MSO Demo Board Manual

Document Version 1.05 (20th January 2025)

Table of contents

Introduction

The Batronix MSO Demo Board provides a wide variety of signals and experiments to demonstrate the functions of your oscilloscope or logic analyser, all on a compact board. This makes it ideal for use in training and education, from the beginner to the advanced user.

Chapter 1 of this manual provides an overview and general information about the demo board. The training section begins with Chapter 2, where you will learn step-by-step how to use your oscilloscope with the help of the demo board. Chapter 3 is about decoding interfaces and Chapter 4 is about radio frequency (RF) effects. Throughout the tutorial there is always room for your own experiments, while at the same time a possible solution to the experiment is presented. The wide range of variations allows for an individual measurement experience.

1 Overview demo sections

Section	Description
1	Power supply, USB type B (5 V DC)
2	Configuration switches • Select 1: Signal type of the function generator and parameters of various serial interfaces • Select 2: Frequency or other signal characteristics (duty cycle, error rate, ...) • Reset button: E.g. reinitialization of counter values used in serial messages
3	Slotted holes for easy access to measurement of digital signals, e.g. using probe clamping hooks
4	Pin header (2.54 mm) for measuring available digital signals, e.g. for logic analyzer Dupont connectors • J1: Serial interfaces (SPI, I2C, UART) • J2: 7-bit bus or CAN-/CAN-FD-/LIN-/Manchester communication
5	Output of PWM signals • "PWM LED": 100 kHz PWM with configurable duty cycle, also drives the LED • "PWM": PWM signal with configurable frequency and duty cycle • "PWM comb": PWM signal mixed with slower square-wave signal
6	Jumper configuration • Selection of termination resistors for sections 7 and 8 • Connection of the Bode plot demo filter to the BNC socket J6 • De-/Activation of available RF test signals
7	RF test track for demonstration of RF effects and terminations
8	BNC sockets • "TERM IN" (J6): BNC input socket with adjustable termination (33/50/75/open), also input for the Bode plot demo filter • "RF OUT" (J7): BNC output socket for the RF signal
9	Signal taps of the Bode plot demo filter • IN: Filter input signal • LP: Low-pass filter output • BP: Band-pass filter output
10	Signal output of the function generator • "DAC BUF" (J4): Amplified but largely unfiltered output of the DAC • "DAC FLT" (J5): Amplified and low-pass filtered output of the DAC

1.1 Power supply (section 1)

The demo board is supplied with 5 V DC via a USB socket. To do this, please connect the supplied USB cable to the demo board and the opposite side of the cable to the USB port of your oscilloscope, your PC or to a suitable USB plug-in power supply. The green LED (PWR) located close to the USB socket should light up clearly visible.

1.2 Input elements (section 2)

There are two selector switches on the demo board. These can be used to configure many aspects of the demo board. For example, the Select 1 switch selects the waveform of the function generator, while Select 2 can be used to vary the frequency. All possible settings are described in the following sections. Long-life selector switches have been used, which are not very easy to turn. A probe adjustment tool or small screwdriver can be used to turn the selector switches if necessary.

Counter values used in some serial messages can be reset using the reset button.

1.3 Digital signals (sections 3 and 4)

On the left-hand side of the board, there are taps for various standard interfaces (SPI, I2C, UART) with bidirectional communication, as well as a 7-bit digital bus that also supports CAN, CAN-FD, LIN and Manchester communication.

Details of the various setting options for these interfaces are explained in Chapter 3 and are presented in tabular form for each interface.

1.4 PWM signals (section 5)

The PWM signals "PWM LED", "PWM" and "PWM comb" are located on the left below the large gold ground area.
- The "PWM LED" signal also controls the LED in the centre of the board. The "duty cycle" and therefore the brightness of the LED changes with the setting of the selector switch Select 2, the frequency for this signal is always 100 kHz.
- For the "PWM" signal, the frequency can be set with the selector switch Select 1 and the duty cycle with the selector switch Select 2. The PWM duty cycle setting is suitable for frequencies up to 100 kHz, for higher frequencies the accuracy decreases or is lost altogether.
- The "PWM comb" signal shows the PWM signal with a square wave that is 128 times slower.

