

Disco Oscilloscope Help

- Quick Start
- Introduction
- Features
- Technical parameters
- Installation
- Interface
- Menu
- Oscilloscope
 - Setup
- Spectrum Analyzer
 - Setup
- Recorder
 - Setup
- Logic Analyzer/Generator
 - Setup
 - Concatenation mode
 - Pins layout
- Reading modes
- The knob
- Filter settings
- Sounds
- Statistics
- Firmware upgrade
- Analysing of standard interfaces:
 - UART
 - SPI
 - I2C
 - 1-Wire
- Terminal
- Settings

Shortcut keys

Quick Start

Here are the steps to start working with the oscilloscope:

- Open the Oscilloscope application and select the Oscilloscope tab
- Set the Channel A knob (red) to the position 5V and the button below is "On"
- Connect an AA battery to the channel A probe (+ and the ground don't matter) and press Cyclic button
- You should see a signal on the screen.

To start working with the Logic Analyzer and Generator do the following:

- Carefully remove the plastic holders from the ribbon cables pins
- Using a bread-board connect the cables pins of one cable to each other of another (like 1 to 1, 2->2, 3->3, etc. The 1st pin is the far one from the blue pin, and the blue one is the ground)
- Connect the ribbon cables to the Bus A and Bus B connectors on the device
- Go to the Logic Analyzer tab
- Click G-button at the right bottom corner
- Using the mouse create the signals on the Bus B. Use left-click to set "1" and right-click to set "0" on the lines
- Click Run

At the Bus A you should see the signals generated on the bus B

Introduction in PC USB Oscilloscope DiSco

When developing or researching many electronic devices, electronics hobbyists frequently need to measure various signal parameters (voltage, frequency, timing, etc.) or view and record inputs and outputs, in order to analyze them and thus determine if the electronic circuit is operating as intended.

To gather measurement data to analyze digital-to-analog circuits, it is useful to use some specific electrical testing equipment, which include: - oscilloscope, - spectrum analyzer, - recorder, and - logical analyzer/generator.

In their conventional form, each of these devices requires a free interface connector (usually the LPT printer ports) and each device requires a separate power source. In addition, many conventional functional measurement instruments, e.g. those made by Tektronix and Agilent, are quite expensive, both for hobbyist electronics enthusiasts and moderate-sized businesses.

In recent times, the appearance of some new peripherals-rich microcontrollers with high-speed data serial communication channel support ([Universal Serial Bus: USB](#)) and no external power supply (power can be sourced directly for the USB port) has allowed the creation of a package combining all such functions at a low price.

Our PC USB Oscilloscope DiSco is this device.

Features

USB oscilloscope has the following operation modes:

- **2-channel oscilloscope:** up to 200KHz, marker measurements, triggering (with absolute, differential, external trigger modes) and adjustable pretrigger, signal voltage and frequency measurement, **filtering**, etc.
- **2-channel spectrum analyzer:** marker measurements, klirr-factor measurement, different window functions, **filtering**, etc.
- **2-channel recorder:** marker measurements, option to add 99 markers with comments, maximal, minimal and average voltage measurement for each channel, waveform record for dozens of hours, etc.
- **16 (8)-channel logic analyzer:** marker measurements, triggering (edge, level, mask) and adjustable pretrigger, external clocking (triggering), preset pulse miss, preset bit sequence/edge, decoding of the interface **UART, SPI, I2C, 1-Wire**, etc.
- **8-channel logic generator:** tabular waveform formatting or direct timing chart plotting with mouse on the screen, etc.

In addition, USB oscilloscope has the following features:

- **Add user comments** to the measurements (to be included in the saved data file).
- **Print all the measurements** to a printer attached to the PC.
- **Copy all the measurements** to the buffer.
- **Set the events and sounds** to standard Windows or user sound files.
- **Calculate various digital filters** and perform analog filtering.
- **Oscilloscope timing chart smoothing.**
- **Display the statistics** for all the channels of logical analyzer and generator.
- **Upgrade the device firmware** via the USB bus which allows supplementing the device's features with newly developed upgrades that may included 'fixes' and/or new features.

Technical parameters

| | | |
|---------------------------------------|---|--|
| Logic analyzer: | | |
| - number of channels | 16 (8 if logical generator is on) | |
| - sample rate | 1 KHz...8 MHz | |
| - memory depth | | |
| buffer reading (Fs=4-8 mHz): | 128 bit/channel | |
| buffer reading (Fs=2-2.66 mHz): | 1160 bit/channel | |
| buffer reading (Fs<=1 mHz): | 1544 bit/channel | |
| buffer reading in concatenation mode: | 1 Mbit/channel | |
| pipe reading (Fs < 500KHz): | 4K to 256M bit/channel | |
| - input voltage | 0...+5 V (available surge voltage protection) | |
| - triggering | for edges, mask, miss pulses, external clocking | |
| - clocking | internal / external | |
| - maximum input/output current | 10 mA | |
| | | |
| Oscilloscope, spectrum analyzer: | | |
| - number of channels | 2 | |
| - sample rate | 100 Hz...200 KHz | |
| - memory depth | | |
| buffer reading: | 1126 sample/channel (1 channel), 563 sample/ | |

| | | |
|--------------------------------|--|--|
| | channel (2 channels) | |
| pipe reading: | 64K sample/channel (1 or 2 channels) | |
| - input voltage | -20...+20 V (hardware 2 sub-band) | |
| - input impedance | 1 MOhm | |
| - ADC capacity | 10 bits | |
| - triggering | absolute (for rising/falling edge) differential (for difference between adjacent samples) external (for rising/falling edge of TTL levels) | |
| - window functions | Hamming, Hanning, Blackman, Blackman-Harris | |
| | | |
| Recorder: | | |
| - sample rate | 0.01 Hz...200 KHz | |
| - maximum record time | 24 hours at $F_s < 100$ Hz | |
| - input voltage | -20...+20 V (hardware 2 sub-band) | |
| - ADC capacity | 10 bit | |
| | | |
| Logic generator: | | |
| - number of channels | 8 | |
| - sample rate | 1 KHz...1 MHz | |
| - memory depth | 1544 bit/channel | |
| - output voltage | "0" - 0 V, "1" - 3.3 V | |
| - maximum input/output current | 10 mA | |

Installation

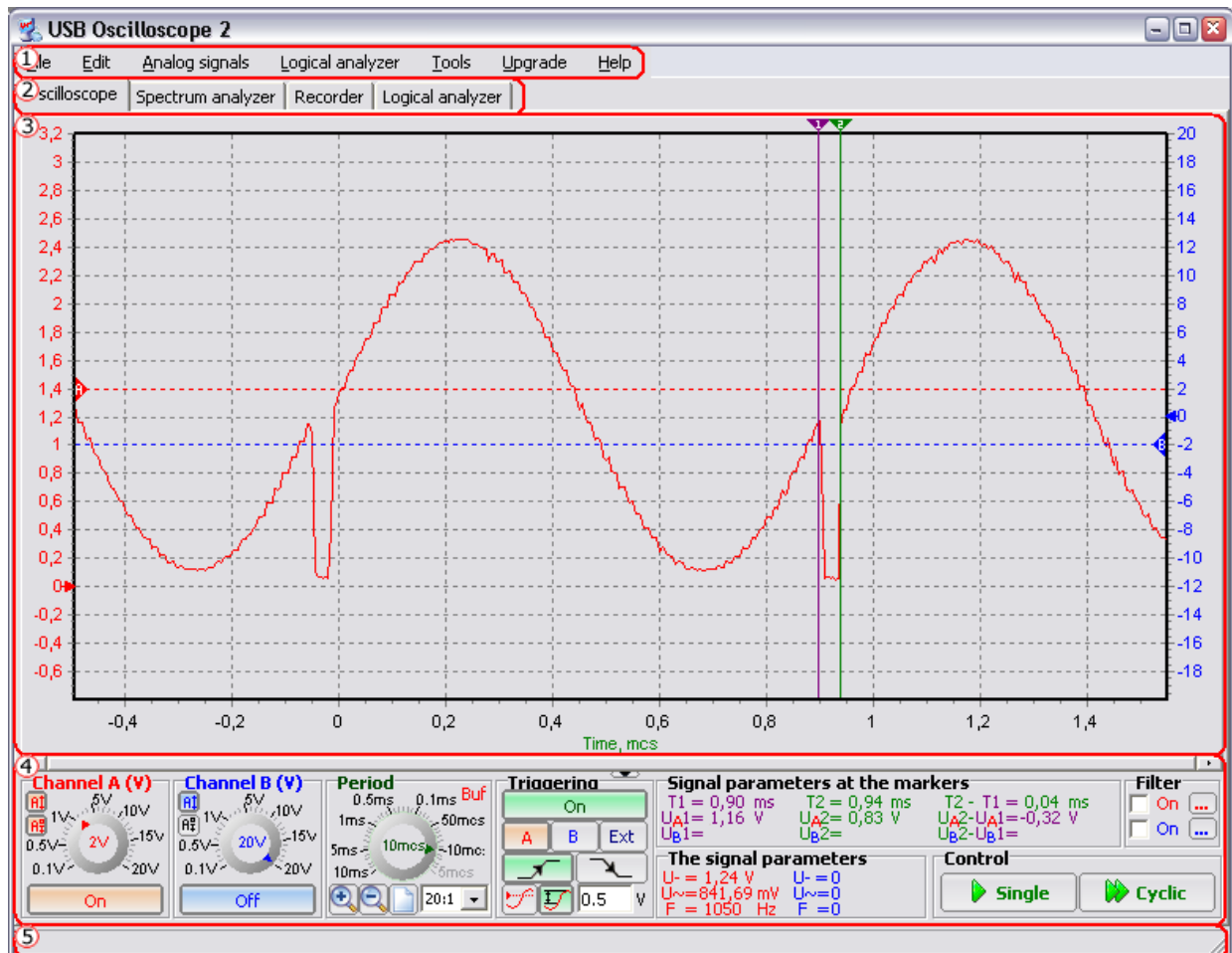
Upon extracting the distribution file the driver setup and shortcut creation window opens.



This window allows automatically installing the drivers (files `usb_osc.inf`, `windrvr6.inf` and `windrvr6.sys`) by copying three files in the corresponding system folders and registering them in the Windows registry. When installing the drivers it is only necessary to specify the operating system (whether NT-based, like Win2000 or WinXP) and start the corresponding *.bat file from the Driver folder (`install_win2k_XP.bat` or `install_win98-ME.bat`). If the automatic driver installation fails, then the drivers must be installed manually via Device Manager. See `ReadMe.txt` file for details about the drivers installation in the Driver folder.

In addition to the driver installation this window allows you to create the program shortcut on the Desktop as well as shortcut group (program shortcut, help files and `ReadMe.txt` file) in the "Start" menu under "Programs".

Interface



This figure displays the main program window (by clicking the window element in the above picture you can go to the corresponding description section).

Near the top is the **menu bar** ① below the window title. This lets you open files, save options, and perform some of the adjustments available for specific modes (that may also be available in the settings panel).

Four **mode tabs** ② are positioned beneath the menu bar that allow selecting the operational mode:

- oscilloscope
- spectrum analyzer
- recorder
- logical analyzer

(Note that you should 'stop' measuring in one mode before switching to

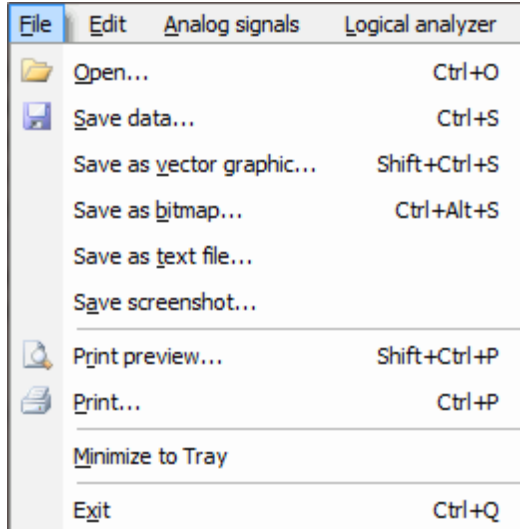
another mode, otherwise the software might 'freeze' and require a restart.)

In the center of the main window there is the main display of the selected operation mode where the traces and measurement results are displayed.

Below this is the settings panel that will correspond to the selected operation mode.

At the bottom of the window there is a status bar where the current device status is displayed. This provides information about the measurement status, as well as brief prompts about relevant operational details, which appear when pointing the mouse at specific parts of the window.

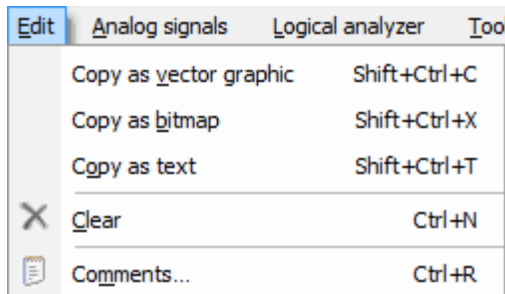
"File" menu consists of the following items:



- **Open** - Opens a data file for recoding the measurements of the oscilloscope, spectrum analyzer, recorder or logical analyzer (corresponding user-specified comments are integrated into such file). Upon opening the file you automatically go to the corresponding tab (operation mode) and all the parameters are set before data storage.
- **Save data** - Saves the measurement results, corresponding comments and current parameters of the selected operation mode (current selected tab) into a data file.
- **Save as vector graphic** - Saves the measurement results of the selected operation mode as vector graphic image in *.emf format (Enhanced Metafile).
- **Save as bitmap** - Saves the measurement results of the selected operation mode as bitmap image in *.bmp format (Windows Bitmap).
- **Save as text file** - Saves the measurement results of the selected operation mode as a set of values in text format.
- **Save screenshot** - Saves the application screenshot in .bmp format.
- **Print preview** - Opens the preview window of measurement results of the selected operation mode to be reviewed prior to printing.
- **Print** - Prints the measurement results of the selected operation mode.
- **Exit** - Exits from the USB Oscilloscope application.

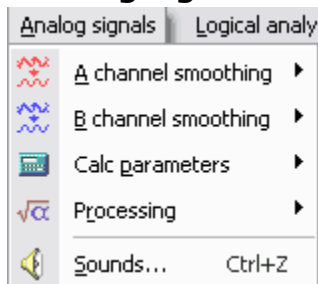
Note that you should stop measurements before closing the application (with either the menu or the X button), otherwise the application may open with limited options the next time it is launched. If this happens, shut the application and re-start it to restore the full functionality.

"Edit" menu consists of the following items:



- **Copy as vector graphic** - Allows copying the measurement results of the selected operation mode to Windows buffer as vector graphic in *.emf format (Enhanced Metafile).
- **Copy as bitmap** - Allows copying the measurement results of the selected operation mode to the Windows buffer as a bitmap in *.bmp format (Windows Bitmap).
- **Copy as text** - Allows copying the measurement results of the selected operation mode to the Windows buffer as text.
- **Clear** - Clears the measurement results of the selected operation mode.
- **Comments** - Opens the comment input window to be appended to the saved measurement results of the selected operation mode. Comments text is integrated into the measurement results file. Maximum comment size totals 64K for Windows 98/ME, however for Windows 2000/XP it is limited by the maximum file size.

