To read out the digital outputs of the Pico, we used one of the two 60 MHz probes supplied.

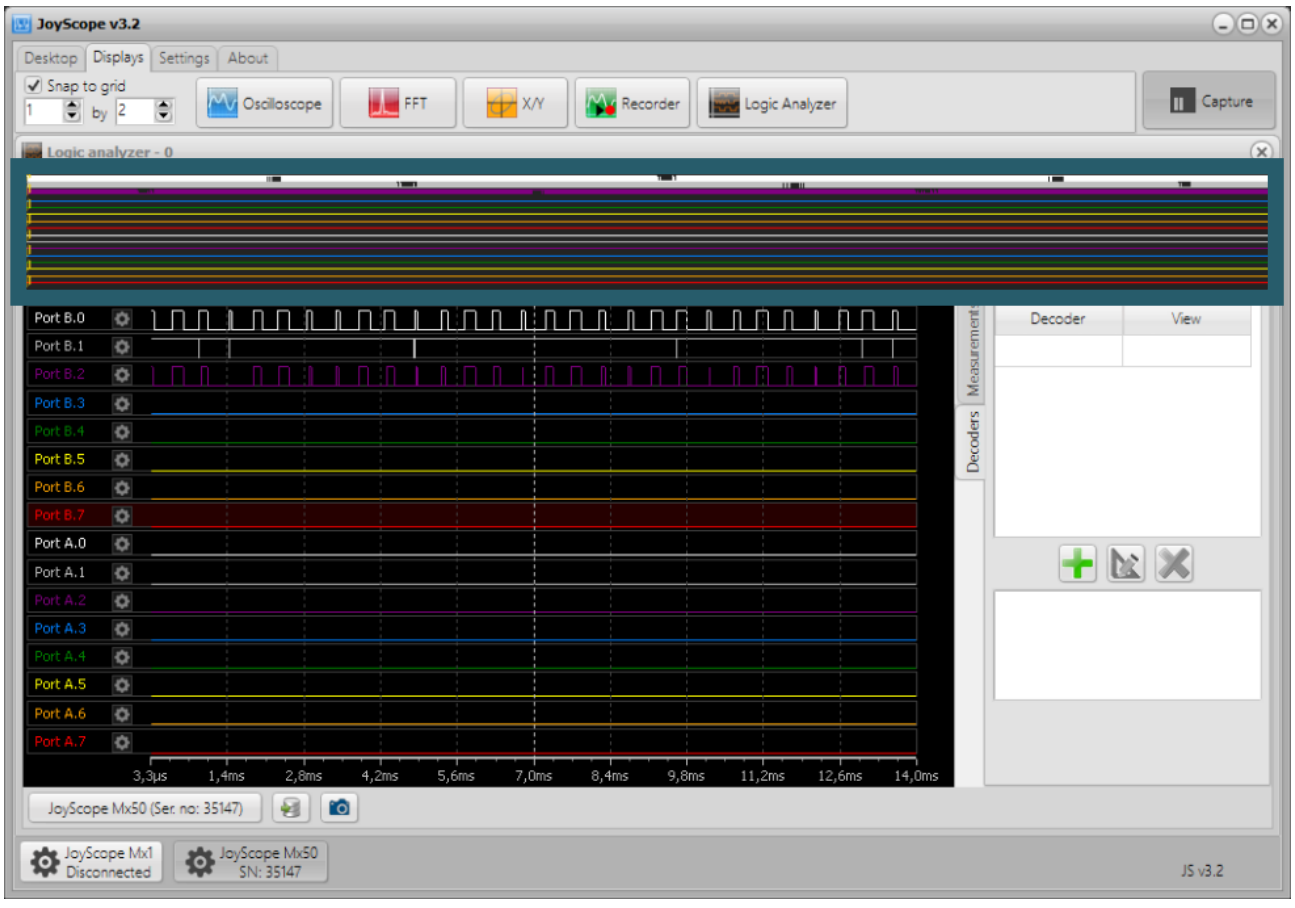
Start the **Logic Analyser** by clicking on the icon on the **Displays** tab of the JoyScope software:



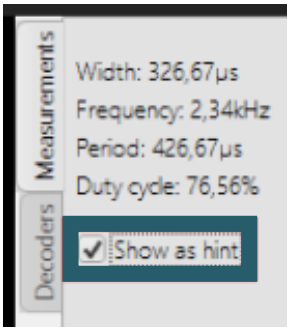
Below you can see an example of the display of the digital ports of a Raspberry Pi Pico. As in oscilloscope mode, you can click the **Analyze** button to view the captured data in detail.



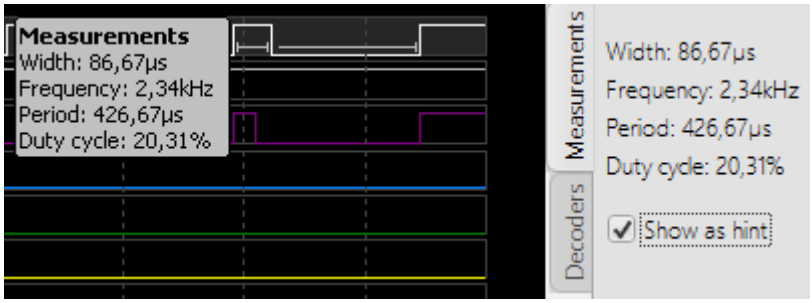
In analysis mode, you can use the yellow selector at the top to select a specific area of the recorded data for a more detailed examination. Adjust the size of the yellow selector using the mouse wheel and place it in the desired position.



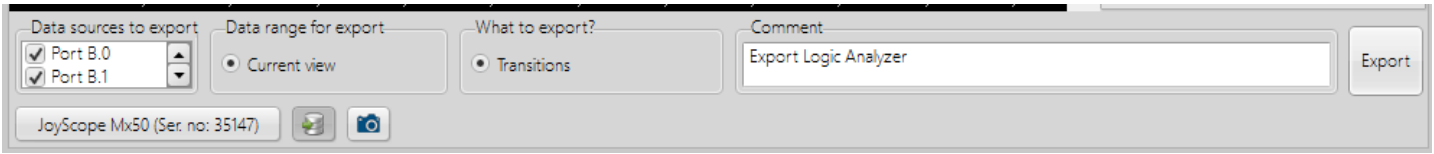
There are two tabs on the right-hand side of the logic analyzer display. Under the **Measurements** tab, you can view values such as Width, Frequency, Period and Duty Cycle. These values adjust dynamically as you move the mouse pointer over the diagram.



You can then move the mouse pointer over the corresponding part of the rectangular waveform to display additional details.



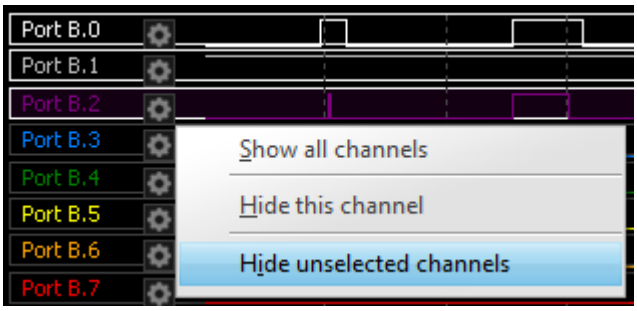
The data selected in the overview window can be exported to an external file, whereby various file formats are available.