Select 1	"PWM" frequency [kHz]	Select 2	"PWM" (up to 100 kHz) and "PWM LED" duty cycle [%]
0	50	0	0.8
1	80	1	5.8
2	99.2	2	12.5
3	100	3	14.1
4	100.8	4	25.8
5	200	5	32.5
6	300	6	39.1
7	400	7	45.8
8	500	8	52.5
9	800	9	59.2
A	1000	A	65.8
B	1200	B	71.6
C	1500	C	78.3
D	2000	D	85.0
E	3000	E	91.7
F	4000	F	98.3

1.5 RF test track, RF OUT and Bode plot (sections 6 to 9)

There are two fast logic gates on the demo board which can be used to demonstrate a range of RF effects, the need for good probe-to-ground connection and proper termination. Jumper assignments are given in Chapter 4.

For Bode plot demonstrations, the "Bode" jumper must be set to J3 and the oscilloscope generator signal applied to "TERM IN" (J6).
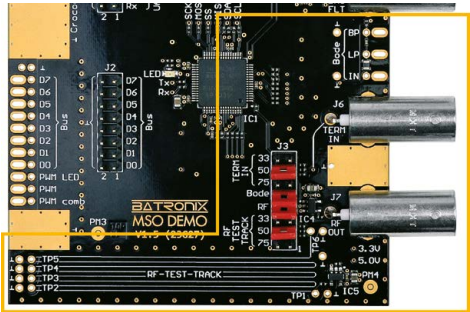
1.6 Function generator (section 10)

The demo board contains a function generator that can generate a number of different signal forms via a 10-bit DAC (digital-to-analog converter). The signals are amplified via operational amplifiers and put out low-pass filtered on one BNC socket and unfiltered on another BNC socket. 16 frequencies from 0.2 Hz to 10000 Hz can be set.

The Select 1 switch is used to select a signal mode (signal type and modification, if any):

Position	Select 1 Waveform	Select 2 Parameter
0	Sine	Frequency
1	Square	Frequency
2	Triangle	Frequency
3	Ramp	Frequency
4	DC	Voltage
5	Pulse with increasing amplitude	Frequency
6	Sweep 200 Hz - 10 kHz sine	Sweep time
7	Burst with 10 kHz sine	Number Periods
8	AM modulated sine (400 Hz / 6 kHz)	-
9	1 kHz PWM with adjustable duty cycle	Duty Cycle
A	1 kHz sine with noise	Noise component
B	10 kHz sine with pseudo-random errors	Error/cycle
C	10 kHz sine with peak after x periods	Events/s
D	1 kHz square wave with noise	Noise component
E	10 kHz square wave with pseudo-random errors	Error/Cycle
F	Various test signals, e.g. for trigger experiments. The selection can be seen in the table after next.	Signal

Depending on the signal mode, the Select 2 switch can be used to set the frequency, voltage, duty cycle or other parameter.

Select 2	Frequency [Hz]	Voltage [V]	Sweep time [ms]	Number Periods	Duty Cycle	Events [1/s]	Error rate Err/Wfms
0	0.2	0.2	1	1	1 %	0.1	~11 ppm
1	0.4	0.4	2	2	2 %	0.2	~22 ppm
2	1	0.6	3	3	5 %	0.5	~55 ppm
3	5	0.8	4	4	10 %	1	~110 ppm
4	10	1.0	5	5	20 %	2	~220 ppm
5	20	1.2	6	6	30 %	5	~550 ppm
6	50	1.4	7	7	40 %	10	~0.11 %
7	100	1.6	8	8	50 %	20	~0.22 %
8	200	1.8	9	9	50 %	50	~0.55 %
9	500	2.0	10	10	60 %	100	~1.1 %
A	800	2.2	15	11	70 %	200	~2.2 %
B	1000	2.4	20	12	80 %	500	~5.5 %
C	2000	2.6	25	13	90 %	1000	~11.5 %
D	5000	2.8	50	14	95 %	2000	~21 %
E	8000	3.0	125	15	98 %	4000	~55 %
F	10000	3.2	250	16	99 %	6000	~94 %