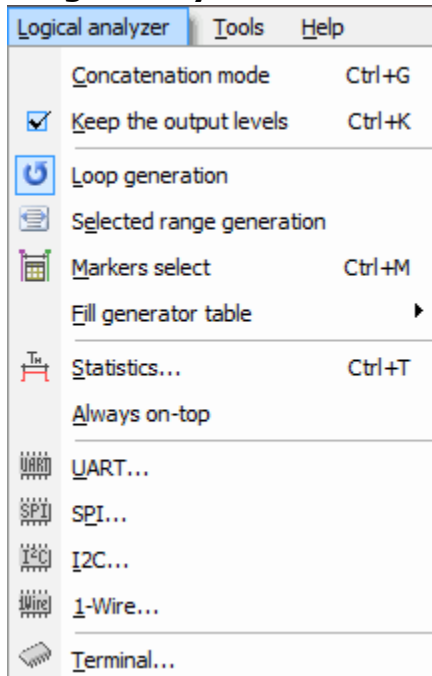
"Analog signals" menu consists of the following items:



- **A/B channel smoothing** - Submenu that allows determining the smoothing level of signal shape and noise reduction (only when operating in the oscilloscope mode).
- **Calc parameters** - Submenu that allows selecting the calculation of common signal parameters (DC and AC voltage and frequency component). Parameters may be calculated before processing (smoothing and filtering) then the real parameters of signal and processing will be calculated. Parameter calculation upon processing may be useful, if, e.g. it is necessary to determine the parameters of each dual-tone spectral component (DTMF) and for this purpose it is first necessary to filter one component and measure its parameters and then to filter the next component and measure its parameters as well.

- **Processing** - Submenu that allows selecting the different variants of measurement results mathematical processing (inverse, sum, difference)
- **Sounds** - Opens the event and [sound settings](#) window. Allows user-settable sounds to correspond to specific signal events. This window may be useful when searching, e.g. voltage 3.3 V to avoid constant jumping from the device on test to the monitor screen. You can set sounds to inform that voltage in the selected channel is near 3.3 V.

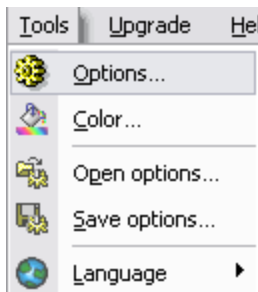
"Logic analyzer" menu consists of the following items:



- **Concatenation mode** - on/off concatenation mode of logical analyzer charts.
- **Keep the output levels** - on/off keep mode of generator outputs in last state. If this mode is off then the output level changes in accordance with the set timing chart only within measurement (generation), i.e., upon pressing the "Run" button and till releasing it (when the button changes color from red to green once again), upon measuring all outputs go to the third level. If the mode keep the output levels is on then upon measuring the output level will correspond to the last value of timing chart. Outputs automatically go to the third level if Keep mode is off or generator button is released. This mode is to eliminate the undesirable signal suppression (variation) if the corresponding generator channels have not been accidentally disconnected from any elements of device on test.
- **Markers select** - this item of menu allows selecting the generator data portion in the table, at that the portion limits are determined by the marker position, i.e. they are positioned between them.
- **Statistics** - opens the statistics window.

- **Always on-top** - enables or disables the statistics window position on top of the program. As the statistics window occupies sufficiently large part of the work window then as necessary it may be hidden but at the same time, e.g., in concatenation mode when calculating the number of sync pulses it is conveniently to position the statistics window on top to avoid constant opening.
- **UART, SPI, I2C, 1-Wire** - open the corresponding interface analysis and decoding windows.
- **Terminal** - opens the [terminal window](#) - hardware support (input/output) of the analyzing interface.

"Service" menu consists of the following items:

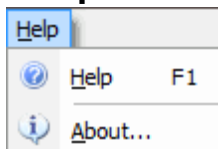


- **Options** - opens the operation mode settings window.
- **Color** - allows the user to specify color of the work area, channels and scale. These colors are also used if an image is saved.
- **Open options** - opens an application options file.
- **Save options** - saves the application options in the file.
- **Language** - submenu of this item allows selecting the available interface languages.

"Upgrade" menu

This item of menu opens the device software (firmware) [upgrade window](#) as well as indicates version and release date of the firmware.

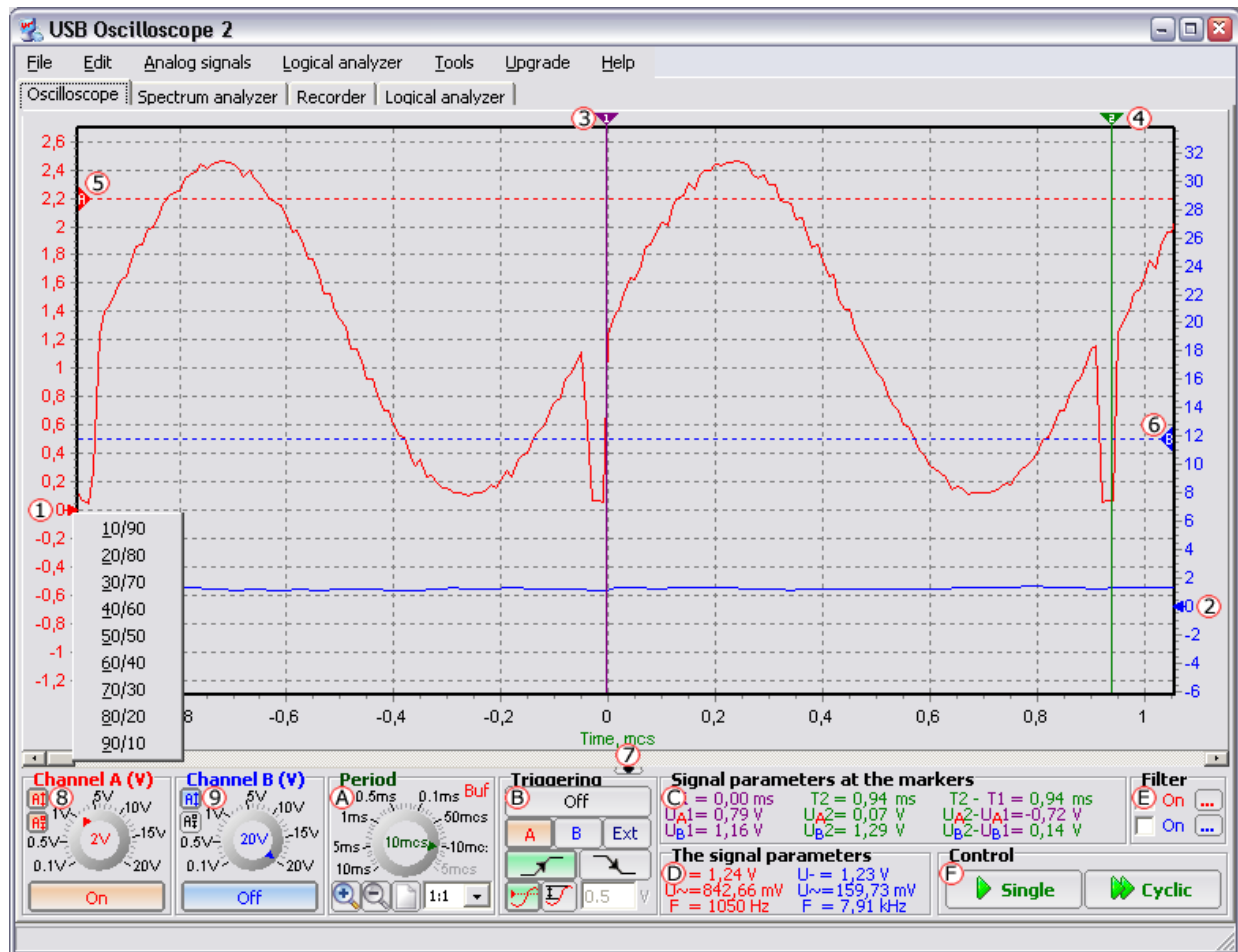
"Help" menu consists of the following items:



- **Help** - opens the help window (file).

About program - opens the program information window, where you can see the version info, etc.



Oscilloscope



This figure displays the main program window when operating in the oscilloscope mode. On the center of the window there is a work screen where oscilloscope displays are shown, red oscilloscope display corresponds to channel A and blue one corresponds to channel B. To the left of the work screen frame there is a voltage scale of channel A (red font), to the right of the frame there is a voltage scale of channel B (blue font). Values of both voltage scales are always expressed in volts. Beneath of the work screen there is a time base (trace).

To the left and right of the work screen there are two **cursors** ① and ② that allow shifting zero of channel A and channel B accordingly. It is expedient to shift zero if the waveforms of both channels overlap on one another too much that complicates their analysis. To install one of 9 zero standard positions you need to right click on the corresponding cursor and then to select one of the possible values of zero positions from the pop-up menu.

In the latest version of the software it's possible to scale the signal chart. There are two buttons on the tool bar to control it:

-  - Enable vertical zoom
-  - Reset zoom

When the vertical zoom is enabled, left click and select the area on the screen (like in common graphic editors). When it's done the selected area will spread vertically on the full screen.


There are two **markers** ③ and ④ above the work screen frame that are to perform the correct measurement of time intervals and voltage amplitude values of each channel. Markers can be moved with the mouse, you need to left click on the triangle and then when holding the left button to move the mouse to the left or right. When moving the marker within the work screen the signal parameters below the marker will be displayed on the panel A.


To set the **triggering level**, two horizontal markers ⑤ and ⑥ are used. Marker ⑤ sets the trigger voltage level (amplitude) of channel A, which is used when triggering (absolute) with the channel A. Marker B similarly sets the level used when triggering with channel B. Triggering markers are moved by clicking and dragging on them ③ and ④. Moreover, upon moving the trigger markers, the level set is correctly displayed on status bar. Note that if the trigger level is outside the range of voltages in the signal, USB oscilloscope may respond slowly - because it lacks a trigger (normally it will take a few seconds or less to display measurements). If this happens, press 'stop' and reset the trigger value for the triggering channel.

The Oscilloscope chart can be moved inside of the work screen by the standard scroll bar ⑦. When changing the time base or zoom, the previous oscilloscope displays may not completely fit inside the work screen, and the slider will get smaller than the scroll bar proportionally with the oscilloscope display length within the work screen to the total oscilloscope display length ratio. There is a small button below the scroll bar that allows hiding the oscilloscope settings panel that allows more area for the work screen.


The **voltage scale** range of channel A/B can be set on the panel ⑧ / ⑨ "Channel A/B (Volt/Div)". Scale range is set by knob (see for details). It is necessary to note that USB oscilloscope supports two input ranges 0-2 V and 2-20 V by hardware. It follows that to show the most reliable oscilloscope display, which amplitude is within ± 2 V, it makes sense to select scale range ± 0.2 Volt/Div or lower. As the turning error within scale range ± 0.2 Volt/Div and lower will total $4V/1024 = 0.0039$ V (capacity 10 bits) while within scale range ± 0.5 Volt/Div and higher the

turning error will be 10 times larger $40V/1024 = 0.039 V$. On/off buttons for the channels are positioned on the same panels. If when analyzing the device it is not required to analyze two analog signals concurrently then it is expedient to turn off one of the channels that will allow increasing the maximum sampling frequency 100 KHz to 200 KHz.

Panel "**Period**"  allows setting the sampling period (time base) within which the analog input digitization starts. For normal previewing the oscilloscope display it is recommended that the sampling frequency is at least 5-6 times higher than the maximum input frequency. In addition to select time base on the panel "Period" you can set the oscilloscope display zoom, i.e. stretch the oscilloscope display up to 4 times for the time base for more detailed analysis, e.g. of any signal time parameters without any change in sampling period. In the upper right corner of the panel "Period" there is information about the reading mode (if enable): buf - reading with microcontroller internal buffer, pipe - pipe reading in computer (sample buffer size increases several times but requirements to the computer are getting tighter ([see for details](#) about the reading modes)). Minimum/maximum period that allows determining the reading mode is set in the Settings window. Double left click on the markup with reading mode information opens the Settings window similar to the [Service/Setup](#).

All the triggering control elements except for the markers are positioned on the panel "**Triggering**" . On/Off button allows turning on and off triggering. A and B buttons select the channel and corresponding marker, that is, which signal will be used as triggering source. Ext button selects an **external source** for triggering that is connected to the **channel B.5 of the logical analyzer** (external triggering is available only when buffer reading). Select edge buttons determine which signal edge (rising or falling) will be used for the absolute triggering, which signal step (rising or falling) will show the differential triggering and which external sync pulse edge (rising or falling) will show the external triggering. Two buttons positioned beneath the panel determine the triggering type: **absolute** or **differential**. An input field near the differential triggering settings button determines the difference between the adjacent waveform samples and if exceeded then the triggering condition is met.

Note that when setting the triggering parameters, if they cannot be met, e.g. if given trigger level is 5 V and maximum signal amplitude does not exceed 2 V, the device will be continually wait for the triggering condition, and will appear to 'freeze'. One of the start measurement buttons (single/cyclic) will be red. In this case it is necessary to stop measuring by pressing the respective Stop button. Adjust to trigger level to fall within the range of the signal, and re-take the measurement.

On the panel "Signal parameters below markers"  the position of each marker

within the time base and signal amplitude of both channels below each marker are displayed. Markers time difference and signal amplitude is also calculated, at that the difference result color is determined by the color of marker, which corresponding value is larger.

On the panel "Common signal parameters" **D** the calculated values of DC and AC voltage component are displayed and if possible, frequency value for each channel as well.

Panel "Filtering" **F** ensures connection and calculation of **digital filters** for each channel. To turn on filtering for the channel it is first necessary to calculate the filter when opening the filter settings window while pressing the "... " button and then tick off On for the selected channel.

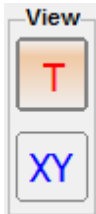
To display traces of the signal, press the **Single** button or the **Cyclic** button on the panel "Control" **F**.

The **Single** button takes only one measurement (sample digitization and accumulation by microcontroller and then their transmission to shell when operating via buffer), and then the newly read oscilloscope displays are shown on the work screen.

The **Cyclic** button carries out similar actions **except** that upon completing the measurement a new measurement starts automatically.

Upon pressing either of the start buttons (single/cyclic) the respective changes its name to **Stop** and becomes red colored. Pressing this 'stop' button causes an immediate device reset and a wait for result measurement termination. The stop button may be the only means to stop measuring, e.g. when the triggering level is set inappropriately.

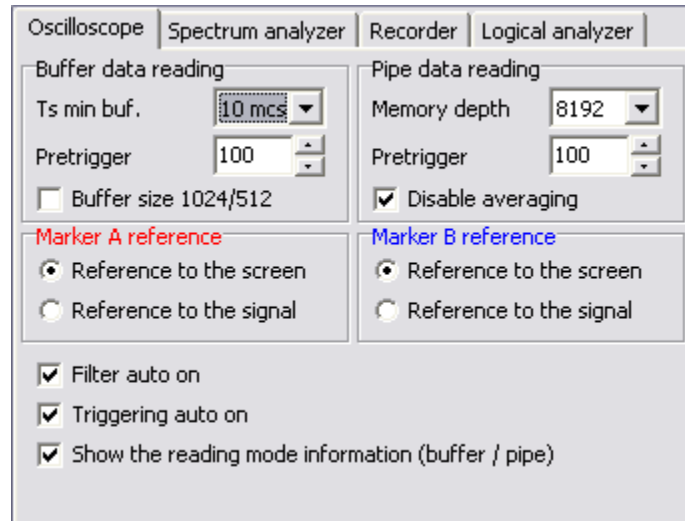
There is View section at the bottom of the screen with T and X-Y buttons. The buttons are used to switch between the oscilloscope general timing and X-Y modes.



In the X-Y mode channel A is used to set the horizontal scale and channel B for the vertical one. Zero points move to the middle of the screen edge and extra-markers show up. Since timing is not necessary for X-Y mode the "Signal parameters at the markers" section shows only voltage of the signal and doesn't the T1, T2 and T2-T1 values

The Period knob in X-Y mode defines the size of data that will be processed per frame. The more the period value the more the time period will be processed to show the signal.