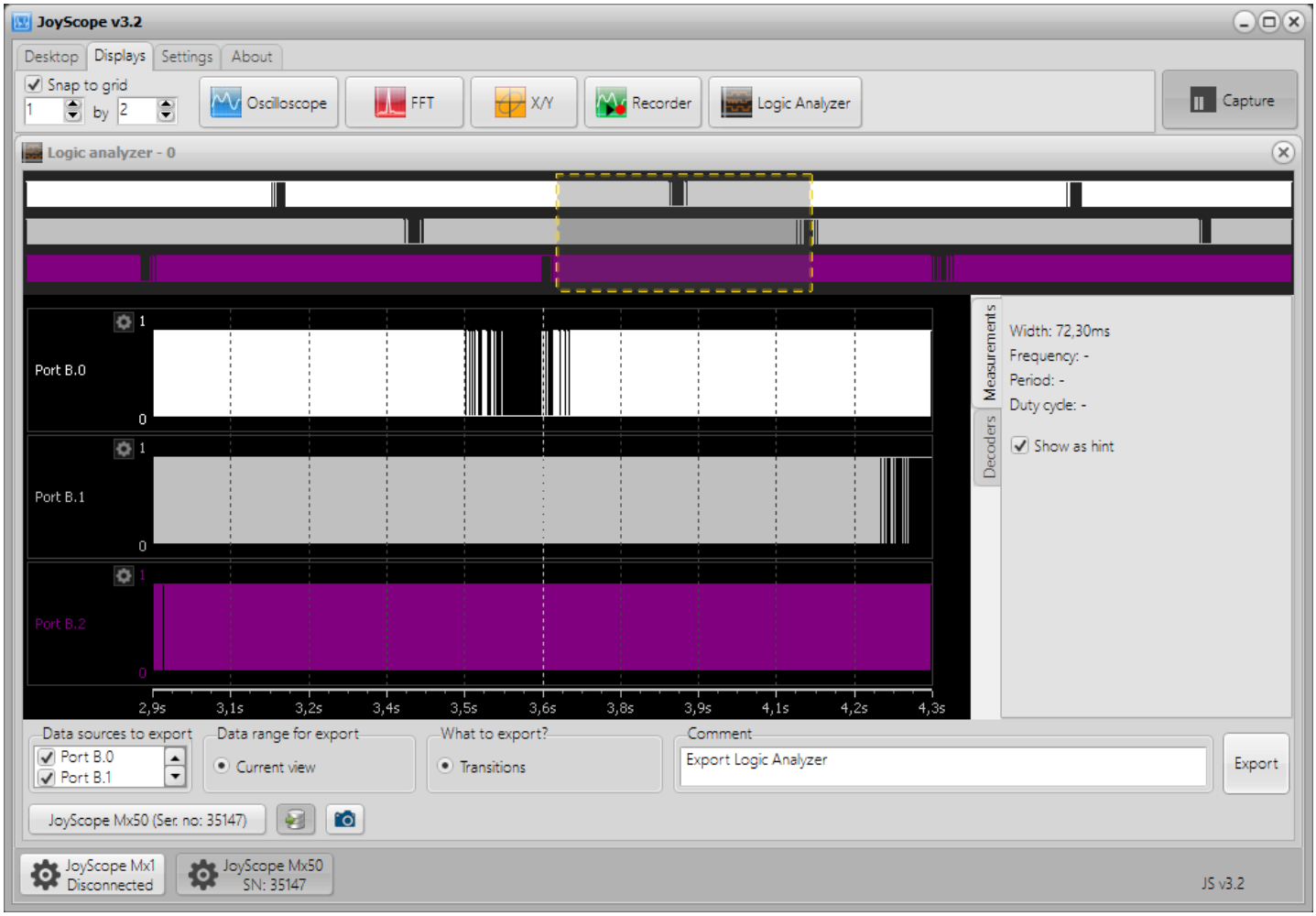
Each channel can be activated or deactivated by simply clicking on it. To select multiple channels, click on Port B.1, hold down the CTRL key and then click on Port B.2 and B.3. This selects all three channels. Now click on any symbol for the channel settings. Various options are displayed in the pop-up menu that then appears:

- **Show all channels** (You can show all channels if any are hidden.)
- **Hide this channel** (You can also hide one of the selected channels.)
- **Hide unselected channels** (You can hide all other unselected channels.)

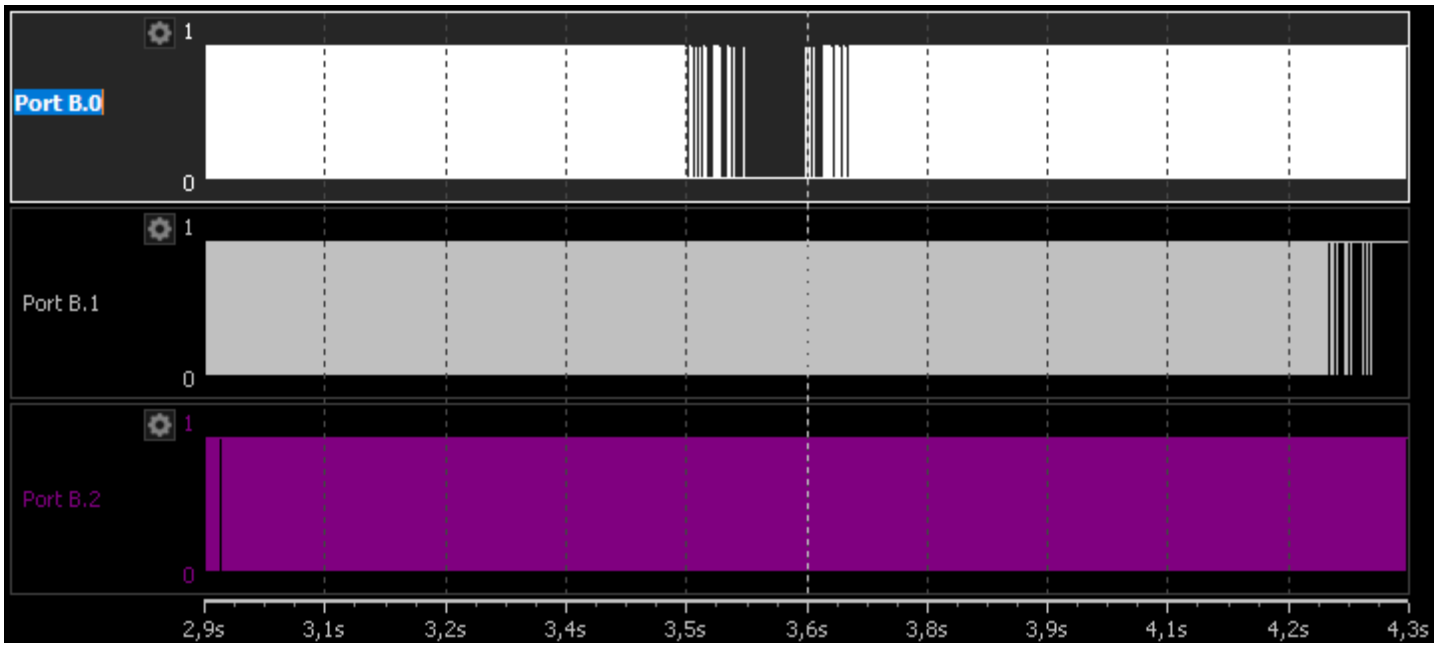
In our case, we click on **Hide unselected channels**.



The overview and the main display are now updated so that only the channels Port B.1, B.2 and B.3 are visible.



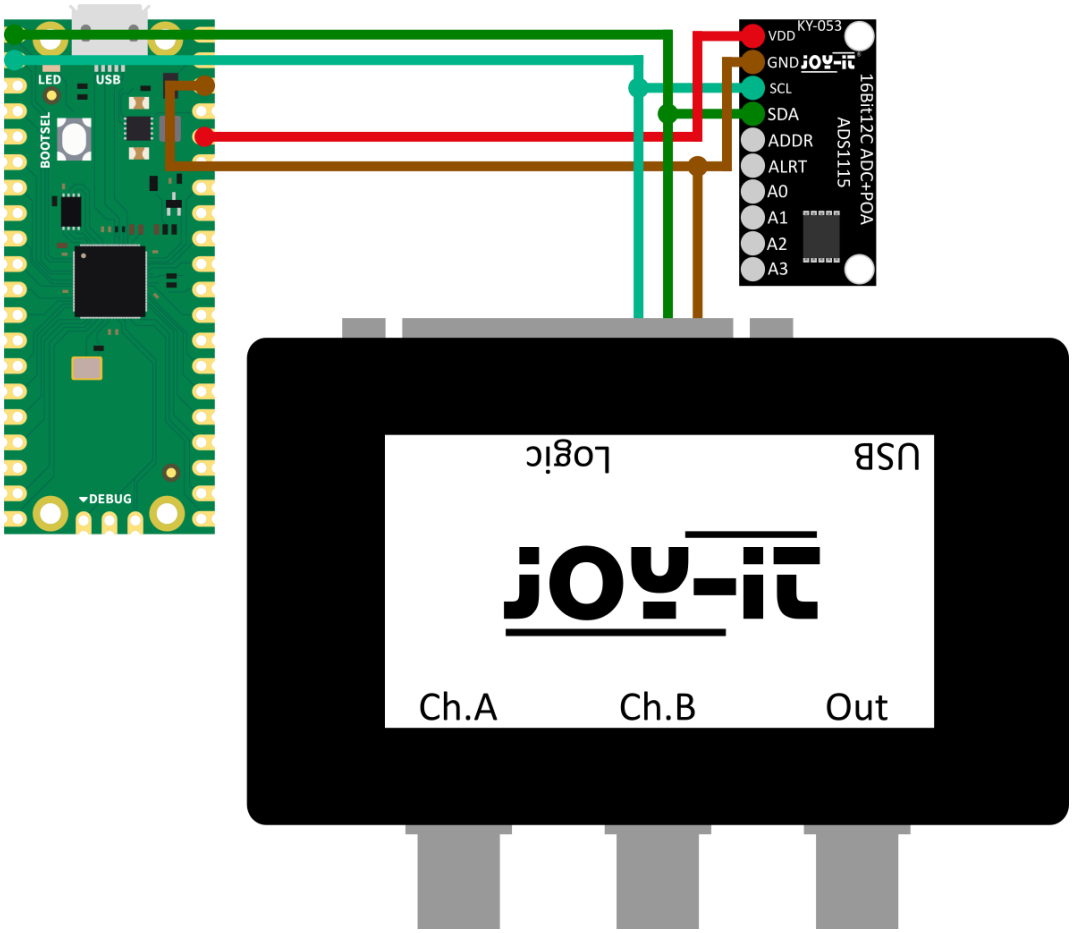
Tip: Double-click on the channel name to rename channels.