When the Select 1 switch is set to "F", the Select 2 switch can be used to output further special signals:

Select 2	Test signal
0	100 kHz clock
1	100 kHz clock with occasional drop-outs
2	50 kHz clock
3	50 kHz clock with occasional shorter or longer pulses / pauses
4	50 kHz clock (low level 1 V, high level 2 V)
5	Pulse with 160 µs rise and fall times
6	Pulse with varying rise and fall times (80 µs/160 µs/320 µs)
7	1 V DC voltage with noise and occasional peaks
8	1 V DC voltage with strong noise

2 Oscilloscope training
2.1 Safety and general information

- To avoid risks, follow all instructions and markings on the product. Refer to this manual for further information before connecting the product. This product complies with the EN61326-1:11-2022 standard.
- Do not operate the device if faults are suspected. If damage is suspected, have it inspected by qualified service personnel.
- Do not operate the device in wet or damp environments. Do not use the device in explosive atmospheres. Keep it clean and dry.
- Power the demo board via the USB cable (see 1.1).
- If probes are used, probe compensation should be performed beforehand. Refer to your oscilloscope manual for instructions.
- Be sure to connect the ground alligator clip on the probes to the ground of the demo board. Use the golden areas with the ground symbol or the outer area of unused BNC sockets.
- In many cases, oscilloscope probes can be fitted with clamping hooks. They are useful here because they can be easily hooked into the slotted holes on the edges of the board or on the leads of the BNC sockets.
- If you are using probes with attenuation (e.g. 10:1), make sure you set the same attenuation factor on your oscilloscope for the corresponding channel.
- Please ensure that you use BNC cables with a characteristic impedance of 50 Ω.
- If you have a "Default" or "Preset" button on your oscilloscope, you can use it to reset certain settings from previous measurements and continue with the default settings.
- The naming of oscilloscope functions can vary from manufacturer to manufacturer. We use a description that is as manufacturer independent as possible. If necessary, consult your oscilloscope manual.
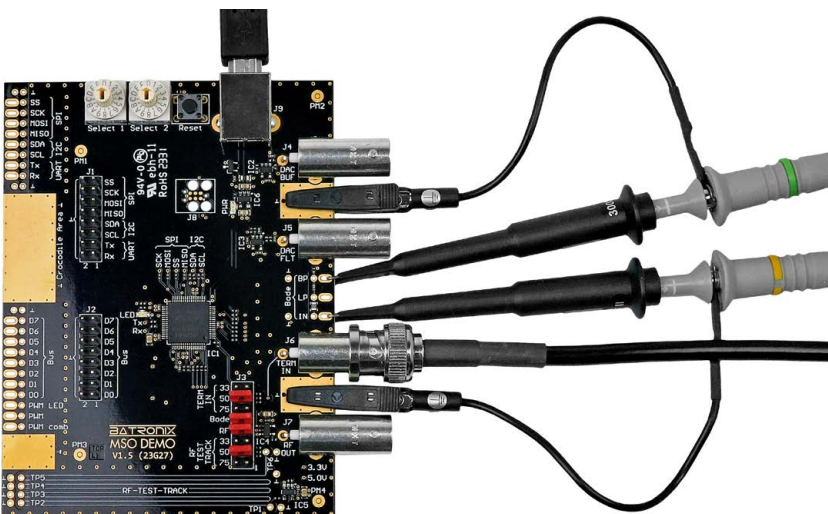
Demoboard with connected probes and BNC cable for a Bode plot measurement

2.2 Introduction horizontal and vertical system

In today's digital oscilloscopes, an analog-to-digital converter (ADC) converts analog signal voltages into digital quantisation levels. Depending on the oscilloscope, ADC resolutions of 8, 10 and 12 bits are common, converting the analog signal into 256, 1024 or 4096 quantisation steps. These steps are distributed over the entire input voltage range of the ADC and are displayed on the screen over the entire vertical display area.

Oscilloscopes can amplify or attenuate signal voltages in their front end (the input range up to the ADCs) to match different voltage levels between a few millivolts and hundreds of volts to the input voltage range of the built-in ADC.