The figure below displays the oscilloscope settings tab.



There are the following elements on the tab:

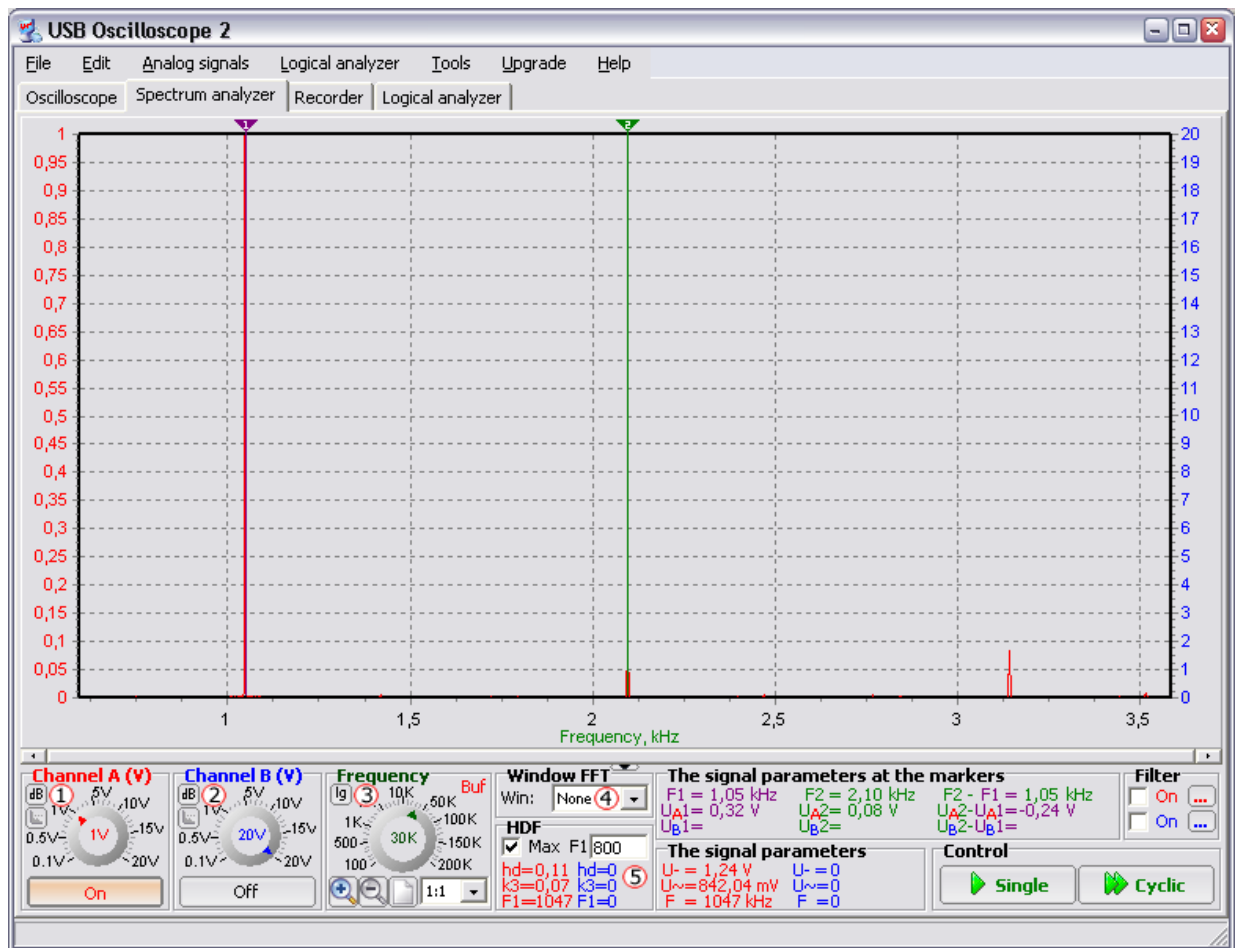
- **Ts min buffer** - list that allows selecting the minimum sampling period and if less it will be buffer reading. In this case the minimum period equal to 10 μ S ($F_s = 100$ KHz) is set, i.e. for the sampling periods of 9 μ S...5 μ S ($F_s = 111$ KHz...200 KHz) it will be buffer reading and for the sampling periods of 10 ms...10 μ S ($F_s = 100$ Hz...100 KHz) it will be pipe reading. Ts min buffer is determined by experiment on the basis of several measurements. For example, it is first necessary to set Ts min buffer = 5 μ S, i.e. within any sampling period it will be pipe reading and perform the measurements for Ts=5 μ S ($F_s = 200$ KHz) as well and if there is not any signal distortion it means that your computer and operating system can transfer data at maximum speed. In case of any distortion, it is necessary to increase the sampling period, e.g. up to 6 μ S (167 KHz) and perform the measurements again and etc. Upon determining the sampling period without any distortion in measurement results it is expedient to set it as min. [What are the reasons?](#)
- **Pretrigger** - allows presetting a number of samples to be stored until triggering. On the time base this waveform portion is within negative part, i.e. signal before triggering event - up to zero and signal after triggering - after zero.
- **Buffer size 1024/512 bytes** - forced reduction of buffer size from 1126(1 channel)/563 (2 channels) samples up to 1024/512. Why do we tick off this flag? FFT is used for [filtering](#) and as is known, the number of samples for FFT has to be 2^n , i.e. if the filter is on for any channel this will lead to automatic buffer depth reduction up to 1024/512 samples that will cause some perplexity, e.g. why the signal for the channel A (no filter) is longer

than the signal for the channel B (available filter). If there is nothing daunted for you, do not tick off this flag that will add 102/51 samples if the filter is not used.

- **Memory depth** - while pipe reading it is determined by the available memory size in computer and in the oscilloscope mode it is artificially limited by 65535 samples for the channel as if the memory depth is large it is inconvenient to preview the periodic waveform (it is recommended to use the [recorder](#) for analysis of nonperiodic or very long signals).
- **Disable averaging** - this flag disables to reduce the period (increase frequency) of sampling and perform averaging for several samples (arithmetic average).
- **Reference to the screen/reference to the signal** - this flag sets reference of triggering markers, if the reference to the signal is selected the triggering marker will synchronously move within the screen when changing time base or shifting zero while keeping the given position in Volts.
- **Filter auto on** - enables to turn on [+] On the corresponding [filter](#) upon setting its parameters.
- **Triggering auto on** - enables to turn on triggering automatically when pressing any button from the panel "Triggering".

Show the reading mode information (buffer/pipe) - enables to show the reading mode information to the panel "Period".

Spectrum Analyzer



This figure displays the main program window when operating in the spectrum analyzer mode. Most part of window elements when operating in this mode is similar to the corresponding elements when operating in the [oscilloscope mode](#). Exception is that the horizontal axis is set as the frequency axis; time base is set as frequency/screen and the voltage scale is negative amplitude-free.

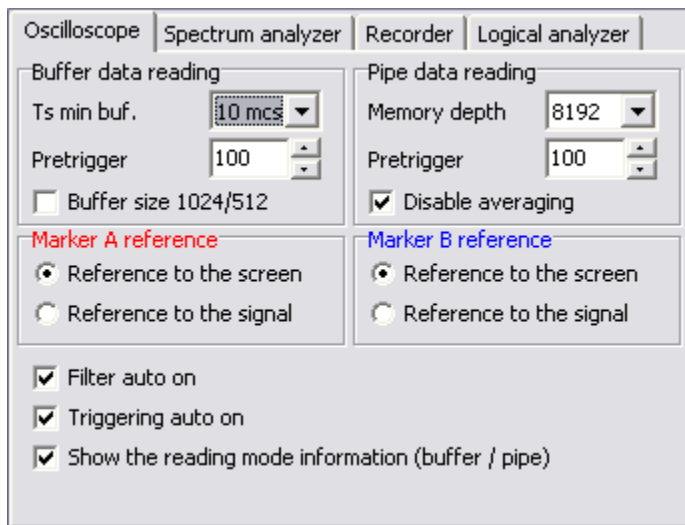
New elements relating exceptionally to the spectrum analysis were added. So, on the panels "Channel A/B (Volt/Screen)" the additional "V / dB" and "0 dB = 0.775 V/0 dB = U_{max}" and buttons were appeared. The first button determines the units to express the vertical scale: in Volts or dB. The second button (it is active only if the first one is pressed) determines what voltage level corresponds to 0 dB: 0.775 V or maximum voltage value along the whole frequency axis for the set channel.

On the panel "Frequency" the button **3** is added that determines whether frequency axis is linear or logarithmic.

Triggering panel is changed to the FFT window function settings panel ④. On the panel there is a dropdown list containing some common window functions: Hamming, Hanning, Blackman, Blackman-Harris.

Panel ⑤ for calculation of nonlinear distortion factor or harmonic distortion factor (HDF) was also added. On the panel you can view the results of general calculation of HDF, HDF of harmonic thirds and frequency of the first (main) harmonic, which can be forced in the field F1 or calculated automatically when ticking the flag [+]
Max.

The figure below displays the spectrum analyzer settings tab.

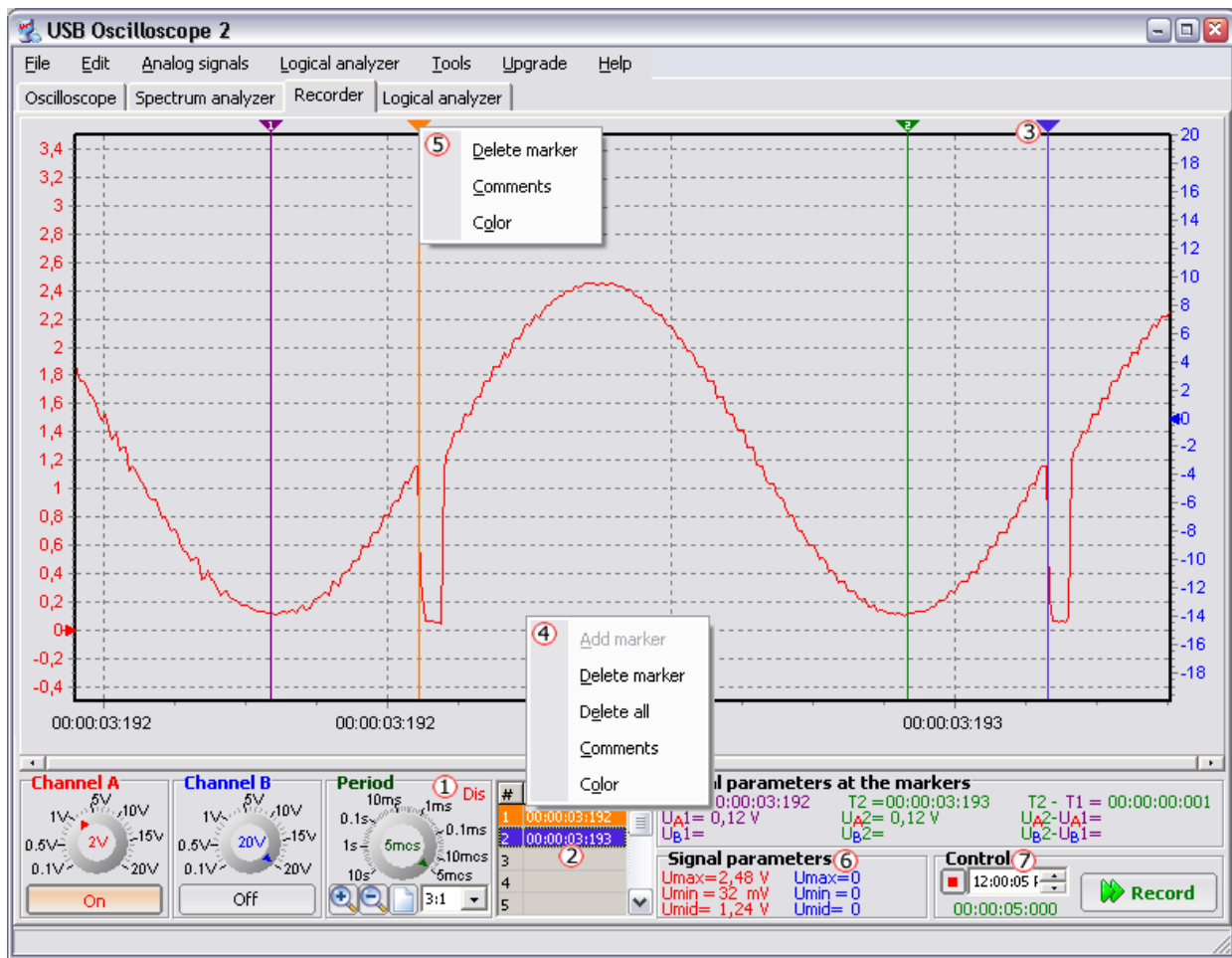


There are the following elements on the tab:

- **Fs max buffer** - a drop-down list that allows the user to select the maximum sampling frequency (if greater than this setting it will be buffer reading). For example, if the maximum frequency is set to 100 KHz, i.e. for sampling frequency of 111 KHz...200 KHz it will be buffer reading and for sampling frequency $F_s = 100 \text{ Hz} \dots 100 \text{ KHz}$ it will be pipe reading.
- **Fs max buffer** can be determined by experimenting with your system and making several measurements. First, set $F_s \text{ max buffer} = 200 \text{ KHz}$, i.e. at any sampling frequency it will be pipe reading and perform the measurements for $F_s = 200 \text{ KHz}$ as well. Then check the measurements for distortion. If there is no signal distortion, this means that your computer and operating system can transfer data at the maximum speed. If there is distortion then it is necessary to reduce sampling frequency, e.g. up to 190 KHz and perform the measurements once again, repeating until you get distortion free measurement. When you have determined the sampling frequency without distortion in the measurement results, use that value for $F_s \text{ max buffer}$. [What are the reasons?](#)
- **Memory depth** - when pipe reading this is determined by the available memory in the computer and in the spectrum analyzer mode it is artificially limited by 65535 samples for channel.
- **Disable averaging** - checking this check-box disables the increase the sampling frequency and disables averaging for several samples (arithmetic average).
- **Total number of the harmonics** - determines the number of the harmonics used when calculating the harmonic distortion factor.
- **Filter auto on** - enables activating the corresponding [filter](#) upon setting its parameters.

Show the reading mode information (buffer/pipe) - enables the display of information about the reading mode on the panel "Frequency".

Recorder



The figure displays the main program window when operating in the recorder mode. Most part of the window elements when operating in this mode is similar to the corresponding elements when operating in the [oscilloscope mode](#) with several exceptions because when operating in the recorder mode the waveform is constantly real time read from the microcontroller. In this consequence in the recorder mode there is no triggering and filtering panel and the signal common parameters panel as well as time base format (hours: minutes: seconds: milliseconds) is changed.

It is necessary to note that in the recorder mode you can perform neither single nor cyclic measurement that's why to start the waveform record it is necessary to press the Start button and then the button will change its name to Stop of red color. To stop recording it is necessary to press the Stop button. When recording the option to change voltage scale range, to turn on/off channels and change time base is blocked. It is connected with the real time waveform record and any change will lead to the significant CPU intensity resulted in skip of the new data portion.

In the upper right corner of the panel "Period" ① instead of information about the reading mode there is information about necessity in waveform plotting immediately in the recording mode (if enable): Dis - disable plotting, En - enable plotting. The point of disable or enable plotting within the measurements is that within the sampling period, e.g. 5 μ S (200 KHz) each second it is necessary to plot 200 thousand dots on the screen but it is out capabilities of even advanced computers and at this plotting rate it will be an obscure image on the screen (waveform will be just leaping). But on the other hand, within the large sampling periods (less frequency) displaying the measurement results immediately in the recording mode enables you to analyze the signal variation dynamics visually and stop recording under certain conditions. Moreover, in the recorder mode you can both expand (increase) and compress (decrease) waveform with the corresponding buttons and change time base, it is useful if you need to view the general waveform pattern, e.g. at $\Delta t = 5 \mu$ S, 1 minute of signal ($60/5 \times 10^{-6} = 12$ million dots) is recorded.