15.1 I2C decoder

An Inter-Integrated Circuit (I2C) interface simplifies the hardware structure of electronic devices as it only requires two wires: one for clock synchronization and one for data transmission. There is one master and several slaves in an I2C bus. Typically, I2C buses operate at frequencies up to 100 kHz, but can reach higher frequencies if required. Typically, 7-bit addresses are used, which are actually 8-bit, as the first bit indicates whether the master should read or write the slave address. The JoyScope Mx50 can decode up to 8 I2C buses, with each bus supporting up to 128 devices - allowing a total of up to 1024 devices to be connected.

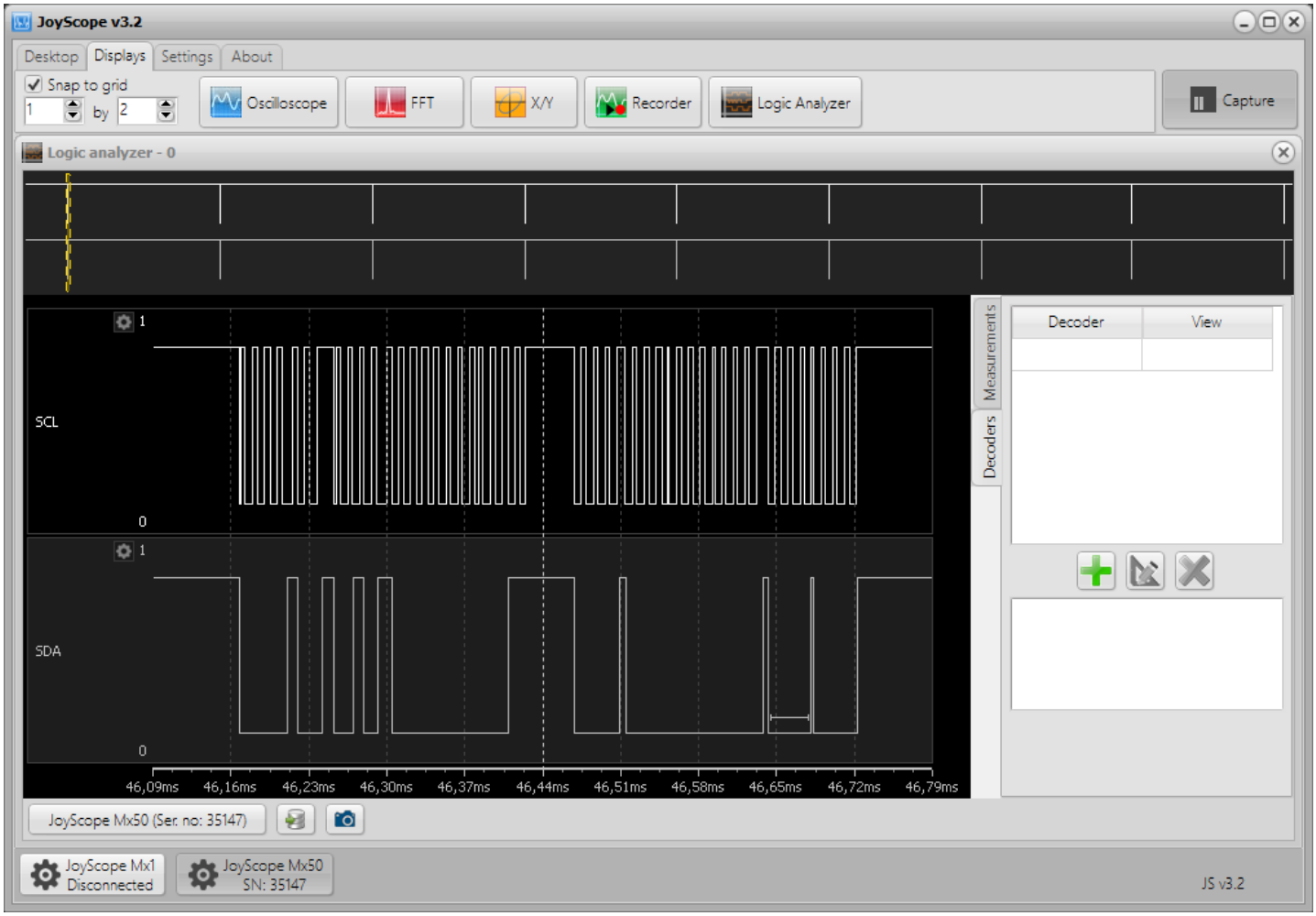
To test the functionality of the I2C decoder, you can, for example, connect a Raspberry Pi Pico to an I2C sensor and tap into the clock and data lines of the I2C bus. These lines can then be connected to the inputs of the logic analyzer.



As soon as the device has been set to digital mode and the logic analyzer window has opened, click on the **Analyze** button.

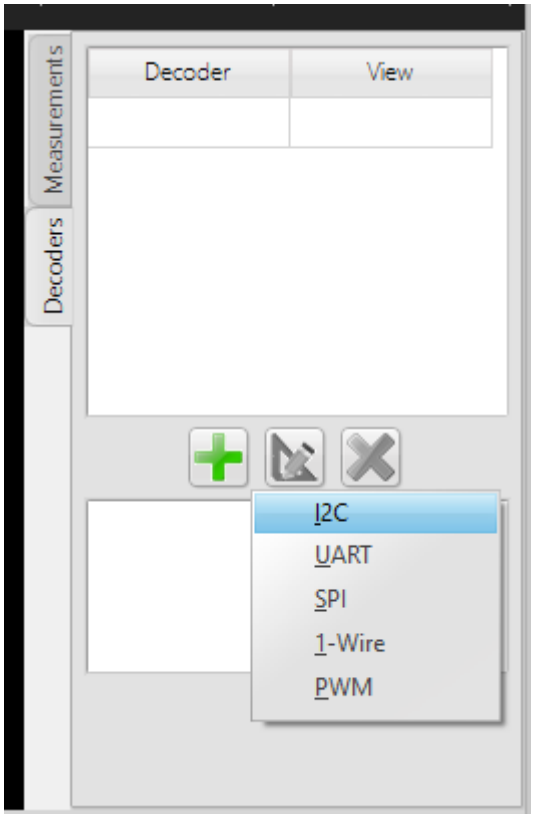
To ensure the quality of the measurements, the sampling rate must be higher than the signal frequency or bus clock rate. As I2C is set to 100 kbit/s as standard (other speeds are also supported), we recommend a sampling rate of 500 kS/s. After about one second, when the buffers are filled, press the **Analyze** button to switch to analysis.

Use the yellow selection button in the overview and the mouse pointer to select a specific section of the data for analysis and decoding. To rename a pin, you can double-click on its name. The second pin represents the data line.