To make the most of the ADC's limited accuracy, you should scale signals to utilize a large portion of the vertical area (and thus a large portion of the ADC's input voltage range).

An offset voltage can also be applied to the signal in the oscilloscope front end, e.g. to shift negative signal voltages into the input voltage range of the ADC. On the screen, the measurement signal moves vertically when the offset voltage is set.

The horizontal axis represents the time course. The time scale can also be set over a wide range. Depending on the oscilloscope, anything from a few picoseconds to thousands of seconds can be displayed across the width of the screen.

The display is usually divided into a grid of "divisions" (divs for short). Depending on the oscilloscope, there are usually 8 or 10 divs vertically and 12 to 16 divs horizontally. The vertical and horizontal scales refer to these divs, so the setting is made as, for example, 2 V/div and 1 ms/div.
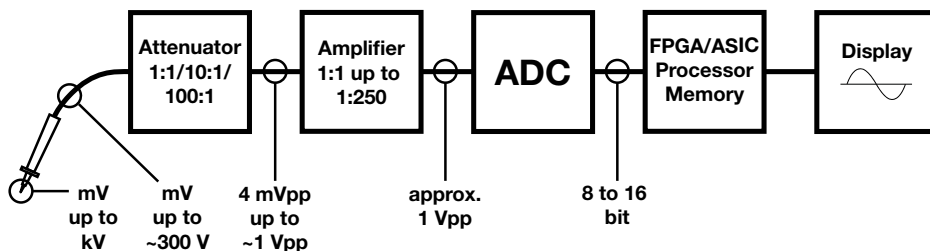

Illustration: From the probe through the oscilloscope front end to the display

2.3 Initial experiments with signal scaling and positioning

Please follow the steps below:
- Set the demo board selector switch Select 1 to "0" and Select 2 to "F".
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable. If this channel is not already switched on on your oscilloscope, switch it on now.
- Depending on your oscilloscope's settings, you may see the signal, parts of the signal or nothing at all. If the signal or parts of it are already visible, they may still be running through the screen or flickering.
- The first thing to do is to adjust the horizontal and vertical settings of your oscilloscope so that the signal is clearly visible and stable on the screen. With a little practice, you will be able to display new and unknown signals on the screen quickly and easily.
- For this first signal, set the horizontal scaling to 50 µs/div and the vertical scaling to 1 V/div.
- Set the trigger to edge, the trigger channel to the channel being used and the trigger level to the centre of the signal.
- The signal should now appear stable on the screen.
- Now some fine tuning follows: Use the vertical scaling and the offset setting of your oscilloscope to display the signal on as large a vertical area of the screen as possible. The signal should not touch the edges of the screen or display vertically.
- Adjust the horizontal scaling so that you can see one or two complete periods of the signal on the screen.
- Now it should look similar to the following screenshot:

Screenshot: A 10 kHz sinus signal, scaled to fit the screen

2.4 Trigger
2.4.1 Introduction to the trigger system

In "run mode" the oscilloscope continuously measures the input signal and displays it on the screen. With the trigger system, signals from successive waveforms are displayed at the same horizontal position (of the time axis). The signal thus "stands" aligned on the screen with a fixed time reference and can be easily viewed and evaluated.

The trigger system of an oscilloscope allows various settings of the trigger condition. When the trigger condition is detected, the measured signal is displayed on the screen in the appropriate horizontal alignment. The horizontal position (at which the trigger condition was detected) can be moved to the left and right, even far outside the screen.

In order to be able to view and evaluate the signal before the trigger system is correctly set, there is an auto trigger option. When this is activated, signals are displayed several times per second, even without triggering. These usually have no time reference and appear at different horizontal positions. They do, however, help to estimate and find the correct trigger setting.

Modern oscilloscopes offer a whole range of special triggers for triggering on different signal shapes. These include edge, pulse width, interval, dropout, slope, window, run and pattern triggers, as well as other triggers such as those for triggering on decoded data from serial interfaces.