To conduct analysis of the sustained signals more conveniently the option of waveform pattern marking is added. To the right of the panel "Period" there is a table ② to add/delete/move to markers ③ and change parameters. So, to add (create a new marker) it is necessary: two left clicks on the free cell of table or select the free cell of table and press the Insert key or right click on the free cell and select "Add marker" item in the dropdown menu ④. After that if enable, two dialog windows appeared to create comments and color of the marker and then in the center of the work screen a newly added (created) marker appeared. This added marker can be moved along the screen similar to other markers. To delete the markers it is necessary: to select the nonempty cell of the table and press the Delete key or right click on the nonempty cell or on the marker to be deleted and in the dropdown menu ④ / ⑤ select "Delete marker" item. As the markers are referenced to the time base (waveform pattern) then when moving or zooming the waveform it may be so that the necessary marker will be out of the viewable area of the work screen. To move to the set marker quickly you need to select the corresponding cell of the table (color similar to the marker) e.g. by left click. Marker parameters (color and comments) can be changed from the corresponding pop-up menu. It is necessary to note that position of the markers and their parameters are saved in the file with the measurement results, i.e. when opening the file all the markers will point the corresponding waveform patterns.

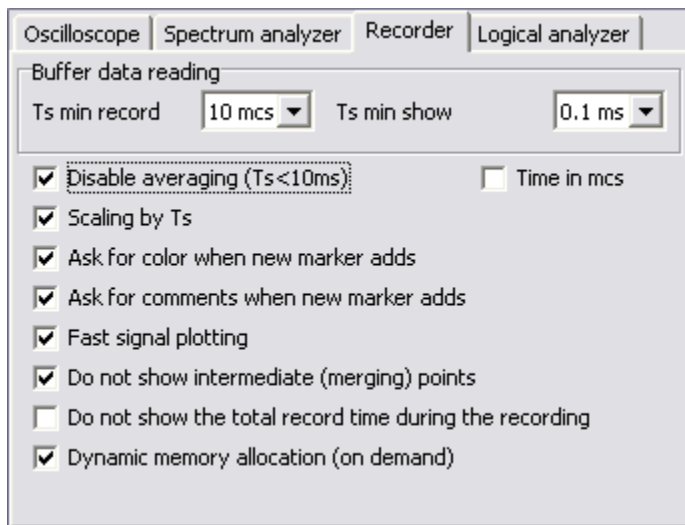
On the panel "General signal parameters" ⑥ the maximum, minimum and average voltage value for each channel is displayed.

On the control panel the elements ⑦ are added that allow setting time after which the waveform record will be automatically stopped. Auto stop is conveniently used

if it is necessary to perform a long but fixed waveform record. Moreover, when recording within the sampling periods of 0.1 ms and lower (frequency higher 10 KHz) it is strictly recommended to preset the necessary length of waveform record as it enables the program to allocate the necessary memory in advance before recording process but avoid its dynamic allotment as required immediately within the waveform record process. There is a bar with information about the total time of waveform record below the auto stop time settings element.

It is necessary to note that the recorder mode is extremely particular about the computer memory size as, e.g. when recording for the sampling period of 5 uS (200 KHz) within 1 minute the measurement results will total: $200\ 000$ (samples/sec) * 60 (sec) * 2 (vertical and horizontal coordinate) * 8 (double size) = 187 MB, and the file with measurement results will total approximately as much.

The figure below displays the recorder parameter settings tab.



There are the following items in the menu:

- **Ts min record** - list that allows selecting the minimum sampling period, if less recording is disabled. In this case the minimum period equal to 10 μ S ($F_s = 100$ KHz) is set, i.e. sampling periods of 9 μ S \diamond 5 μ S ($F_s = 111$ KHz...200 KHz) cannot be selected. Ts min record is determined by experiment on the basis of several measurements. For example, it is first necessary to set Ts min record = 5 μ S, i.e. enable recording within any sampling period and perform the measurements for Ts=5 μ S ($F_s = 200$ KHz) as well, if there is not signal distortion it means that your computer and operating system can transfer data at the maximum speed. If there is distortion you need to increase the sampling period, e.g. up to 6 μ S (167 KHz) and perform the measurements once again and etc. Upon determining the sampling period without distortion in the measurement results, it is expedient to set it as minimum. So, this option just disables to set the sampling periods if you consider that within those there is distortion while reading. [What are the reasons?](#)
- **Ts min show** - list that allows selecting the minimum sampling period if less the waveform plotting immediately in the record mode is disabled. The point of disable or enable plotting within measurements is that within the sampling period e.g. of 5 μ S (200 KHz) each second 200 thousand dots are to be plotted on the screen, but it is out capabilities of even advanced computers and at this plotting rate it will be an obscure picture on the screen (waveform will be just leaping). But on the other hand, within the large sampling periods (less frequency) displaying the measurement results immediately in the recording mode enables to analyze the signal variation dynamics visually and probably stop recording under certain conditions. In addition, it is necessary to note that Ts min show is more or less dependent on a number of the

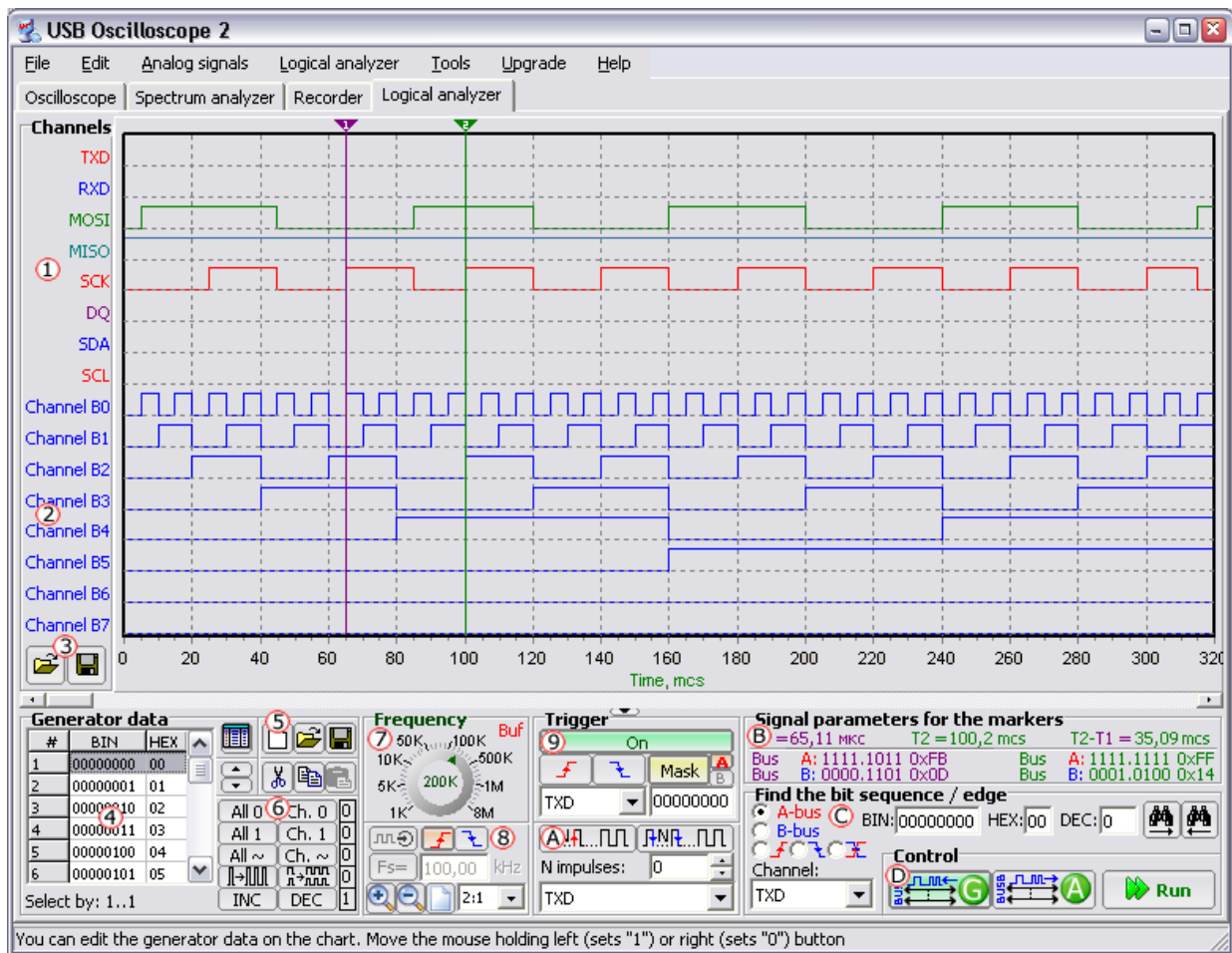
recorded samples, e.g. in Athlon XP 1700, 512M you can record waveform with Ts min show for 20 ms (50 KHz) but in 3-5 minutes of recording (about 10 million samples) time for the next waveform portion plotting sufficiently increases due to necessity to calculate the additional diagram parameters.

- **Disable averaging (Ts<10 ms)** - this flag disables to decrease the sampling period (increase frequency) and perform averaging for several samples (arithmetic average). This flag is active only for the periods less than 10 ms (frequency higher than 100 Hz), with longer sampling periods (less frequency) averaging will be turned on automatically. When operating within the sampling periods close to Ts min show and Ts min record, it is recommended to disable averaging. In case if the sampling period is much longer than Ts min show, it is recommended to enable averaging based on considerations that the microcontroller data are read for 409 samples (512/1.25 (10 bits)) at a time and then plotting starts, e.g. for the sampling period of 10 ms (100) plotting will be performed every $409 * 0.01 = 4$ seconds and if averaging is enabled and Ts min show is set, e.g. to 1 ms (1 KHz, i.e. when artificially limiting the maximum increase of sampling frequency up to 1 KHz) then plotting will be performed every $409 * 0.001 = 0.4$ seconds for $409/10=50$ samples, at that it will be much convenient to view the waveform variation.
- **Scaling by Ts** - enables the signal compression/stretch when changing the sampling period. The point is only when the mass signal compression is necessary if you need to view the general waveform pattern.
- **Ask for color when new marker adds** - if the flag is set, then upon adding (creating) a new marker the color settings dialog window automatically opens.
- **Ask for comments when new marker adds** - if the flag is set, then upon adding (creating) a new marker the comments input dialog window automatically opens.
- **Fast waveform plotting** - if the flag is set, then the measurement results plotting is performed with faster simplified functions. In the operating system Windows 98/Me the result show accuracy can be reduced in some way.
- **Do not show intermediate (merging) points** - if the flag is set, then when plotting the points with the same coordinate in pixels on the screen are not displayed. Waveform plotting will be much faster if a large waveform portion is shown concurrently, e.g. within mass compression but if there is the abrupt signal variation then it will be shown incorrectly within mass compression.
- **Do not show the total record time during the recording** - if the flag is set and the sampling period is fixed at which display of the measurement results is disabled immediately within the waveform record then the total record time will not be shown during the recording. At that, if the autostop time is not fixed, then the total record time has to be calculated by any external timer. It is recommended to set this flag at the maximum frequency

and in low-speed computers, as the very bar plotting with the total time requires greater resources of CPU than the next data processing.

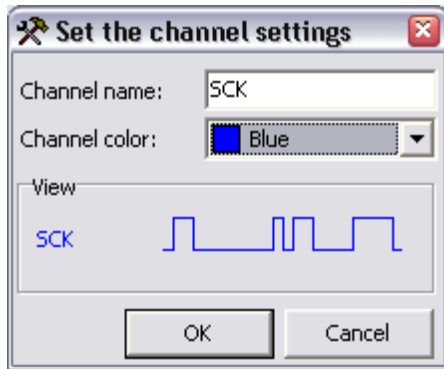
Dynamic memory allocation (on demand) - if the flag is set, and autostop is not set, then the memory for the waveform samples storage is allocated immediately in the waveform record mode for which greater resources are consumed and the maximum sampling frequency for the low-speed computers decreases. If the flag is not set, then before start the program tries to allocate all available memory but at most 256M for which the waveform record starts with delay. If you know or can estimate the exact interval to record waveform it is strictly recommended to set the autostop time as it will enable program to allocate the necessary memory early (there is no excess memory allotment) before the record process starts.

Logic Analyzer



The figure displays the main program window when operating in the logical analyzer mode. Such elements as markers, scroll bar and sampling frequency settings panel (time base) are completely similar to the corresponding elements when operating in the [oscilloscope mode](#). That's why we enlarge upon new elements incidental only to this mode.

To the left of the work screen there is the panel containing the names of all 16 channels, which are functionally divided into two buses. The first 8 channels are related to the bus A ① and next eight channels are related to the bus B ②. Channels of the bus A (further bus A) are always used as 8-channel analyzer. Channels of the bus B (further bus B) can be used both in the mode of 8-channel logical generator and as additional 8-channel analyzer or generally disabled. Operation mode of the bus B is set by the corresponding buttons on the control panel ④. Moreover, color and name of each channel can be changed and for this you need to point the name of the selected channel with the mouse cursor and left click and then the channel settings window opens.



To change the channel name you need to input the new name in the field "Channel name". To change the channel color you need to left click on the dropdown list "Channel color" and then select the necessary color, if there is not desirable color in the list then click on color "Custom..." and the standard color settings window will open. To apply these changes you need to press OK button and to undo all changes inserted you need to press Undo button.

There are two buttons ③ below the panel "Channels" that allow opening and saving all the names and colors of the channels in the file.

In the bottom left corner of the main window there is a panel "Generator data", which is active only if the bus B is used in the mode of 8-channel logical generator. As the panel name suggests, it is designed for setting (creating) the generator timing charts. In the left part of the panel there is a table ④ containing the generator data in binary and hexadecimal formats. Data in the table can be changed in two ways: when immediately keying or selecting the necessary part in the table and pressing the relative buttons of timing chart generation ⑥. To go immediate editing you need to press the button positioned next to the upper right corner of the table or left double-click on the table. To go selecting the data you need to press the button positioned next to the upper right corner of the table once again or press the shortcut keys Ctrl D. To select the data portion in the table it is necessary first to select the range start by left click on the selected cell and then while holding the left mouse button to move the mouse up and down. If you need to select the large data portion then it is expedient upon selecting the range start to move to the range end when moving the slider and then left click on the last cell of range while holding the Shift key. To select all the cells you need to press the shortcut keys Ctrl A. Data portion in the table can be selected with the markers, which positions on the work screen determine the limits of the portion to be selected. It can be convenient if, e.g. it is necessary to generate pulse in one of the generator channels, which length is equal to pulse in any analyzer channel. For this purpose it is necessary first to determine the analyzer pulse limits with the markers and then select the data portion in the table, when pressing, e.g. Ctrl M and then press the "Ch.=1" button. There is a line with information about the limits of the current selected data portion below the generator data table.

There is also a row of buttons **5** on the generator data panel that allows clearing the generator data, reading the generator data from the file or saving the generator data in the file, shifting the selected data portion for backspace character up and down, cutting or coping the selected portion in the buffer (other than Windows buffer) as well as pasting the data from the buffer in the table from the selected cell.