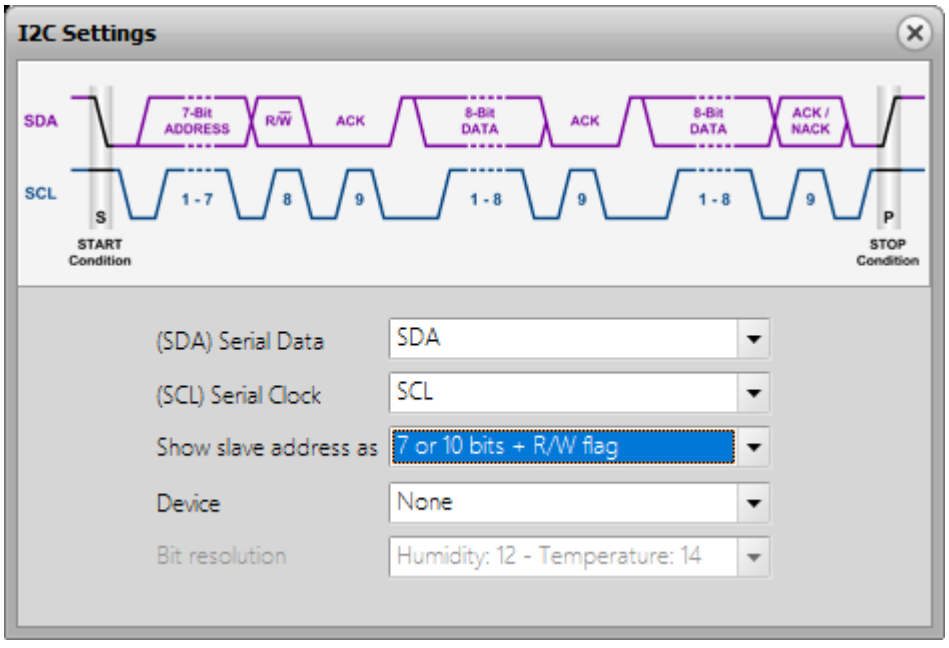


To add an I2C decoder, click on the + button under the **Decoder** tab in the I2C panel.

Please note that the JoyScope can only successfully decode 8 or 10 bit sensors.



The application offers parameters for data (SDA) and clock (SCL) as well as an option to select the number of bits for displaying the slave addresses. It also supports a wide range of I2C devices, including SHT20, SHT21, SHT25, STS21, HTUA21A, HTU20D, HTU21D, HTU3800, Si7006, Si7020, Si7021 and Si7013.



If you name your pins with the I2C standard names (SCL, SDA), the decoder settings window automatically recognizes the lines and uses them for decoding. Otherwise, you must select the pins manually as data sources.

Under **View**, you can specify the format in which the decoded data is to be displayed. You can choose between HEX, DEC, CHAR, BIN, HEX & CHAR and DEC & CHAR. The measured values can either be displayed in the side window or appear as a hint when you move the mouse pointer over the I2C data.

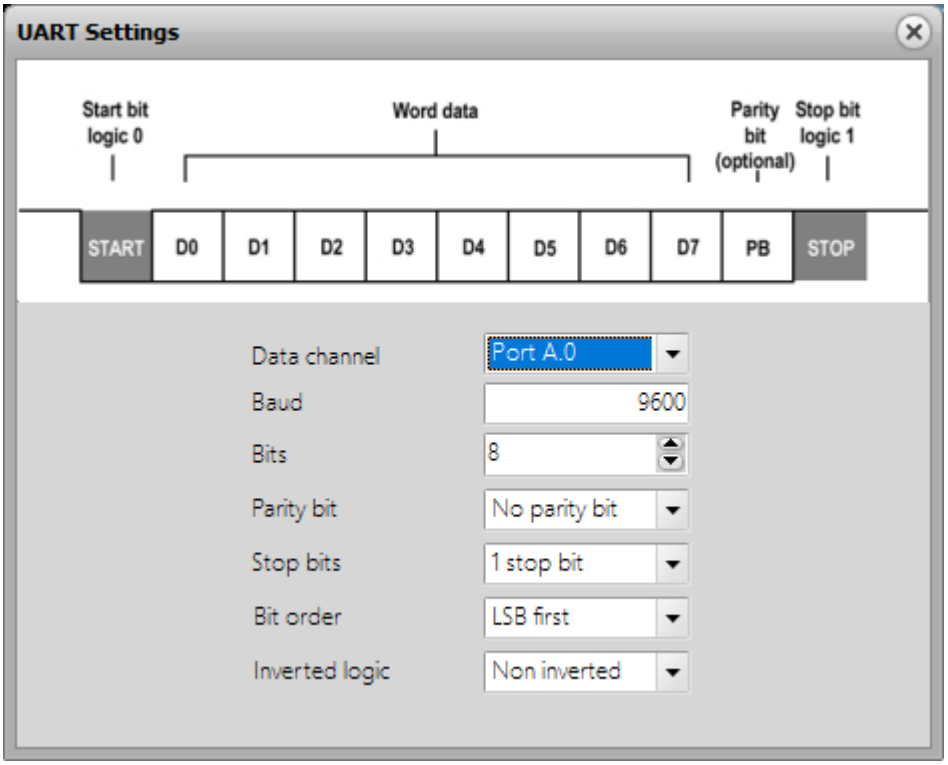
In the display window you can see:

- Start bit
- Master read initialization
- Address from which the master wants to read
- Acknowledge bit
- Transferred data
- Stop bit

Where small squares are displayed, there is not enough space for the complete packet description. If you move the mouse over the square, the complete description is displayed in the hint field.

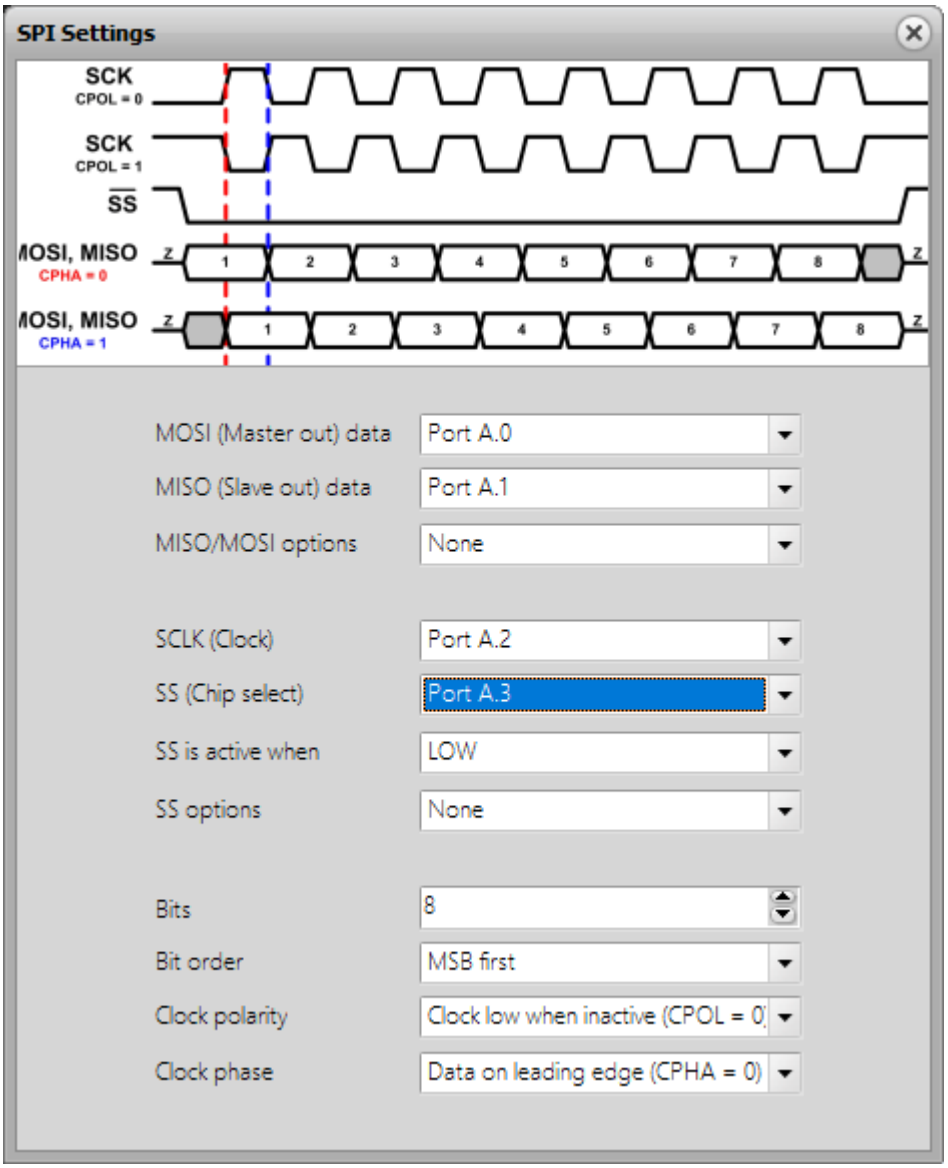
15.2 UART decoder

A UART decoder (Universal Asynchronous Receiver/Transmitter) interprets the information transmitted via a data channel by synchronizing the transmission speed with a baud rate. It reads each data packet, which is usually configured with a specific data length (in bits). It checks the accuracy using a parity bit, determines the end of the data packet with a stop bit and decodes the bit sequence according to the specified bit order (LSB first or MSB first). The decoder also adapts to the logic of the signal and distinguishes between inverted logic, in which the logic levels are reversed, and non-inverted logic in order to interpret the transmitted data precisely.