2.4.2 Edge trigger

Please follow the steps below:
 • Carry out the steps in chapter 2.3 if you have not already done so.
 • Adjust the trigger level and see which part of the signal is displayed at the trigger point.
 • Adjust the horizontal position and observe the temporal (horizontal) shift of the display.
 • Press the "Stop" or "Run/Stop" button to stop further measurements and continue to display the measured signal.
 • Press the "Single" button to make a single measurement.
 • The signal stopped in this way can be moved with the vertical offset setting and the horizontal position on the screen.
 • Press the "Run" or "Run/Stop" button to restart the continuous measurements.

2.4.3 Pulse width trigger

The pulse width trigger triggers on positive or negative pulses that are shorter or longer than a predefined time interval. This can be used, for example, to trigger on periodic or repeating patterns.

Depending on the oscilloscope manufacturer, this trigger is also called pulse trigger or width trigger.

Please follow the steps below:
• Set Select 1 to "7" and Select 2 to "B" (burst of 12 pulses).
• Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
• Set the vertical scale to 500 mV/div and the horizontal scale to 500 µs/div.
• If you trigger this signal with the edge trigger, different pulses will cause trigger events and you will see an alternating non-stationary display.
• Enter the "Trigger" menu and set the trigger type to pulse width trigger.
• Set the trigger level slightly below the "pause voltage", e.g. 0.8 V, and the polarity of the trigger to positive. The pause time in the signal is about 800 µs, so set the trigger time to > 400 µs.
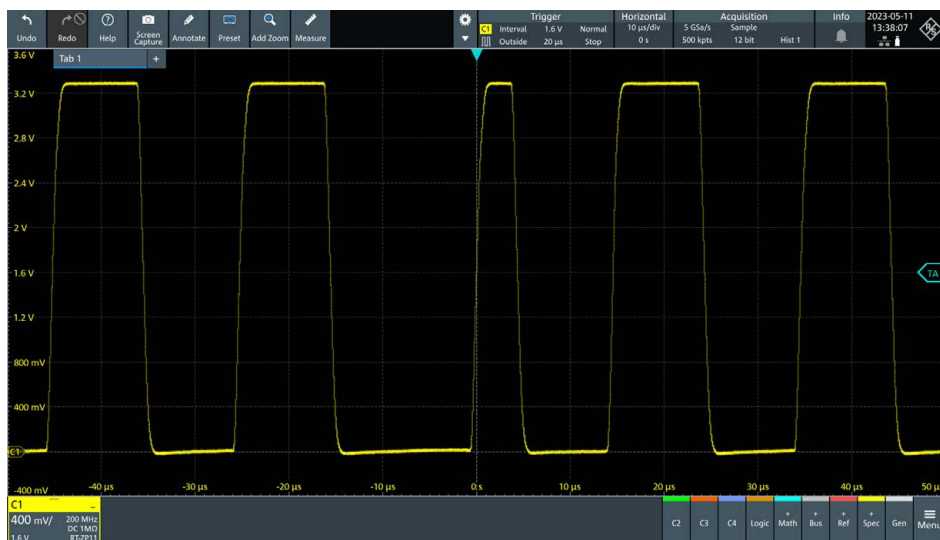

Screenshot: Triggering with the pulse width trigger on a burst

2.4.4 Interval trigger

The interval trigger is similar to the pulse width trigger, but triggers on period times instead of pulse times. It can be triggered on the overshoot or undershoot of a time or on a time range.

This makes it possible, for example, to trigger on temporal anomalies of a clock line. Please follow the steps below:

- Set Select 1 to "F" and Select 2 to "3" (50 kHz clock with occasional shorter or longer pulses/pauses).
- Measure the "DAC BUF" signal (J4) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Set the horizontal scale of your oscilloscope to 10 µs/div. Set the vertical system so that you can see as much of the signal as possible on the screen.
- First trigger the square wave signal with the edge trigger. The square wave signal will usually look correct. You may (depending on your oscilloscope) see occasional and brief "flashes" of incorrect displays.
- Now use the interval trigger on your oscilloscope. Set the trigger level to 1.6 V and the interval duration to < 18 µs or > 22 µs. You will now see a number of erroneous signals in the mass of correct signals, which were not or hardly noticeable with the simple edge trigger.
- Switch from auto trigger mode to normal trigger mode so that only the erroneous waveforms are displayed. Switch back to auto trigger mode for subsequent measurements.