Generator timing charts can be set both as table and plot immediately on the work screen with the mouse. To plot the timing chart of one of the generator channels it is necessary to set the mouse cursor between the dash horizontal lines determining the limits of the selected channel and press the left or right mouse button and then when holding this button to move the mouse to a certain direction (left or right). If the left mouse button is pressed then the plotted timing chart portion will accept the logical unit status and if the right mouse button is pressed then - logical zero.

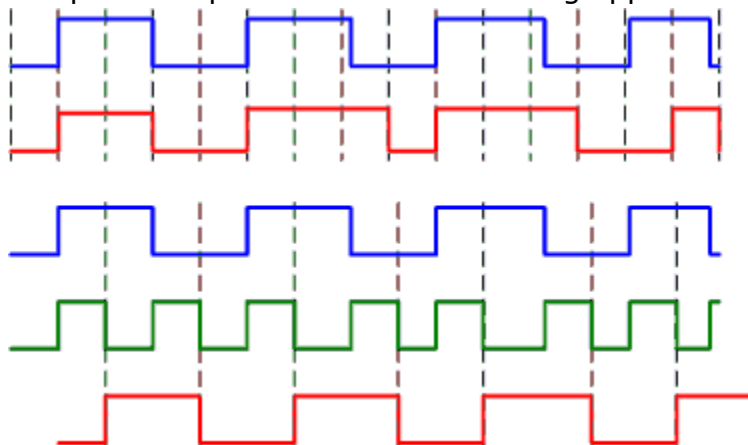
Panel "Frequency" **7** allows setting the sampling frequency (time base). This panel is functionally similar to other time base settings panels. Exclusion is only that there are several actual submodes of the logical analyzer, which are automatically selected depending on the set time base:

- **4...8 MHz** - buffer size is just 128 bytes, only one bus A is functioning, triggering by mask is only for the bus A (ignoring don't care condition), triggering by edge is only for the channels of the bus B (i.e. you can analyze 8-bit bus and trigger by signals WR, RD and etc.), no pretrigger, no pulse miss. This submode is for analysis of the moderate-speed 8-bit buses that enable triggering for the additional channels.
- **2...2.67 MHz** - buffer size is equal to 1160 bytes, only bus A is functioning, triggering by mask is only for the bus A (considering don't care condition), triggering by edge is only for the channels of the bus B (i.e. if required triggering by edge for one of the channels of the bus A the cord of the bus B is to be connected as well, that is inconvenient), fixed pretrigger buffer length equal to 8 bytes, no pulse miss. This submode is for analysis of the low-speed 8-bit buses that enable triggering for the additional signals and small pretrigger.
- **from 1 MHz and lower (when buffer reading)** - buffer size is equal to 1544 bytes, all the modes are available (analyzer, analyzer generator, analyzer analyzer). All the types of triggering are active and there is pulse miss, adjustable pretrigger buffer depth 8 to 120 bytes. External clocking (triggering) option is also added. This submode is for general purpose, you can analyze both low-frequency data buses and bit-sequence interfaces (just analyze but acquire data) and logical generator is active in this submode as well.
- **from 500 KHz and lower (when pipe reading)** - buffer size is equal, limited by the computer memory. It is not recommended to set the

buffer size more than several Mbytes as it is extremely inconvenient to analyze such long signals. One channel show will take memory that 16 ($2 * \text{SizeOf}(\text{double})$) times more than the buffer size, i.e. to show the whole bus it requires memory that $16 * 8 = 128$ times more, providing that the waveform value changes for every sample capture. If there are pauses or pulses longer than one sample capture then they are coded with just 16 bytes ($2 * \text{SizeOf}(\text{double})$). When pipe reading the bus A is only active, all the types of triggering are available, adjustable pretrigger buffer depth is 1 to 99% of the given buffer depth. Option of external clocking (triggering) is also added. This submode is mainly for analysis of interface sequence. When using the computer memory as a waveform buffer you can record very long waveform portions for its subsequent analysis.

There are elements ⑧ below the sampling frequency settings knob that ensure turning on the external clocking (triggering) mode. With these elements you can select the edge of external clock signal to perform sample capture. External clock signal frequency can be set both by knob or when inputting the exact frequency value in the field Fs. External clock signal frequency value is setting only for the correct show of charts in the time base.

It makes sense to explain the point of external clocking application by example:



The figure displays the following binary waveforms:

- blue color - input
- red color - received (shown) signal
- green color - external clock signal

In the first case the internal sampling frequency is used, e.g. 1 MHz where the point of waveform sample capture is shown with the dash lines in 1 uS. Input frequency is APPROXIMATELY equal to 250 KHz, i.e. 4 times less than the sampling frequency that is good enough. As is evident from two upper charts the first pulse and pause when receiving signal completely correspond to input. The next pulse will be almost by 50% longer because in the point of sample capture the input was at

the peak logical level for just several fractions of μs longer than for the first pulse. As is evident from the figure as the input frequency is APPROXIMATELY equal to 250 KHz, i.e. there is a minor pulse and pause time jitter and the resulting signal is sufficiently distorted (it is necessary to consider if input parameters are to be correctly measured).

In the second case the external clocking is used. As is evident from the figure the input level changes as to the rising-edge of clock signal and the falling edge of clock signal is approximately at the midlevel of input pulse or pause and it follows that it is expedient to set clocking as to the falling edge. The point of waveform sample capture is shown with the dot lines, which concurs with the falling edge of clock signal. As is evident from the figure this signal completely corresponds to input except for bias for a half of period of clock signal, which has no influence.

Based upon this information it may be concluded that given external clock signal synchronous with the input (generally it is a common practice for protocol sequence) "value" of each sample capture increases. So, when internal clocking there were 16 sample captures and minor signal distortion and when external clocking there were only 8 sample captures and this signal completely corresponds to input. "Value" of each sample capture increases because the sample capture is performed within the "correct" point in time to be determined by external clock signal. Moreover, if we suppose that instead of 4 input pulses we will receive only 2 input pulses and 4 clock ones (e.g. in default of exchange) then when internal clocking we will anyway receive 16 sample captures, which half is out of exchange and at the same time when external clocking we will receive only 2 "correct" sample captures. For the external clocking functions the microcontroller hardware is used about which the external clock signal can be fed only to the channels of the bus B, i.e. when applying the bus B as 8-channel generator the external clocking is inaccessible.

All the triggering controls (called trigger for digital signals) except for the markers are positioned on the panel "Trigger" ⑨. On/Off button allows turning on and off triggering below which there are the triggering operation option buttons: with rising or falling edge of the selected channel signal or triggering with mask. There is the dropdown list below the triggering signal edge option buttons that is to select the triggering source channel only by edge. To the right of the mask triggering settings buttons there are two small select bus buttons, to which the given mask will be applied at triggering. Mask is set in the field positioned below the mask triggering settings button in binary format at that the symbol 'x' or '-' means the don't care condition of the corresponding channel. In the bottom half of the triggering panel there are the elements determining necessity to skip a set number of pulses ⑩. Miss pulses will be active only if triggering is on. To start the miss pulses mode it is necessary to press one of the buttons determining the pulse start if no button is pressed the miss pulses mode is failed. In the field "N pulses" upon

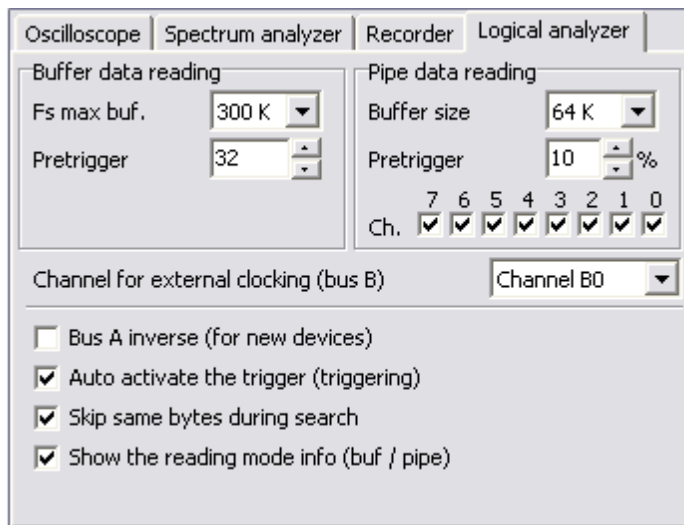
meeting triggering condition the required number of missing pulses is set and in the dropdown list positioned beneath the panel the channel is selected for which calculating and missing the given pulses number will be performed. It is expedient to use the miss pulses mode if concatenation mode is off.

On the panel "Signal parameters below markers" **B** similar to operating in the oscilloscope mode the position of each marker on the time base and status of each bus below the marker are shown. Markers time lag is also calculated.

Panel **C** is to find the bit sequence/edge on one of two buses. Before you start it is necessary to select the bus/channel to find and set the bit sequence/edge in binary, hexadecimal or decimal format. To start finding it is necessary to press one of the search buttons. If the set bit sequence/edge has been found then the marker 1 will be automatically shifted to it otherwise the message window with information about no set bit sequence/edge to found will open.

On the control panel **D** there are the buttons determining the operation mode of the bus B. Button with G icon moves the bus B into the operation mode of logical generator and the button with A icon - into the mode of additional 8-channel analyzer. If no buttons are pressed then the bus B is deemed off. On the control panel there is also the start single measurement button. It is necessary to note that if the bus B is used in the logical generator mode and triggering is on then the data will be fed to the generator bus upon completing the triggering event but not immediately upon pressing the Run button.

The figure below displays the logical analyzer settings tab.



There are the following elements on the tab:

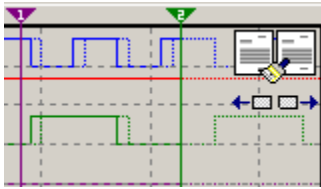
- Fs max buf** - list that allows selecting the sampling frequency if more it will be buffer reading. In this case the maximum frequency equal to 300 KHz is set, i.e. for the sampling frequency from 400 KHz and higher it will be buffer reading and for the sampling frequency of 1...300 KHz it will be pipe reading. Fs max buf is determined by experiment on the basis of several measurements. So, e.g. you can first set Fs max buf= 300 KHz and perform the measurements for Fs=300 KHz and if there is not signal distortion it means that your computer and operating system can real time transfer the data at the preset sampling frequency Fs=300 KHz. If there is distortion then it is necessary to reduce the sampling frequency, e.g. up to 250 KHz and perform the measurements once again and etc. Upon determining the sampling frequency where there is no distortion in the measurement results, it is expedient to set it as maximum. Theoretically this microcontroller can transfer data at speed of 1 \diamond Bytes/sec but practically it accelerates only to 256 KBytes/sec, as there are not adequate Silabs drivers when operating in the isochoric mode. If the set buffer is small then the sampling frequency can increase up to 400 KHz as, e.g. when reading only 4K data, it will not be enough time for FIFO MK (1024 bytes) to overfull and there will not be distortion accordingly. [What are the reasons?](#)
- Pretrigger** - allows setting a number of samples (bytes) to be stored until triggering. On the time base this waveform portion is negative, i.e. signal prior to triggering event is up to zero and signal upon triggering is after zero.
- Memory depth** - when pipe reading it is determined by available memory size in the computer and in the logical analyzer mode it is artificially limited by 256M. It is not recommended to set the buffer size more than several Mbytes as it is extremely inconvenient to analyze such long signals and by

the way, the memory is greatly applied to show (plot) charts on the screen.

- **Ch. [7]...[0]** - flags enable the data plotting in the corresponding channel of the bus A when pipe reading. As the standard sequential interfaces have a number of channels less than 8 then the bus A is always 8-bit but when plotting the charts of the sequential interface the computer capabilities can be applied in vain because of data plotting in the unused channels.
- **Channel for external clocking (bus B)** - allows selecting one of 8 channels of the bus B to connect the external clocking generator.
- **Bus A inverse (for new devices)** - this flag is only required for new devices as to reduce the circuit plate size the channels of the bus A are in inverse order.
- **Auto activate the trigger (triggering)** - enables to turn on the triggering automatically when pressing any button on the panel "Trigger".
- **Skip same bytes during search** - when this option is active then when searching it allows to skip a group of consecutive same bytes. For example, bus A contains the following bytes AA' BB CC AA" AA* AA DD AA"', if you need to find sequence AA then upon the first pressing the search button the marker will point AA', upon the second pressing the marker will point AA *and upon the third pressing it will be AA"' (if enable skip) or AA* (if disable).*

Show the reading mode info (buf/pipe) - enables to show information about the reading mode on the panel "Frequency".

Concatenation Mode.



Concatenation mode is to increase the depth memory for one channel when microcontroller buffer reading. The using microcontroller (C8051F321) allows selecting only 128 to 1544 bits for each channel when using only 8 channels as the logical analyzer.

Performance of the logical analyzer in the concatenation mode can be demonstrated by the following example. Let there is a device with triggering channel and data transmission channel, at that the data transmission starts, e.g. when triggering signal goes from high to low level. It is necessary to analyze all transmission considering that the information portion length proved to be larger than one to be read by this device at a time.

To read all data from the analyzed device, you need to do the following:

1. To illustrate analysis, let us name the channel A0, as SCK and the channel A2 as MOSI ([interface SPI](#)).
2. Turn on triggering as to the falling edge for the channel SCK.
3. Press the "Run" button and restart the analyzed device.
4. As the triggering condition lies in going SCK signal from high to low level then the start of timing charts is to match the start of transmission by the device.
5. Open the [statistics window](#) when pressing Ctrl+T and determine a number of falling edges for SCK channel, let it be 50.
6. Turn on [skip pulses](#) from falling edge and set a number of missing pulses, e.g. 48 to SCK channel.
7. Turn on the concatenation mode when pressing Ctrl+G.
8. Press the Run button and restart the analyzed device.
9. As the triggering condition holds constant then the start of timing charts is to match 48 triggering pulse.
10. Upon completing the measurements in the upper right corner of the work screen the button with icon of two concatenated sheets under which there are two buttons with left and right arrows shall appear. When moving the scroll bar slider right to the end of the main chart (solid lines) the chart to be pasted (dot lines) shall appear (there are results of the last measurement with missing 48 triggering pulse).
11. As there are 50 triggering pulses in the main chart but it was missed by 2

less then it is necessary to shift the chart to be pasted by 2 SCK pulses. For this purpose it is necessary to press the arrow button left and hold pressing until the start of this chart is shifted by two SCK pulses, at that the main chart is to match the chart to be pasted for the data channel under two "excessive" SCK pulses. So, e.g. the figure displays that the marker 2 points at the end of the main chart and the marker 1 points at the start of the chart to be pasted considering shift and as is clear from the figure it is necessary to shift this chart left a very little more to match it with the main one.

- 12.If the read concatenation matches the main chart incorrectly then you can, e.g. change a number of missing pulses and repeat all the actions from item 8. After that the previous read concatenation will be changed to the newly read one at that the duplicate concatenation measurement will have no influence on the main chart.
- 13.If the chart to be pasted matches the main one correctly you can press the Concatenate button and then the chart to be pasted continues the main one.
- 14.If transmitted data are incompletely read as before then you need to repeat the steps from item 5.

When using the concatenation mode it allows increasing the memory depth for one channel up to 1 Mbit, i.e. almost 1000 times for additional steps of end user.

Pins Layout

The Logic Analyzer cable pins layout:

- 1 - Red: 3.3V or absent for devices shipped after 10/25/2006
- 2 - line 7
- 3 - line 6
- 4 - line 5
- 5 - line 4
- 6 - line 3
- 7 - line 2
- 8 - line 1
- 9 - line 0
- 10 - Blue: Ground

Reading Modes.

This section describes the reading modes for analog signals, which are the same for discrete (digital) signal.