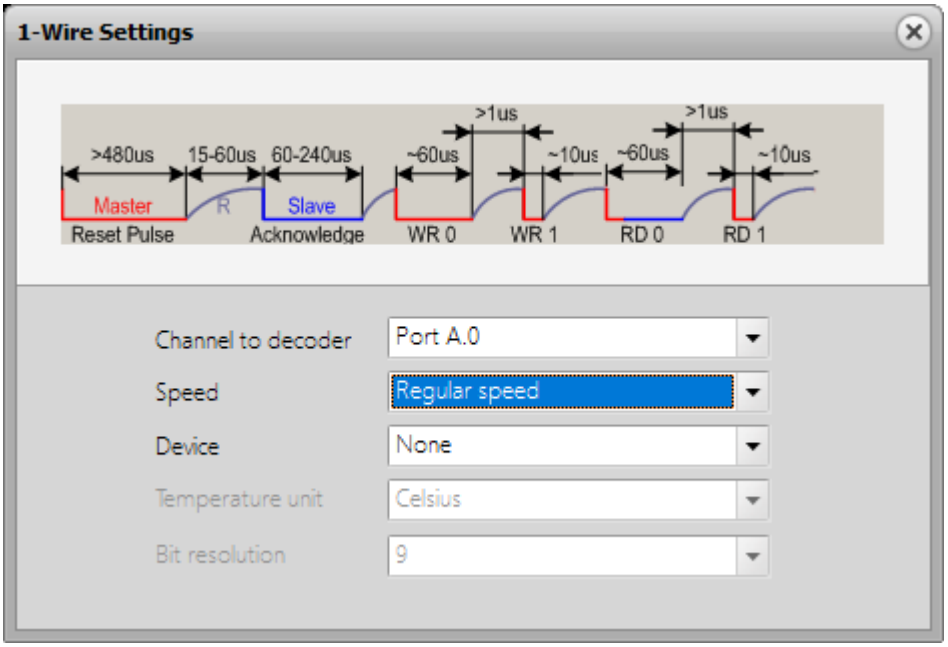
15.3 SPI decoder

An SPI (Serial Peripheral Interface) decoder analyzes the communication between devices using certain signals: MOSI (Master Out Slave In) transmits data from the master to the slave, MISO (Master In Slave Out) sends data from the slave to the master, and SCLK (Serial Clock) takes care of the timing. The active device is selected via SS (Chip Select), whose logic can be active-low or active-high. In addition, the system allows configurations with multiple SS options for different slaves. The communication protocol also defines the data length (in bits), the bit sequence (MSB first or LSB first), the clock polarity (idle state of the clock) and the clock phase (sampling edge) to ensure synchronized data transmission between the devices.



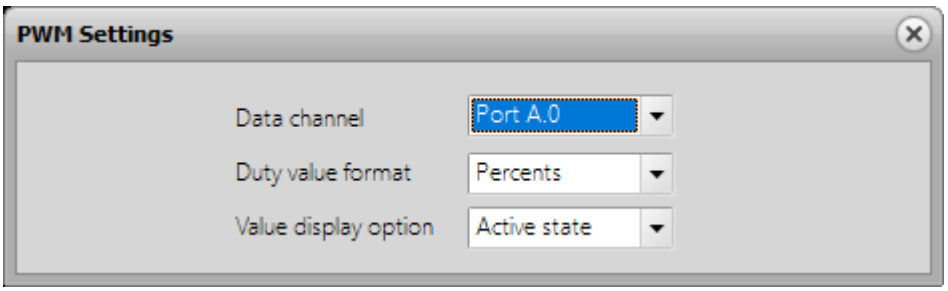
15.4 1-Wire-Decoder

A 1-wire decoder interprets data transmitted over a single channel and is designed to communicate with devices that can transmit serial data at low speed, such as temperature sensors. It configures the speed of data transmission (Speed), identifies the connected device and interprets the data according to the device's protocol, including temperature readings in a specific unit (e.g. Celsius or Fahrenheit). The decoder also takes into account the bit resolution of the data, which determines the accuracy of the measurements reported by the device.



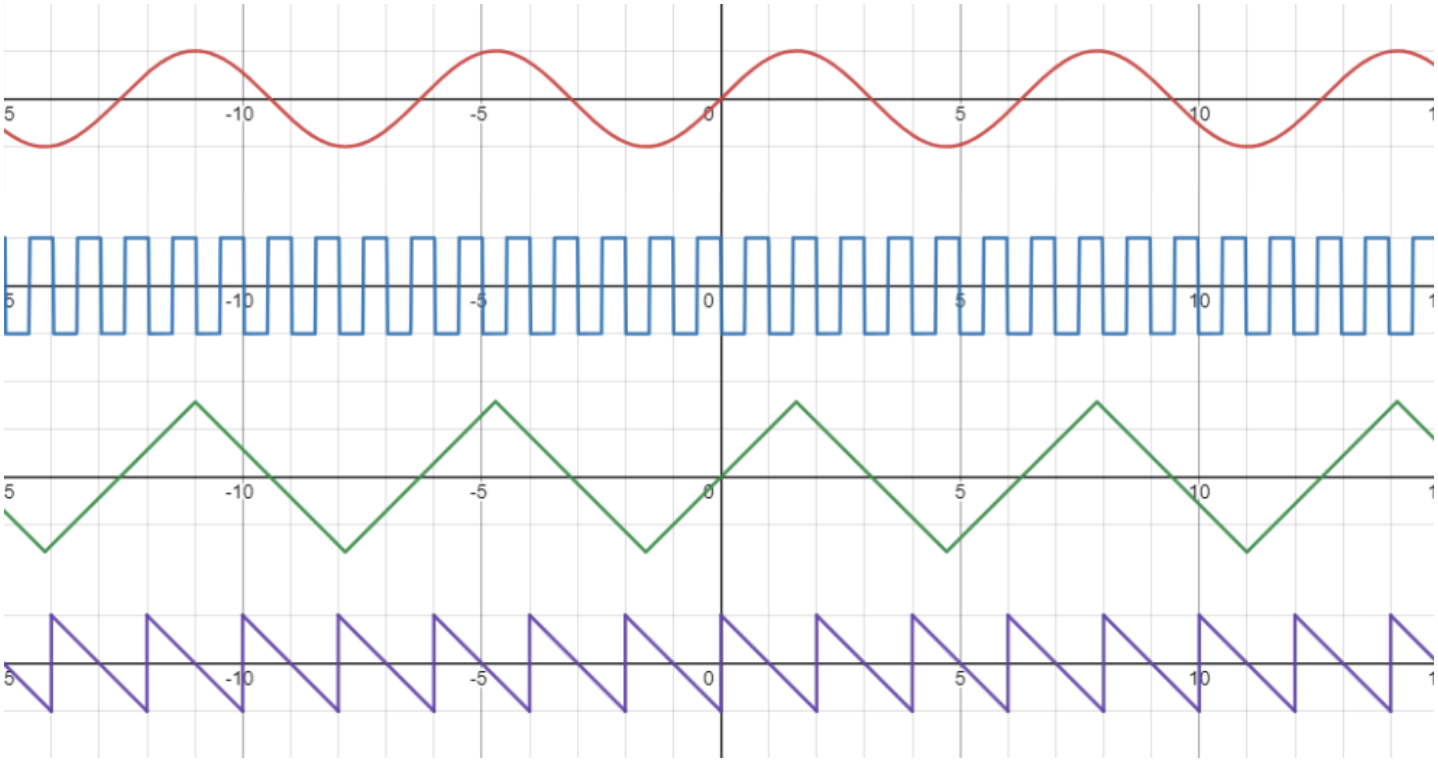
15.5 PWM-Decoder

A PWM (pulse width modulation) decoder analyzes signals on a data channel to determine the ratio of the "on" time to the total cycle time, known as the duty cycle. This value is expressed in a specific format (e.g. percentage or ratio). The decoder interprets the duty value of the signal to control the behavior of the device or to display data. Various options are available to display this value, for example in the active or passive state.