Screenshot: Triggering with the interval trigger on deviating period times

2.4.5 Dropout / timeout trigger

The dropout trigger allows triggering for an adjustable time without relevant edges.

This can be used, for example, to trigger at the end of a data transmission.

Please follow the steps below:
 • Set Select 1 to "0" and Select 2 to "8".
 • Measure the "I2C SCL" signal (section 3) with a probe.
 • First set the vertical scale to 500 mV/div and the horizontal scale to 200 µs/div.
 • First trigger the signal with the edge trigger.
 • Now use your oscilloscope's dropout trigger (also called timeout trigger, depending on the oscilloscope). Set the trigger level to 1.6 V and the trigger time to 100 µs. If your oscilloscope has a "range" setting, set it to "stays high".

Screenshot: Triggering with the dropout or timeout trigger on the end of a data transmission
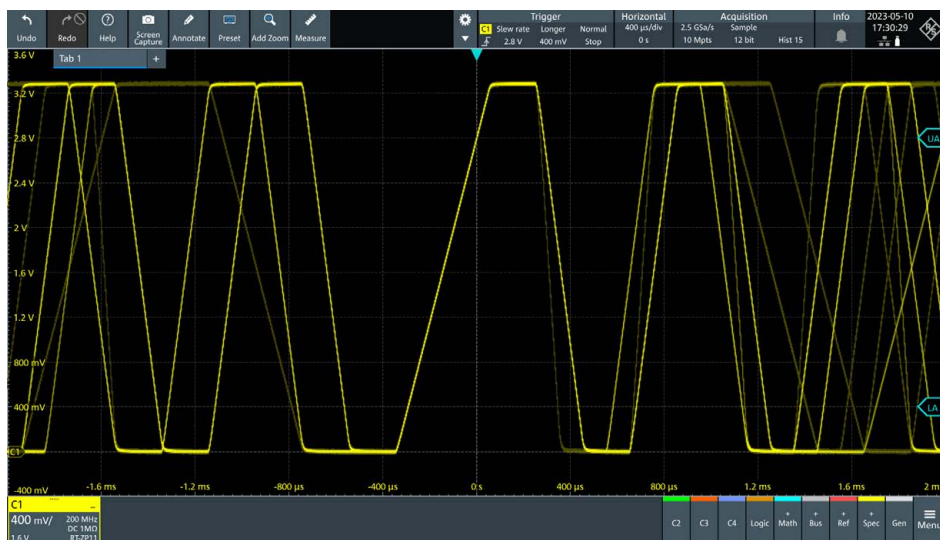
2.4.6 Slope / slew rate / rise time trigger

The slope trigger allows triggering on an adjustable slope of the signal. Two voltage levels and a time span can be set in general.

This can be used, for example, to trigger on unexpectedly fast or slow signal slopes.

Please follow the steps below:
- Set Select 1 to "F" and Select 2 to "6".
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Adjust the vertical and horizontal system so that three to six waveforms are displayed almost full screen.
- First trigger the signal with the edge trigger. The signal will normally look correct. Occasionally (depending on the oscilloscope) you may see brief "flashes" of incorrect waveforms.
- Now use the oscilloscope's slope trigger (also called slew rate or rise time, depending on the oscilloscope). Set the trigger levels to 300 mV and 3 V and the trigger rise time to > 200 µs.
- Switch from auto trigger mode to normal trigger mode so that only the faulty waveforms are displayed. It should now resemble the following image.
- You can also set the slope trigger to "Outside" and the time range to 140 µs to 180 µs as well as rising and falling edges, if supported by your oscilloscope. This should enable triggering on all errors of this test signal.
- For subsequent measurements, set the auto trigger mode again.

Screenshot: Triggering with the slope / slew rate or rise time trigger to rise times > 200 µs

2.4.7 Window trigger

The window trigger is triggered by signals entering or leaving a voltage range. Depending on the oscilloscope, a time can be set for the signal to remain in or out of the voltage range in order to trigger.

This allows, for example, a power supply to be monitored for positive and negative voltage spikes simultaneously.