In this software two data reading modes are active:

- pipe reading
- microcontroller buffer reading

When **pipe reading**, the measurement process consists of the following steps:

1. Set the sampling frequency.
2. Start the pipe.
3. Read the packets of 409 samples ($512/1.25$ (10 bits)) from microcontroller in real time.
4. When computer reads the packet, microcontroller writes the samples to its FIFO (1024 bytes) to ensure measurement continuity.
5. Extract the packets, calibration, triggering as required.
6. Stop the pipe upon filling the preset buffer.

When microcontroller **buffer reading**, the measurement process consists of the following steps:

1. Set the sampling frequency.
2. Read 1126 10-bit samples (microcontroller buffer depth is $1024(\text{xdata}) + 256(\text{EP2}) + 128(\text{EP1}) = 1408$ bytes/ 1.25 (10 bits) = 1126 samples) at the preset sampling frequency and their compress in the internal buffer, triggering as required.
3. Notify the computer about samples obtained.
4. Transmit the microcontroller buffer to the computer.
5. Extract the buffer and calibrating the samples.
6. Display the measurement results.

It follows that when reading with the buffer it is impossible to read more than 1126 waveform samples for one continuous measurement. It is necessary to set frequency to enable normal viewing both low and high-frequency signals. So, e.g. if the sampling frequency is always 200 KHz then when analyzing tone with frequency of 1 KHz (i.e. 200 samples for signal period) $1126/200 = 5$ full signal periods would be displayed that is sufficiently and when analyzing tone with frequency of 50 Hz (i.e. 4000 samples for signal period) then less than 1/3 of period would be displayed that is unacceptable for any analysis.

The question arises why we use reading with microcontroller buffer. The reason is

that not all computers are equipped with USB1.1 and NT-base operating system, i.e. Windows 2000 and XP. This is important because the NT based operating systems ensure the general process priority and processor resource allotment allows for smooth transfer of data on USB.

As the data exchange process is continuous and run in real time, increased demands are imposed on the computer (especially on USB bus controller) and operating system.

To ensure the maximum sampling frequency when pipe reading, USB bus controller must manage to read 512 data bytes (EP3) from MK End Point each millisecond. At that rate, the operating system must allocate the necessary resources for the data reading process, i.e. generally keep the process up-to-date and the pipe priority high. If you have at least Celeron 600MHz, 128M RAM and operating system Windows 2000 or XP they have to run at maximum frequency (as tested on 3 computers). Of course if any resource-intensive application is concurrently operated. performance will suffer accordingly.

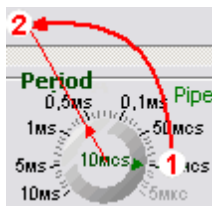
If, upon connecting the device, your computer gives an "Out of bandwidth" error message, you can operate at the maximum sampling frequency, partly because of low-speed computer and partly because of the device capabilities. In this case the microcontroller buffer reading mode can help you as in this mode the maximum sampling frequency is independent on the features of your computer.

But you can use the pipe reading even in the low-speed computers when selecting the maximum sampling frequency by experiment and if more it will be the automatic buffer reading as, e.g. in Celeron 300MHz, 64M Windows 98, the pipe reading run at $F_s \leq 30$ KHz, in Celeron 600, 128M Windows 98 at $F_s \leq 100$ KHz but in Windows 2000 the pipe reading run at the maximum frequency. It is also necessary to note that when concurrently applying 2 channels the sampling frequency doubles as, e.g. at $F_s = 100$ KHz the sampling frequency is actually $F_s = 200$ KHz. That's why when concurrently operating 2 channels the sampling frequency setup is automatically disabled if higher 100 KHz.

The knob.

Using the knob(s) allows positioning more discrete values on the same area compared with the buttons. The set value is always shown in the knob-center, so it is not necessary to find what button is pressed. Moreover, working knob looks more aesthetically pleasing than buttons.

These control knobs were specially designed to be used in USB oscilloscope, that's why there is a number of particularities to know when usefully operating these controls that can save you time and increase your work comfort and productivity.

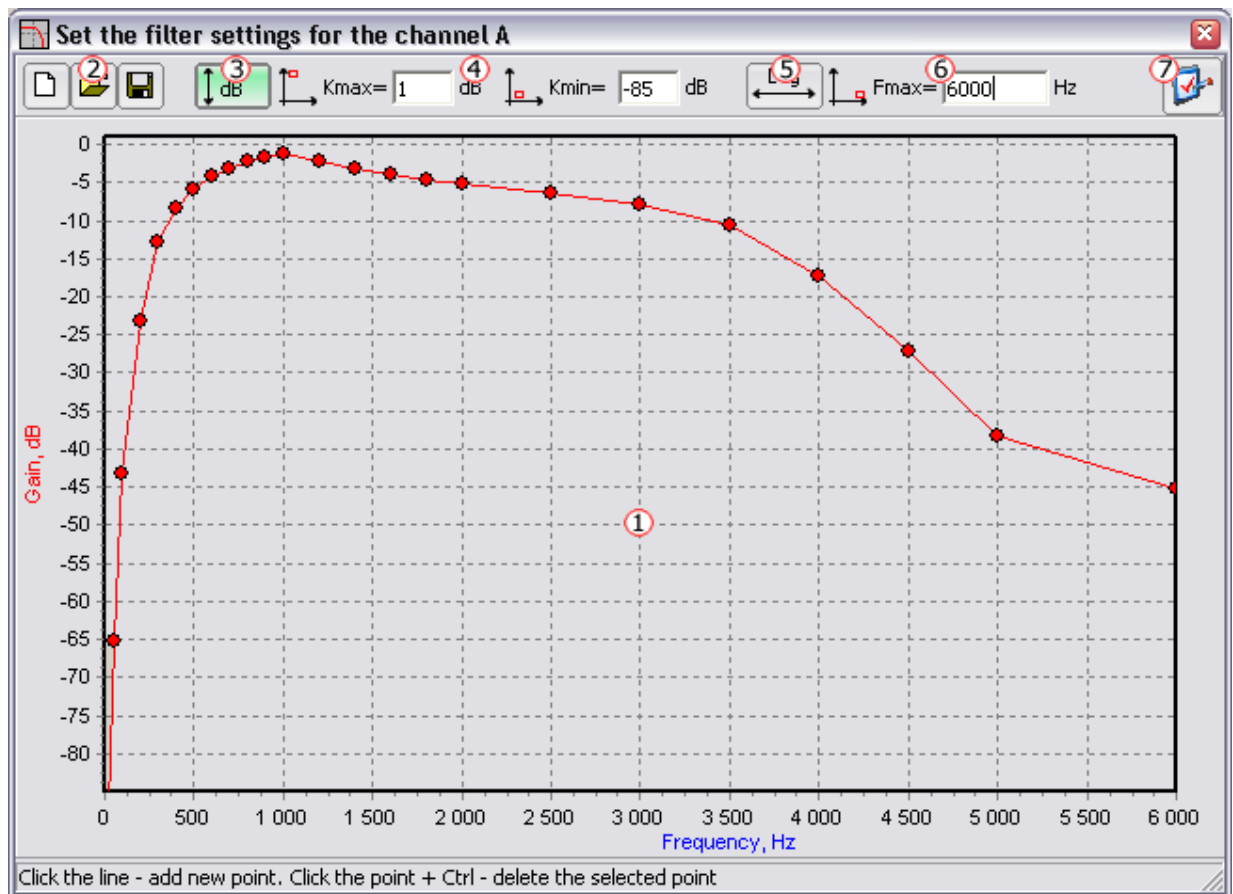


Methods to change the knob position:

1. Using the **mouse** - when left clicking on the knob(s) (cursor takes a hand form) and moving the mouse but not exactly over the knob. When moving the mouse the knob cursor will be positioned along a radial line drawn from the center of the knob to the mouse cursor position, i.e. like mechanical control knob, and very intuitive - you move the edge of the knob's dial to the desired value. Upon clicking on the knob this element is within the input focus, i.e. can be key- or mouse wheel operated even if the mouse cursor is not on the knob. It is useful if you know that most of other usable elements are out of input focus, i.e. you can change the different triggering modes/levels and adjust the sampling period or peak-to-peak voltage by the mouse wheel.
2. Using the **mouse wheel**.
3. Using the **keys** Left, Right, Up, Down, Home, End, PageUp, and PageDown.

Also, note that when the mouse **cursor** is over the knob, a prompt regarding the value to be set/set is shown on the status bar.

Digital Filter



The figure displays the filter settings window. Diagram AFC ① of the filter to be calculated occupies the major portion of the window.

The horizontal axis of this diagram is the **frequency axis**.

The vertical axis is the **gain axis**, the scale of which can be either a multiplication factor or decibels.

Necessary AFC is constructing by using the mouse. Diagram AFC consists of the reference points and straight lines connecting these reference points. That is, the gain values between the user-defined reference points are calculated by linear approximation.

AFC construction starts when adding new reference points by clicking the line (if the mouse cursor is over the reference point or line the cursor takes a hand form). Upon clicking the line new reference point appears, which can be moved both vertically and horizontally (except for the end reference points) - but not further

than the adjacent reference points.

To move the reference point(s) it is necessary to drag it with the mouse cursor, which has to take a 'hand' form, and then left click and start moving the reference point in the selected direction when holding the left mouse button pressed. When moving the reference point both of the current coordinates will be shown on the status panel: gain and frequency factors. To delete a reference point, point at it with the mouse cursor and left click when holding the Ctrl key pressed.

In the upper part of the filter settings window there is the tools panel. At the top of the tools panel there is a row of buttons ② that allow undoing the changes made, as well as reading the filter parameters from the file and saving the filter parameters in the file.

Button ③ determines dimension of the vertical axis if the button is pressed then the axis is in terms of decibels otherwise the axis is in terms of multiplication factors.

Input field "Kmax" and "Kmin" ④ sets the limits of the vertical axis either in times or in decibels.

Button ⑤ determines the scaling of the vertical axis: either linear or logarithmic. If the button is pressed then the frequency axis is logarithmic otherwise it is linear.

Input field "Fmax" ⑥ is the maximum value of horizontal axis frequency.

Button ⑦ determines the filter settings and closes the work window.

Sounds



The figure displays the event and sound settings window. Sounds are active **only** when operating in the oscilloscope or spectrum analyzer mode. Sounds may be useful when searching the set waveform, e.g. searching the circuit with voltage of 3.3 V. or for indicating an out-of-range or undesirable value.

Functionally, the window is divided into two side by side panels - one panel for each channel (A & B). Each panel is further divided into 3 subgroups of settings, each one for one of the measured signal parameters (DC component of the voltage, AC component of the voltage, and signal frequency).

For each of the measured parameters, 3 'trigger' events can be specified:

- play the specified sound if the parameter is **higher** than set threshold,
- play the specified sound if the parameter is **lower** than set threshold, and
- play the specified sound if the value is **approximately equal** to the set threshold.

The threshold is set in the field positioned left of the corresponding parameter.

The sounds are set by clicking the relevant dropdown list and selecting one of the standard sounds or pressing the relevant button with the open hard drive icon (on the right) and selecting the desired sound file.













To listen to the selected sound you can click the playback button (positioned on the same bar with event name) with loudspeaker icon (far right side). All standard sounds are reproduced with the built in speakers, selected sound files are played when using the computer sound card, if any. To disable all sounds for the selected channel, click the Disable sounds button at the bottom of the corresponding panel.

Also, note that when comparing the parameter with the set threshold there is a small delta (a certain range around the set value which will trigger the event). This means the sound will occur approximately at the threshold, but not necessarily exactly.

- For **voltage** the delta is determined as **$\pm 5\%$ of the scale range** of the measured channel, e.g. if for the channel A ± 10 Volt/Screen is set and value $U_{\text{const}} = 3.3$ V then sound will be reproduced if DC voltage component for the channel A is within 2.8...3.8 V.

For **frequency** the delta is always DC and equal to **± 20 Hz**.

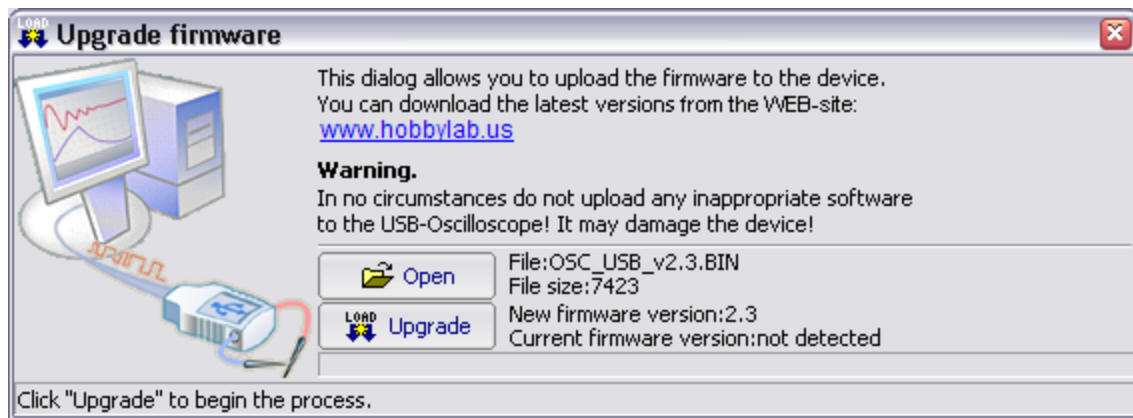
Statistics

| Statistics | | | | | | | | | |
|--------------|----------|----------|---|---|---|---|---|---|------------|
| Bus A | | | | | | | | | |
| Channel | Number ↗ | Number ↘ |  |  |  |  |  |  | Favg |
| Channel A0 | 131 | 130 | 15,00 mcs | 15,00 mcs | 20,00 mcs | 20,00 mcs | 19,62 mcs | 19,58 mcs | 25,52 kHz |
| Channel A1 | 0 | 0 | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 Hz |
| Channel A2 | 33 | 33 | 35,00 mcs | 35,00 mcs | 200,00 mcs | 315,00 mcs | 78,13 mcs | 77,42 mcs | 6,43 kHz |
| Channel A3 | 0 | 0 | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 Hz |
| Channel A4 | 0 | 0 | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 Hz |
| Channel A5 | 0 | 0 | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 Hz |
| Channel A6 | 0 | 0 | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 Hz |
| Channel A7 | 0 | 0 | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 mcs | 0,00 Hz |
| Bus B | | | | | | | | | |
| Channel | Number ↗ | Number ↘ |  |  |  |  |  |  | Favg |
| Channel B0 | 512 | 511 | 5,00 mcs | 5,00 mcs | 5,00 mcs | 5,00 mcs | 5,00 mcs | 5,00 mcs | 100,00 kHz |
| Channel B1 | 256 | 255 | 10,00 mcs | 10,00 mcs | 10,00 mcs | 10,00 mcs | 10,00 mcs | 10,00 mcs | 50,00 kHz |
| Channel B2 | 128 | 127 | 20,00 mcs | 20,00 mcs | 20,00 mcs | 20,00 mcs | 20,00 mcs | 20,00 mcs | 25,00 kHz |
| Channel B3 | 64 | 63 | 40,00 mcs | 40,00 mcs | 40,00 mcs | 40,00 mcs | 40,00 mcs | 40,00 mcs | 12,50 kHz |
| Channel B4 | 32 | 31 | 80,00 mcs | 80,00 mcs | 80,00 mcs | 80,00 mcs | 80,00 mcs | 80,00 mcs | 6,25 kHz |
| Channel B5 | 16 | 15 | 160,0 mcs | 160,0 mcs | 160,0 mcs | 160,0 mcs | 160,0 mcs | 160,0 mcs | 3,13 kHz |
| Channel B6 | 8 | 7 | 320,0 mcs | 320,0 mcs | 320,0 mcs | 320,0 mcs | 320,0 mcs | 320,0 mcs | 1,56 kHz |
| Channel B7 | 4 | 3 | 640,0 mcs | 640,0 mcs | 640,0 mcs | 640,0 mcs | 640,0 mcs | 640,0 mcs | 781,3 Hz |

The figure displays the statistics window when using both buses; i.e. when the bus B is active either in the [logic generator mode](#) or in the mode of additional logical analyzer. Statistics window contains the following elements: to the left there are the names of all usable channels, number of rising (front) and falling (rear) edges of the measured channel and then minimum/maximum/average length of signal pulse and pause, at that the average length is calculated for the whole signal but as arithmetic average between minimum and maximum length and average signal frequency.