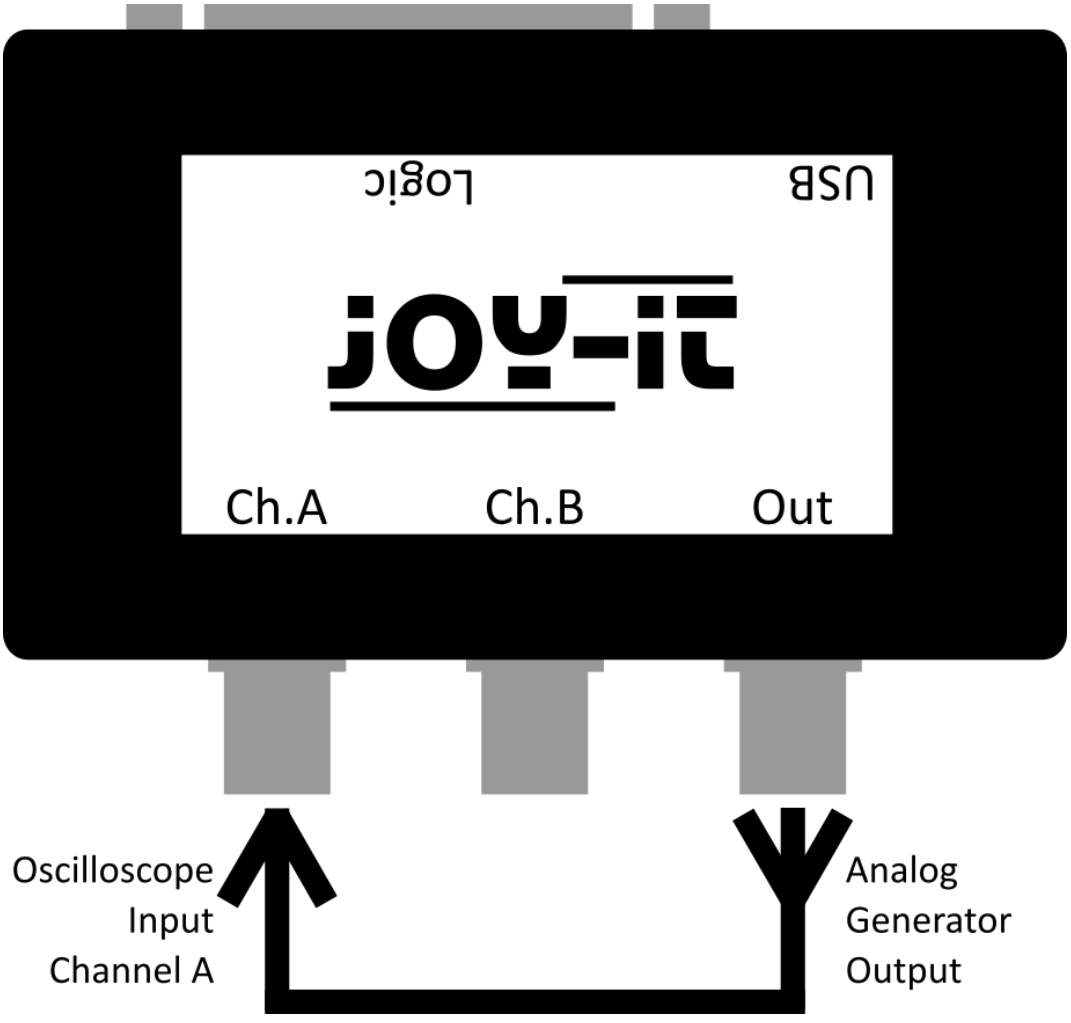


16. FUNCTION GENERATOR

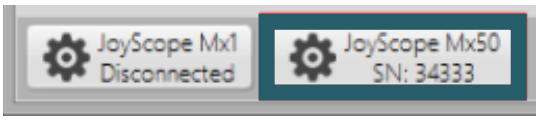
The function generator can generate various types of periodic electrical waveforms over a wide frequency range, from 10 Hz to 12.5 kHz. It is capable of generating sine, square, triangular and sawtooth waves (see below). Function generators are used in the development, testing and repair of electronic devices, for example as a signal source for testing amplifiers. However, the analog generator can only be used if the device is set to analog DAQ mode.



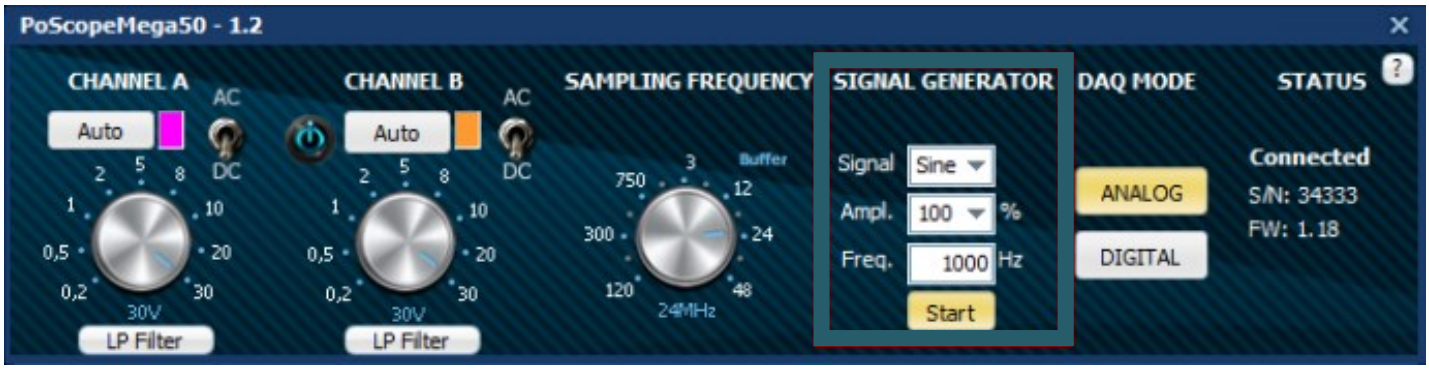
To demonstrate the basic use of the function generator, we have connected two probes to the JoyScope Mx50: the first to the output of the function generator and the second to channel A of the oscilloscope. We connected the signal and ground connections of the two probes together and set the function generator to a frequency of 5 kHz.



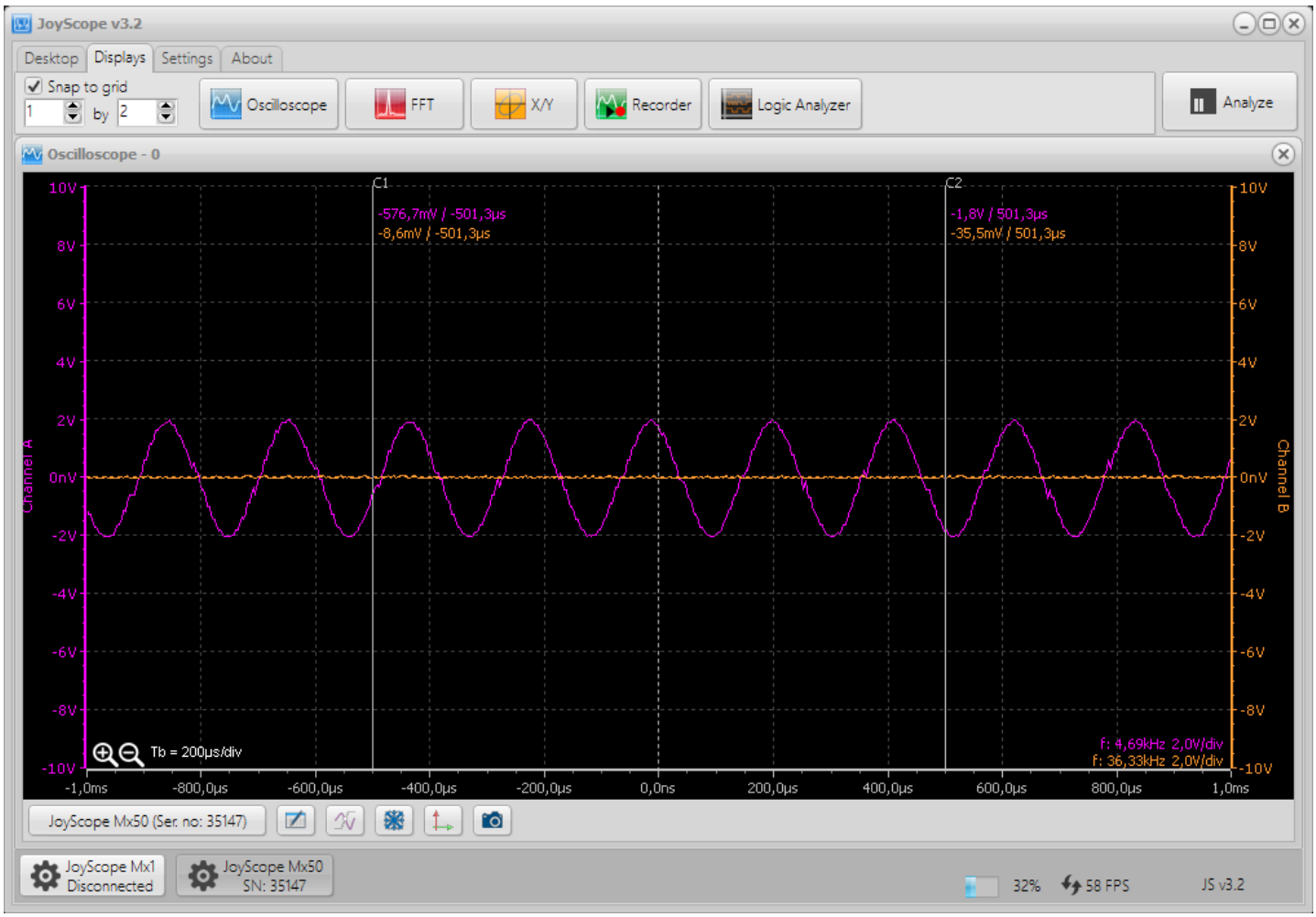
Open the JoyScope Mx50 control GUI by clicking on the icon in the device buttons area of the JoyScope software.