Please follow the steps below:
- Set Select 1 to "F" and Select 2 to "7" (1 V DC voltage with noise and occasional spikes).
- Measure the signal "DAC FLT" (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Set the vertical scale to 200 mV/div and the horizontal scale to 100 μs/div. Set the vertical position (channel offset) to the bottom of the display area so that you can see the signal completely.
- Now use the oscilloscope's window trigger. Set the two trigger levels so that one is above and one is below the DC voltage (e.g. 0.9 V and 1.1 V).

Screenshot: Triggering with the window trigger on positive and negative peaks of a power supply line

2.4.8 Runt trigger

The runt trigger enables you to detect signals that reach a first threshold but do not exceed a second threshold.

This can be used, for example, to trigger on incorrect (too low) signal level rises.

Please follow the steps below:
- Set Select 1 to "E" and Select 2 to "9" (10 kHz square wave with pseudo-random errors).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Adjust the vertical, horizontal and trigger system so that three to six signal sequences are displayed almost full screen.
- First trigger the square wave signal with the edge trigger. The square wave signal will usually look correct. You may (depending on the oscilloscope) see occasional and brief "flashes" of incorrect displays.
- Now configure and activate the runt trigger. Set the trigger low level to 300 mV and the trigger high level to 3 V. Now you will see a number of erroneous signals that were not or hardly noticeable with the normal edge trigger in the mass of correct signals.
- Switch from auto trigger mode to normal trigger mode to see only the faulty signals.
- Adjust the error rate by moving the Select 2 switch.
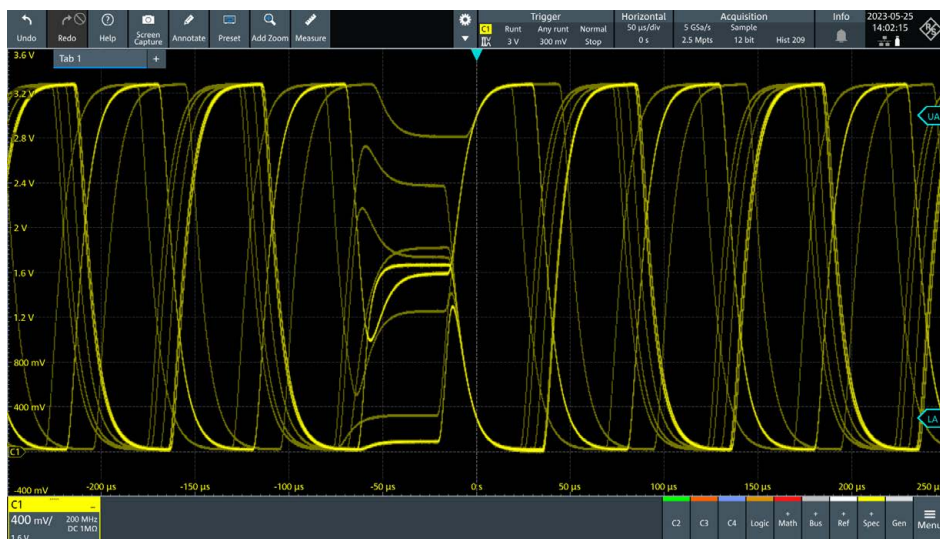
Screenshot: Triggering with the runt trigger on signals with insufficient amplitude

2.5 Input coupling (AC/DC)

The oscilloscope inputs can capture signals with their DC component or only the AC component. There is a capacitor in the input path of the oscilloscope for this purpose, which is bridged in the case of DC coupling.

With AC coupling enabled, this capacitor forms a high-pass filter that attenuate signals in the hertz range and below. The setting can be made individually for each channel and can be found in the channel settings accordingly.

With AC coupling, for example, the noise on a DC supply voltage can be measured. The DC component is blocked, the vertical scaling can be set to a few mV/div and the noise can be displayed and measured as large as possible.