Change in any data of the logical analyzer or generator immediately leads to recalculation of all statistics. To view statistics on-line you can enable the statistics window [always on top](#), at that you avoid constant clicking it to move it higher the main window.

Firmware Upgrade



The figure above shows the device firmware upgrade window. This window is necessary because the software for USB oscilloscope consists of two programs:

- **shell program** that the user works with (a Windows application), and
- **embedded program firmware** uploaded in the microcontroller.

The embedded firmware performs the preliminary processing of the measured signals and transmits them using the USB bus to the shell program running on a PC, and the shell application receives this information and formats it in a user-friendly presentation on the PC screen.

Shell upgrade

Shell upgrades are inevitable. There will inevitably be minor defects and errors in the program that go unnoticed by the author/programmer, usually for minor functions or aesthetic defects. These are frequently corrected in new releases, and can be eliminated in previously installed software by downloading the latest shell version and exchanging the old file `usb_osc2.exe` (installed into the `C:\Program Files\USB Oscilloscope` folder by default) for the new file. For the latest shell updates, see: [Download](#).

Embedded Program Firmware Upgrades

If it is necessary to upgrade the embedded firmware (I.e., changes, improvements, and fixes which cannot be accomplished just by changing the shell file, e.g. to organize the additional 8 analog channels of the [recorder](#) instead of the bus A of the [logic generator](#)). The firmware upgrade window allows you to download and install the latest version of microcontroller firmware.

To upgrade, you need to:

- **open the firmware** file by pressing the **Open** button, and then
- **the Upgrade button becomes active** and information about the firmware file size and version will be displayed. When opening this file,

any error conditions will be displayed. Read these carefully, and fix them before proceeding (usually by redownloading the firmware file)

- **press the Upgrade button** and the device firmware upgrade will start. The progress indicator displays (above the status panel) a message about the upgrade status. Upon completing the upgrade a verification message window will appear. **IMPORTANT: DO NOT DISCONNECT THE USB CABLE OR SHUT OFF THE PC UNTIL THIS PROCESS COMPLETES.**

Attention! *Under no circumstances should you attempt to upload an inappropriate file to the microcontroller (be SURE to use only appropriate files downloaded from [Download](#)) - typically with a filename like "OSC_USB_v2.4.BIN" - by right clicking its link and choosing 'Save As'. DO NOT open this file using any other programs (like a text editor, web browser, etc.), as you may corrupt the file (possibly with characters that are not displayed).*

It is important to follow these instructions carefully, because upon completing the updated firmware upload, microcontroller control is transferred to the uploaded program, which in turn enables communications with the loading program. If the uploaded program fails to communicate with the loading program, your device will be completely inoperable, and the working firmware version cannot be uploaded any more (since the device no longer communicates).

Standard interfaces.

When testing any digital device you may need to analyze one or more data communication channel(s). In particular, you may want to see explicitly how the timing of these signals relate to each other.

At first glance, it appears to be a trivial task to take the timing charts from the necessary communication channel lines, determine the interface type and manually decode the recorded timing charts. But after making a few attempts to decode such timing charts, you may find it obvious that you do not want to perform this routine but tedious and laborious procedure very often.

PC USB oscilloscope can help by presenting some standard communications hardware interface protocols in the clearest possible format for your analysis. To start decoding you must become familiar with specification of the analyzed interface and master its timing chart formats. This can take quite a lot of time, especially when you do not have a normal interface description handy. After familiarizing yourself with the interface you may find that decoding is actually interesting.

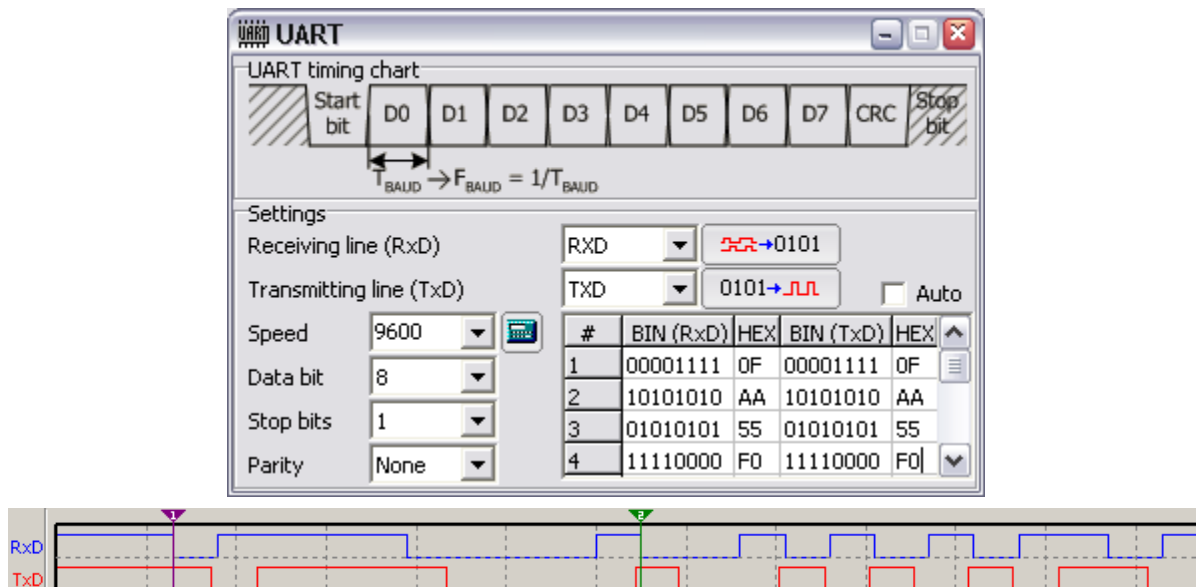
Generally, in most of interfaces there are some signs of bytes transmission start and end, except for [SPI](#), when the data are transmitted without any delay. That's why decoding starts with the signs searching. Upon finding the start transmission sign, one starts decoding the data. This may raise questions about particularities of the analyzed protocol that require examination the specification of the analyzed interface, comparing the timing charts with their analogs in specification.

This can take a lot your time and divert you from other and probably more interesting tasks. We can state from experience about time expenditure required for interface timing charts decoding [I2C](#) (used in video capture board). When manually decoding for three and a half hours we managed to allot even 48 bytes, and even at that slow rate all the obtained information was thoroughly recorded on a paper subject to many potential errors. When decoding the same interface with analogous program (subsequent pilot model of logical analyzer) the whole communication cycle of 1026 bytes long was analyzed for 8 minutes but it took one and a half day to write the analysis program [I2C](#).

This does not mean that without any preliminary preparation one can decode the data of any interface by pressing a couple of buttons for several minutes. In any case, it is necessary to read the selected interface specification before analyzing it, as no program can help you without unless you comprehend the interface operation

principles.

UART interface.



The figure displays UART interface window, beneath it there is an image with work screen area containing the timing charts for the receiving channels (RxD) and transmitting channels (TxD). Marker 1 is set to the start bytes 0x0F and marker 2 is set to the following start byte 0xAA of the channels RxD.

The upper part of UART window contains an image of typical timing chart of one byte of UART interface. Beneath it there is the panel of the main settings incident to the analyzed interface. Before you start decoding it is recommended to set the names of the interface channels on the panel "Channels" of logical analyzer. If you set the names RxD and TxD they automatically appear in the fields of receiving and transmitting channels.

As is known from UART interface specification, start bit that is always logical zero goes before the data bit transmission, then the data bits are transmitted, at that the low bits come first and then the parity bit can go and requisite one or two stop bits that is always a high level. If there is not data communication then the logical level of the corresponding channel (TxD or RxD) will be always high. That's why it is expedient to turn on triggering as to the falling edge in one of the channels advisably in the same channel, where stratum bit goes first. Before you start measuring it may be well to turn off the analyzed device and then press the Start button and turn on the device. If triggering is set correctly then start of timing charts has to concur with start of communication. For the normal timing charts decoding it is necessary that the sampling frequency is 3-4 times higher than communication rate (frequency), in rough way, rate is inversely proportional to the minimum pulse length, e.g. for the channel RxD.

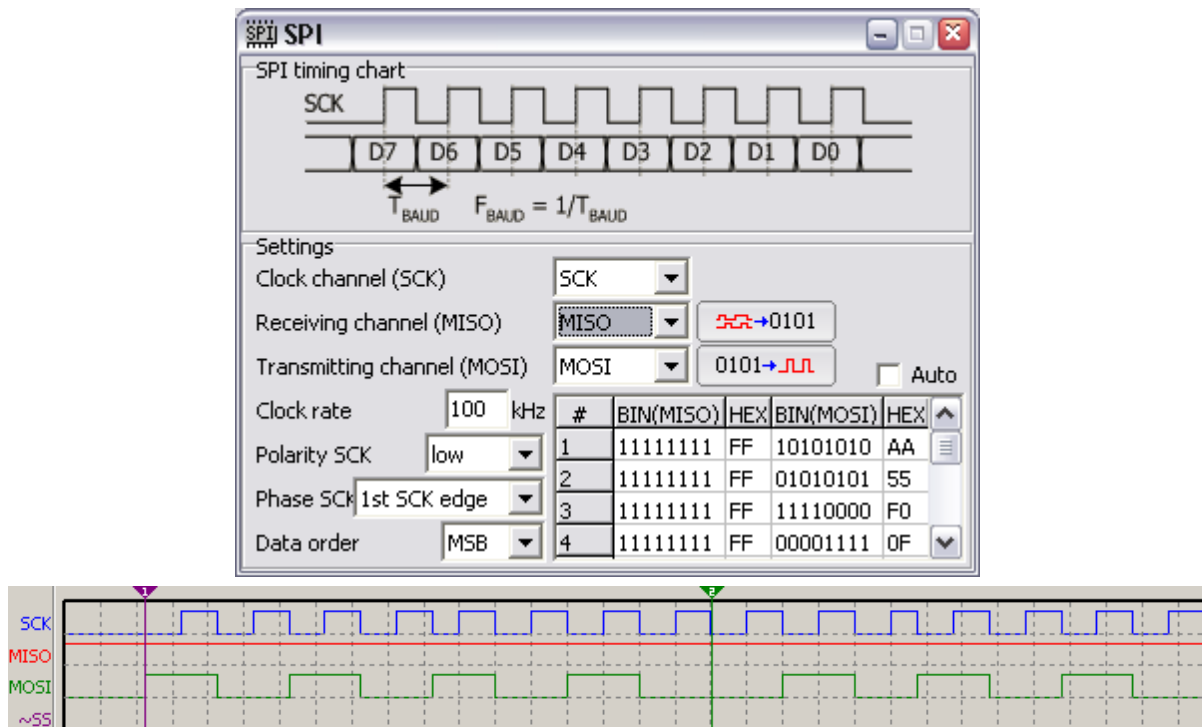
Upon taking the timing charts it is necessary to determine the communication rate. To determine the communication rate automatically you may press the button with calculator icon. Communication rate is determined automatically only on the basis of the timing chart of the channel RxD and if possible, corrected on the basis of the standard series of speeds. Upon determining the communication rate it may be well to set the values of other interface settings correctly, which cannot be determined automatically. The most common settings of UART interface are set as default. Upon setting all the parameters it is desirable to set the marker 1 to start bit of the channel RxD and marker 2 to start bit of the channel TxD otherwise the first 1-3 bytes can be skipped or decoded incorrectly.

Upon completing all such procedures you can start decoding when pressing Decode button. Decoding results will be displayed in the table in binary and hexadecimal formats. When double clicking on the selected cell with the decoding results the marker 1 goes to start of the corresponding byte.

It is necessary to note that in addition to the timing charts decoding UART window also allows generating the timing chart on the basis of interface settings and table data (column TxD) for the channel TxD, which has to be one of the logical generator channels.

It makes sense to use the flag Auto (table field autofill) only if signal generation is required.

SPI interface.



The figure displays SPI interface analysis window, beneath it there is an image with work screen area containing the timing charts for the clock channel (SCK), select slave channel (~SS), receiving channels (MISO) and transmitting channels (MOSI). Marker 1 is set to start byte 0xAA and marker 2 is set to start of the following byte 0x55 of the channel MOSI.

The upper part of SPI windows contains an image of typical timing chart of one byte of SPI interface. Beneath it there is the panel of the main settings incident to the analyzed interface. Before you start decoding it is recommended to set the names of the interface channels on the panel "Channels" of logical analyzer. If you set the names SCK, MISO and MOSI then they automatically appear in the fields of the clock channel, receiving channel and transmitting channel.

As is known from [SPI interface specification](#), there are not any special signs to determine start byte, if the data are transmitted without pause and select slave signal is not used. The only way to determine start byte is to start measuring before communicating with "correct" triggering in SCK channel. As is known from SPI interface specification, SCK channel level in the standby mode (if no communication) may be both high and low. That's why to set "correct" triggering it is necessary to determine SCK level in the standby mode. To this effect it is necessary to perform the measurements several times before a long pulse/pause appears in SCK channel, which length has to differ sufficiently from adjacent

pulses/pause. If a long pulse is found then SCK channel level in the standby mode is high, i.e. triggering has to go as to the falling edge if a long pause is found then SCK channel level in the standby mode is low and triggering has to go as to the rising edge. Before you start measuring it may be well to turn off the analyzed device and then press the Run button and turn on the device. If triggering is set correctly then the start of timing charts has to concur with start of communication. For the normal timing charts decoding it is necessary that the sampling frequency is 3-4 times higher than the triggering frequency.

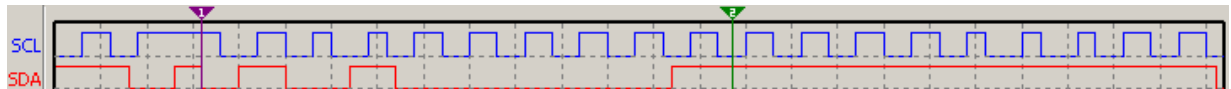
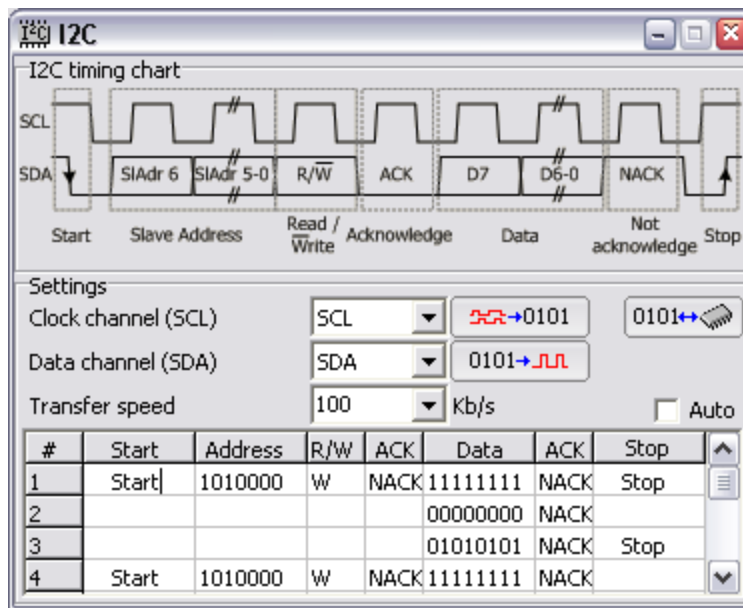
Upon taking the timing charts it is necessary to set the values for all interface settings correctly as without it you cannot start the waveform decoding. The most common settings of SPI interface are set as default. Upon setting all the parameters it is necessary to set the marker 1 to start communication (byte).