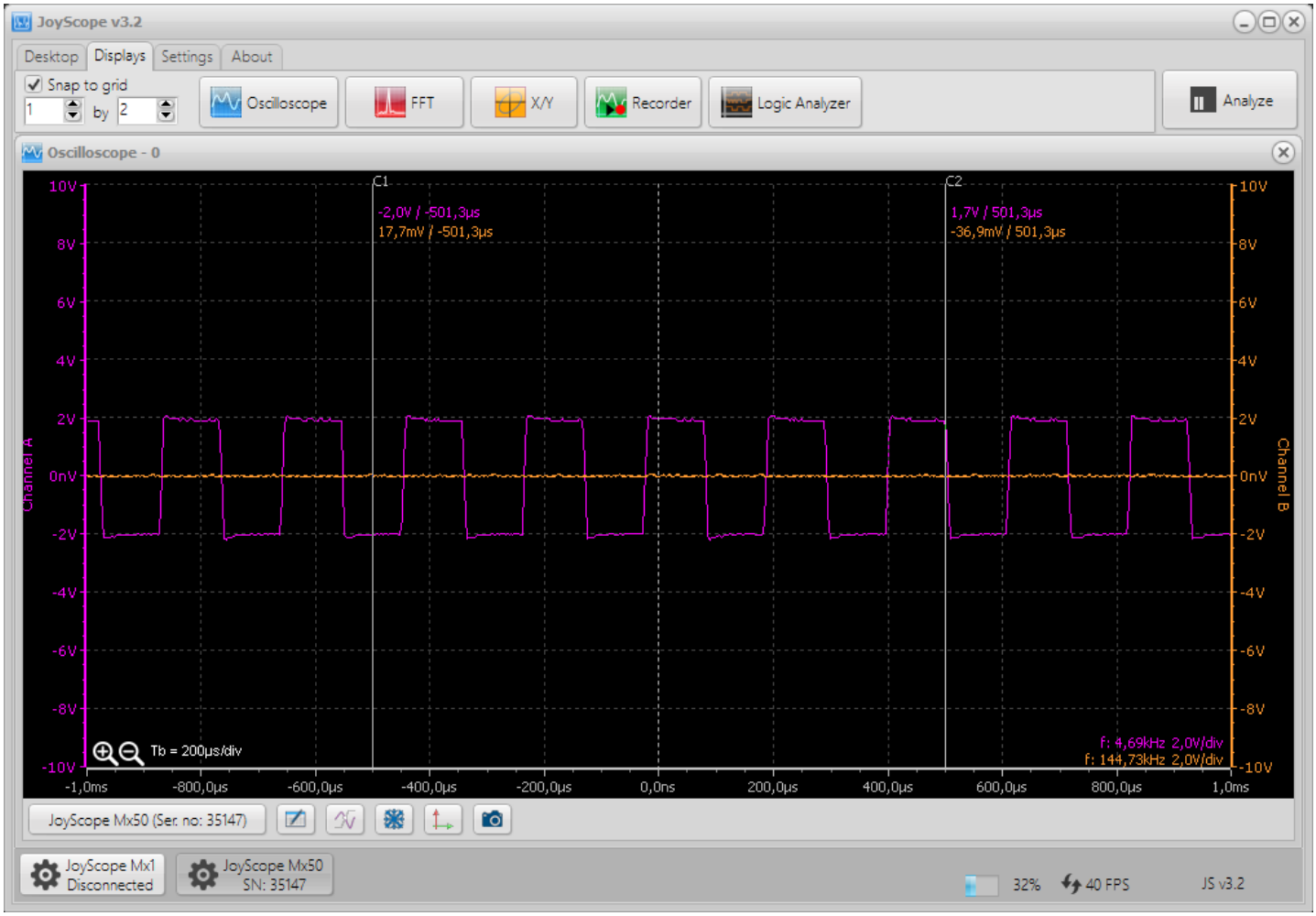


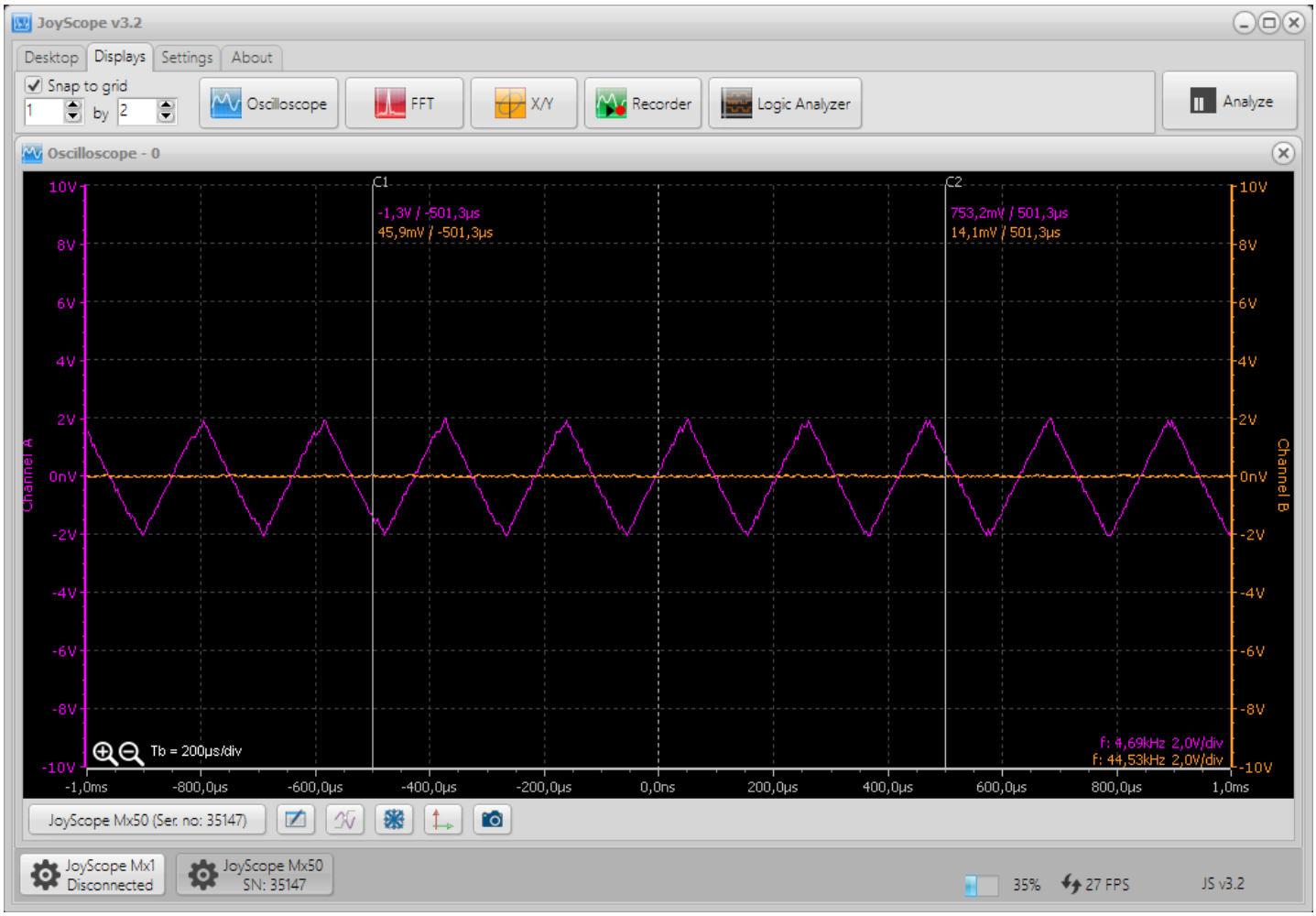
The analog generator controls are located on the right-hand side of the JoyScope Mx50 user interface. Here you can set the signal shape (sine, square, triangle, sawtooth), the frequency (up to 12.5 kHz) and the amplitude, which is specified as a percentage of 1.8 Vpp.

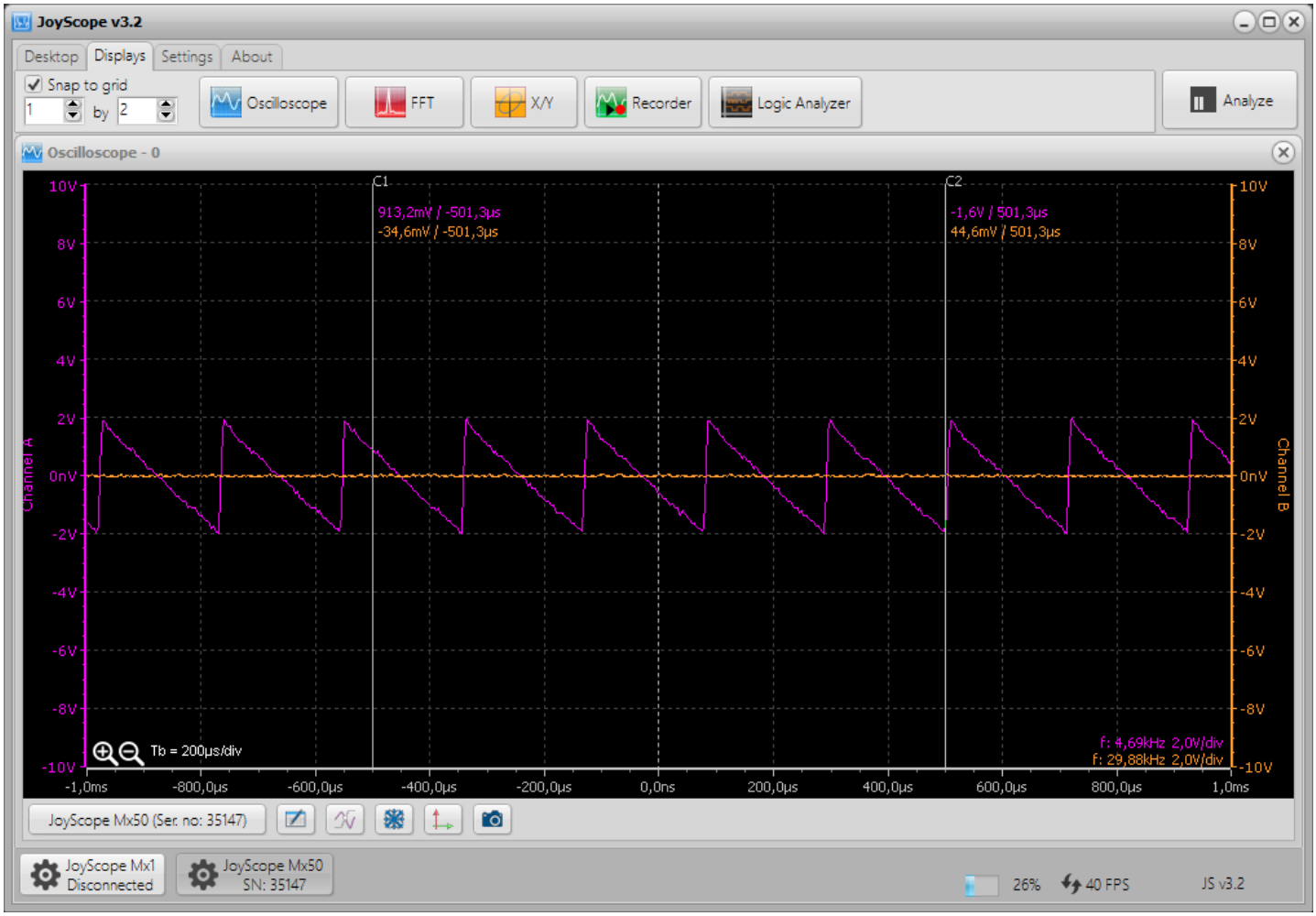


The waveforms on the oscilloscope display change according to the selected waveform in the analog generator section of the JoyScope Mx50 controller.




JoyScope v3.2
Desktop
Displays
Settings
About
Snap to grid
1
by
2
Oscilloscope
FFT
X/Y
Recorder
Logic Analyzer
Analyze
Oscilloscope - 0
C1
-2,0V / -501,3µs
17,7mV / -501,3µs
C2
1,7V / 501,3µs
-36,9mV / 501,3µs
10V
8V
6V
4V
2V
0nV
-2V
-4V
-6V
-8V
-10V
Channel A
Channel B
Tb = 200µs/div
f: 4,69kHz 2,0V/div
F: 144,73kHz 2,0V/div
-1,0ms
-800,0µs
-600,0µs
-400,0µs
-200,0µs
0,0ns
200,0µs
400,0µs
600,0µs
800,0µs
1,0ms
JoyScope Mx50 (Ser. no: 35147)
JoyScope Mx1
Disconnected
JoyScope Mx50
SN: 35147
32%
40 FPS
JS v3.2


JoyScope v3.2
Desktop
Displays
Settings
About
Snap to grid
1
by
2
Oscilloscope
FFT
X/Y
Recorder
Logic Analyzer
Analyze
Oscilloscope - 0
C1
-1,3V / -501,3µs
45,9mV / -501,3µs
C2
753,2mV / 501,3µs
14,1mV / 501,3µs
10V
8V
6V
4V
2V
0nV
-2V
-4V
-6V
-8V
-10V
Channel A
Channel B
Tb = 200µs/div
f: 4,69kHz 2,0V/div
f: 44,53kHz 2,0V/div
-1,0ms
-800,0µs
-600,0µs
-400,0µs
-200,0µs
0,0ns
200,0µs
400,0µs
600,0µs
800,0µs
1,0ms
JoyScope Mx50 (Ser. no: 35147)
JoyScope Mx1
Disconnected
JoyScope Mx50
SN: 35147
35%
27 FPS
JS v3.2


JoyScope v3.2
Desktop
Displays
Settings
About
Snap to grid
1
by
2
Oscilloscope
FFT
X/Y
Recorder
Logic Analyzer
Analyze
Oscilloscope - 0
C1
913,2mV / -501,3µs
-34,6mV / -501,3µs
C2
-1,6V / 501,3µs
44,6mV / 501,3µs
10V
8V
6V
4V
2V
0nV
-2V
-4V
-6V
-8V
-10V
Channel A
Channel B
Tb = 200µs/div
f: 4,69kHz 2,0V/div
f: 29,88kHz 2,0V/div
-1,0ms
-800,0µs
-600,0µs
-400,0µs
-200,0µs
0,0ns
200,0µs
400,0µs
600,0µs
800,0µs
1,0ms
JoyScope Mx50 (Ser. no: 35147)
JoyScope Mx1
Disconnected
JoyScope Mx50
SN: 35147
26%
40 FPS
JS v3.2

17. FURTHER INFORMATION

Our information and take-back obligations according to the Electrical and Electronic Equipment Act (ElektroG)

Symbol on electrical and electronic equipment:
This crossed-out dustbin means that electrical and electronic appliances do not belong in the household waste. You must return the old appliances to a collection point.
Before handing over waste batteries and accumulators that are not enclosed by waste equipment must be separated from it.

Return options:
As an end user, you can return your old device (which essentially fulfills the same function as the new device purchased from us) free of charge for disposal when you purchase a new device.
Small appliances with no external dimensions greater than 25 cm can be disposed of in normal household quantities independently of the purchase of a new appliance.

Possibility of return at our company location during opening hours:
SIMAC Electronics GmbH, Pascalstr. 8, D-47506 Neukirchen-Vluyn, Germany

Possibility of return in your area:
We will send you a parcel stamp with which you can return the device to us free of charge. Please contact us by email at Service@joy-it.net or by telephone.

Information on packaging:
If you do not have suitable packaging material or do not wish to use your own, please contact us and we will send you suitable packaging.

18. SUPPORT

We are also there for you after your purchase. If any questions remain unanswered or problems arise, we are also available by e-mail, telephone and ticket support system.

E-mail: **service@joy-it.net**
Ticket system: **http://support.joy-it.net**
Telephone: +49 (0)2845 9360 - 50

You can find our current opening hours at
www.joy-it.net/en/service

For further information please visit our website:
www.joy-it.net

Published: 2024.11.21