Upon completing all such procedures you can start decoding when pressing the Decode button. Decoding results will be displayed in the table in binary and hexadecimal formats. When double clicking on the selected cell with the decoding results the marker 1 goes to start of the corresponding byte.

It is necessary to note that in addition to the timing charts decoding SPI window also allows generating the timing chart on the basis of interface settings and table data (column MOSI) for the channels SCK and MOSI, which are to be one of the logical generator channels.

It makes sense to use the flag Auto (table field autofill) only if signal generation is required.

I2C interface.



The figure displays I2C interface analysis window, beneath it there is an image with work screen area containing the timing charts for the clock channels (SCL) and data channels (SDA). Marker 1 is set to start signal and marker 2 is set to start data byte transmission 0xFF.

The upper part of I2C windows contains an image of typical timing chart of I2C interface address and one byte transmission. Beneath it there is the panel of the main settings incident to the analyzed interface. Before you start decoding it is recommended to set the names of the interface channels on the panel "Channels" of logical analyzer. If you set the names SCL and SDA then they automatically appear in the fields of clock channel and data channel.

As is known from [I2C interface specification](#), the data transmission session starts with the start signal transmission (when SDA goes from high level to low level while SCL is in high level) then address or data bits are transmitted, at that the high-order bits go first and then confirmation bit is transmitted. Stop signal completes the data transmission (when SDA goes from low level to high level while SCL is in high level). If there is not data communication in the bus then the logical level of SCL and SDA lines will be always high for the raising resistors. That's why it is expedient to turn on triggering as to the falling edge for the channel SDA. Before you start measuring it may be well to turn off the analyzed device and then press the Start button and turn on the device. If triggering is set correctly then the start

of timing charts has to concur with start of communication. For the normal timing charts decoding it is necessary that the sampling frequency is 4 times higher than the triggering frequency.

Upon taking the timing charts it is desirable to set the marker 1 to start signal otherwise the nearest start signal right to the marker 1 will be searched and then one session may be skipped.

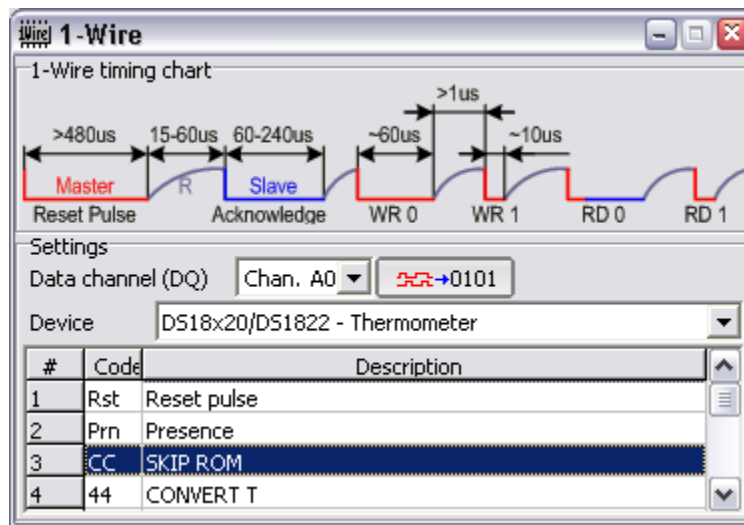
Upon completing all such procedures you can start decoding when pressing the Decode button. Decoding results will be displayed in the table in character format (start, stop, communication direction signal and confirmation signal) and in binary format (address and data). When double clicking on the selected cell with the decoding results the marker 1 goes to start of the corresponding I2C interface element.

It is necessary to note that in addition to the timing charts decoding I2C window also allows generating the timing charts on the basis of interface settings and table data (columns, address, R/W, data) for the channels SCL and SDA, which are to be one of the logical generator channels.

It makes sense to use the flag Auto (table field autofill) only if signal generation is required.

Button with microcircuit icon opens the terminal window - hardware support (input/output) of the analyzed interface.

1-Wire interface.



The figure displays 1-Wire interface analysis window, beneath it there is an image with work screen area containing the timing chart of data channel (DQ), at that the measurement was performed at $F_s = 200$ KHz but the figure displays the timing chart for $F_s = 50$ KHz. Marker 1 is set to start reset pulse and the marker 2 is set to start bytes transmission 0xCC.

The upper part of 1-Wire windows contains an image of typical timing chart of 1-Wire interface. As in the timing chart the red lines imply that bus is moved to logical zero by master, blue ones - by slave and gray ones imply that the bus is moved to logical item by raising resistor. Beneath it there is the panel of the main settings incident to the analyzed interface. Before you start decoding it is recommended to set the names of the interface channels on the panel "Channels" of logical analyzer. If you set the name DQ, then it automatically appears in the field of data channel.

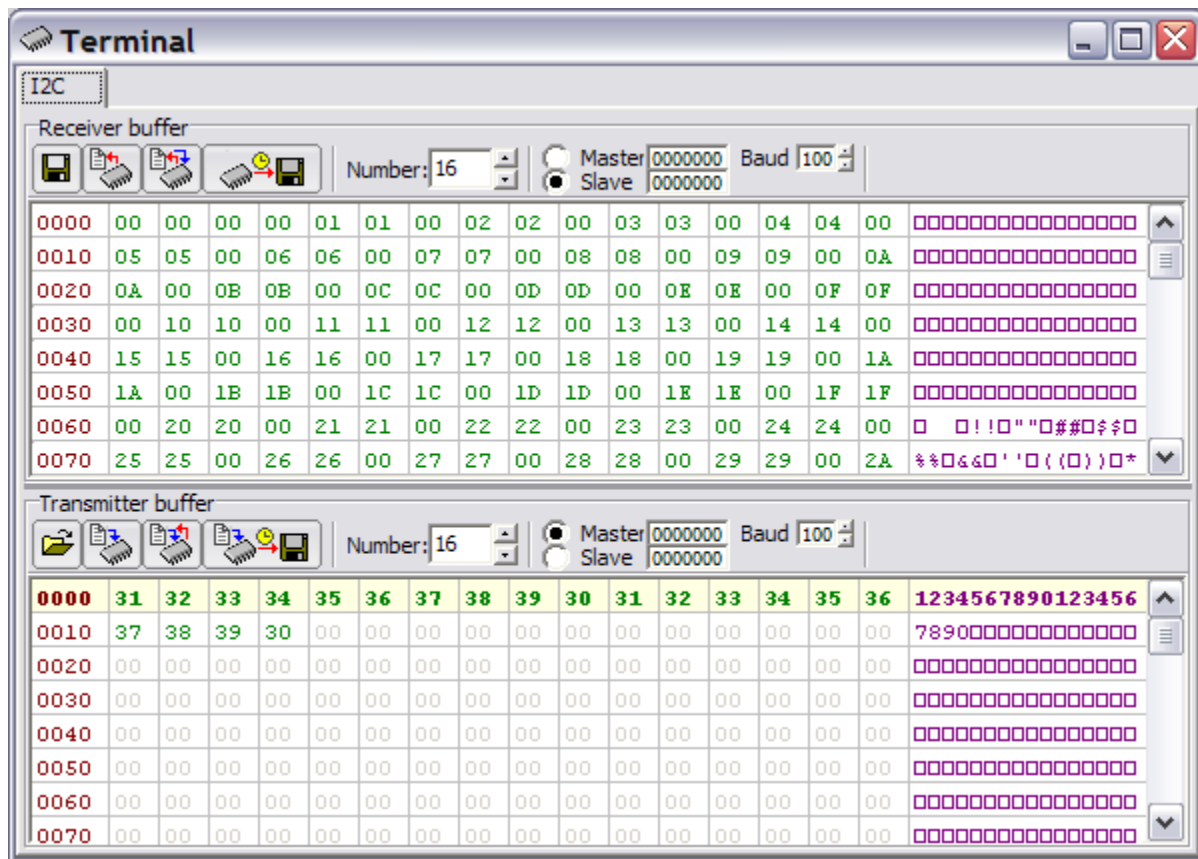
As is known from [1-Wire interface specification](#), the session starts when master sets the reset signal (low level of more than 480 uS long) and then confirmation signal is set in 15-60 uS (low level of 15-60 uS long) if there is only one slave in the bus. Upon receiving the confirmation signal the master transmits one or more bytes, at that the low-order bits go first, slave responses master as required. As the session starts when DQ line goes from high to low logical level then it is expedient to turn on triggering as to the falling edge. Before you start measuring it may be well to turn off the analyzed device and then press the Start button and turn on the device. If triggering is set correctly then the start of timing chart has to concur with start of communication. For the normal timing charts decoding it is necessary that

the sampling frequency is at least 200 KHz.

Upon taking the timing charts it is desirable to set the marker 1 to reset signal otherwise the nearest reset signal right to the marker 1 will be searched and then one session may be skipped. For detailed decoding and description of transmitted bytes you can select one of the standard devices supporting 1-Wire interface, of course if its instruction set is in agreement with the instruction set of analyzed device. There is only one type of devices - thermometers DS18x20/DS18x22 in the dropdown list that can be subsequently added by other devices.

Upon completing all such procedures you can start decoding when pressing the Decode button. Decoding results will be displayed in the table in hexadecimal format (Code) except for start and confirmation signal and in character format (Description). When double clicking on the selected cell with the decoding results the marker 1 goes to start of the corresponding element of 1-Wire interface.

Terminal.



The figure displays the I2C interface hardware support.

The Terminal dialog allows reading and writing bytes to the I2C bus. All four I2C modes are supported:

1. **Master Transmitter**, i.e. we notify that we shall transmit the data
2. **Master Receiver**, i.e. we notify that we shall receive the data
3. **Slave Receiver**, i.e. we are notified that we have to receive the data
4. **Slave Transmitter**, i.e. we are notified that we have to transmit the data

The signals need to be set: SDA - B0, SCL - B1

It is also possible to set the number of receiving / transmitting bytes, the Slave addresses and the transfer speed in Master mode.

Note: The following operations will be implemented in the future releases:

- Read the number of bytes and write the number of bytes to the external device
- Write the number of bytes and read the number of bytes from the external

device

- Start pipe reading from the external device

Write the number of bytes and start pipe reading from the external device

Device Settings.

The USB Oscilloscope program operates satisfactorily on any PC on which Windows 2000/XP will run smoothly, i.e. **at least Celeron 500MHz and 64MB RAM**. In lesser computers, it is likely that the program will operate incorrectly. This is especially so in computers running Windows 98/ME, where upon pressing the **start measurements** button it changes its name to **Stop** when no measurement results are displayed. This is related to "slow" event processing of USB bus, i.e. the system just has not enough time to transmit all information within the fixed period.

To eliminate this undesirable behavior on slower computers, it may be necessary to open the file `usb_osc.ini`, which will be on the hard drive where it was downloaded upon installation (by default in `C:\Program Files\USB Oscilloscope`) and find the lines:

```
[system]
usb_wr_timeout=50 - write timeout, ms
usb_rd_timeout=50 - read timeout, ms
```

These two parameters set the time interval in milliseconds - ms (timeout) within which the system has time to transmit (wr) and read (rd) data block from the device. Both timeouts are equal by default to 50 ms.

For example, a PC with an Athlon XP 1700MHz CPU and 512MB RAM running Windows XP transmits for 50 ms, while a PC with a Celeron 600MHz CPU and 192MB RAM running **Windows XP** the timeout value can be increased up to 100 ms though it runs for 50 ms. But with a Celeron 600MHz and 192MB RAM running **Windows 98** the timeout values have to be at least 400-500 ms for normal operation.

These tests were also conducted for PC with Pentium 200 and 32MB RAM running Windows 98 where curiously it was seen that the program runs even in this computer, but the required timeouts were about 1000 ms.

Moreover, there are also two important system parameters in the `usb_osc.ini`:

```
usb_reset_timeout=50 - delay after reset, ms
wait_reset_trigger=300 - delay before reset of hang triggering, ms
```

Upon resetting the device, it is necessary to leave an interval to allow Windows to have time to configure the device, i.e. perform the standard start-up functions, allocate address and etc. The slower the computer, the longer

these steps take, which it means that the delay will be longer as well. The **usb_reset_timeout** should be approximately equal to the communication timeout, i.e. when increasing any timeout it is necessary to increase the delay after reset.

The value of **wait_reset_trigger** determines the time allowed to reset the device if the triggering parameters are changed and the device shows no response, i.e. it hung. We can explain this parameter by the following example: if we turn on absolute triggering, set the level 5 V, maximum signal level 2 V, i.e. triggering condition will be never met because this device will be constantly in standby mode and no changes are shown on the screen. We change the triggering level up to 1 Volt, upon this change the program is waiting for `wait_reset_trigger` ms, if it is no response from the device that is expected in this case because it is waiting to execute an impossible triggering condition then the device is reset and measurements are completed. This delay is necessary to enable the device to response if triggering conditions are executable that is not often, e.g. every 250 ms. If previous triggering condition is executed all the same then the device will not be reset within `wait_reset_trigger` upon changing the triggering condition and it means that measurement will not be reset and the results will be displayed.

There is the additional line in the section [system]:

```
show_exchange_error=1
```

This line was added to indicate some problems when operating the device and laptop as well as those related to the installed old driver version were detected. They are as follows: upon starting the program only one tab of logical analyzer was displayed because software determined it as a cut-off version due to the core driver (Silabs USBXpress driver) read data from the device incorrectly. As no error messages are displayed and we have not faced these problems in any of 10 computers where the device was tested and all the devices from the first lot were operated correctly as well then possibility of this error was not considered. This line was added to determine why, i.e. which error occurred within communication. If this device runs incorrectly or generally is out of operation then it is kindly recommended to set the value of this parameter to 1 and send me the error codes but if it runs satisfactorily then you can disable this function and set the value of the parameter to 0.

Shortcut Keys.

| Shortcut keys | Description |
|---------------|--|
| Ctrl+O | Open file with data of oscilloscope, spectrum analyzer, recorder or logical analyzer. |
| Ctrl+S | Save measurement results and current parameters of selected operation mode (current selected tab) in file. |
| Shift+Ctrl+S | Save measurement results and current parameters of selected operation mode (current selected tab) in file. |
| Ctrl+Alt+S | Save measurement results of selected operation mode as bitmap in *.bmp format (Windows Bitmap). |
| Shift+Ctrl+P | Open measurement results preview window. |
| Ctrl+P | Print measurement results of selected operation mode. |
| Ctrl+Q | Close program. |
| Shift+Ctrl+C | Copy measurement results of selected operation mode as vector graphic in *.emf format (Enhanced Metafile) to Windows buffer. |
| Shift+Ctrl+X | Copy measurement results of selected operation mode as bitmap in *.bmp format (Windows Bitmap) to Windows buffer. |
| Ctrl+N | Clear measurement results of selected operation mode. |
| Ctrl+R | Open comments input window. |
| Ctrl+Z | Open event and sound settings window. |
| Ctrl+G | On/Off concatenation mode of logical analyzer charts. |
| Ctrl+K | On/Off keep the generator output levels mode. |
| Ctrl+M | Select generator data portion in table. |
| Ctrl+T | Open statistics window. |
| Ctrl+A | Select all data table of logical analyzer. |
| Ctrl+D | Go from direct logical analyzer data edit mode to select mode. |