Please follow the steps below:
 • Set Select 1 to "4" and Select 2 to "9" (approx. 2 V DC signal).
 • If possible, measure the "DAC BUF" signal (J4) using a BNC cable or probe with a 1:1 divider to keep the inherent noise of your oscilloscope low.
 • Set the vertical scale to 1 V/div, the vertical offset to 0 V and the horizontal scale to 10 µs/div.
 • After making the above settings, switch the channel you are using to AC coupling. The signal will "move" to the vertical position of the channel (GND line of the channel).
 • Now set the vertical scale as fine as possible (1 to 10 mV/div). The noise on the supply voltage becomes clearly visible.
 • Reset the DC coupling for subsequent measurements.
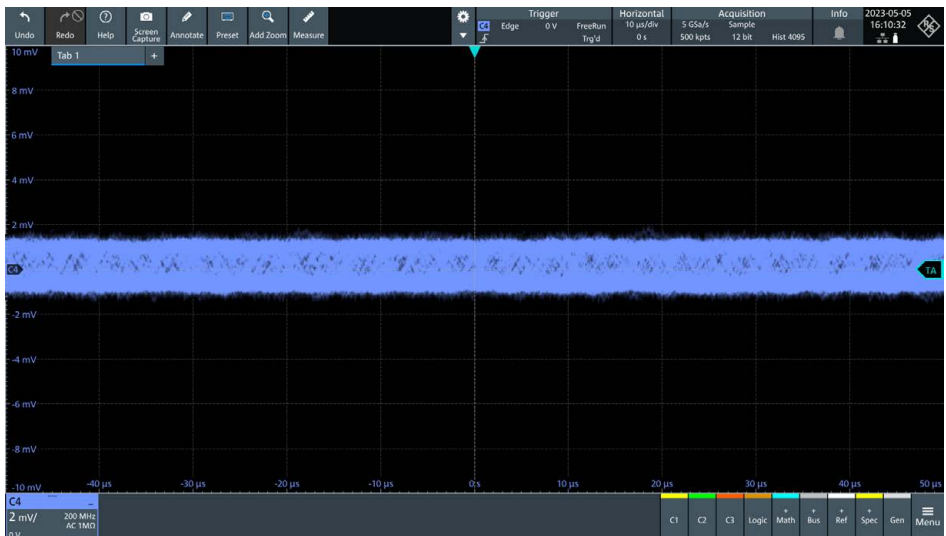
Screenshot: Measuring the noise of a 2 V DC voltage with AC coupling

2.6 Roll mode

The roll mode allows slow signals to be displayed continuously without pauses. Depending on the oscilloscope, the signal usually enters the display area on the right-hand side of the screen and then moves to the left. There is no triggering in this mode.

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "2" (1 Hz sine wave).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Set the horizontal scale of the oscilloscope to 500 ms/div. Set the vertical system so that you can see as much of the signal as possible on the screen.
- Activate your oscilloscope's roll mode. Depending on the oscilloscope, this is done automatically at certain horizontal scales, or can be switched on in the "Horizontal" or "Acquire" menu, or via a separate "Roll" key.
- Inspect the signal trace for a few seconds.
- Turn the Select 1 switch step by step to "3" and look at the different signal shapes.
- Set different values on the Select 2 switch and see the effect on the "speed" of the signal.

Screenshot: Roll mode with changing signal curves, achieved by switching Select 1 during measurement

2.7 Acquisition system

Oscilloscopes offer different sampling modes that can be set in the "Acquire" menu. In addition to "Normal" (or "Sampling"), there are "Average", "Peak Detect" and other options depending on the oscilloscope.

2.7.1 Average
The average mode superimposes an adjustable number of measurement curves and averages their course. This measurement mode can lead to strong noise suppression and more precise measurement. Depending on the oscilloscope, it also increases the final measurement resolution (the number of quantisation steps). The prerequisite for using the average mode is a suitable trigger setting and a recurring signal curve.

Please follow the steps below:
- Set Select 1 to "D" and Select 2 to "B" (noisy square wave).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Adjust the vertical and horizontal system and the trigger so that you see one or two full periods on the screen.
- Now set the acquisition mode to "Average" and the average number to 16 in the "Acquire" or "Acquisition" menu of your oscilloscope.
- There should now be much less noise on the screen.
- Switch back and forth between Average and Normal and observe the effect.
- Try other average numbers such as 4 and 256 and see the effect.
- Set the acquisition mode back to Normal or Sample for the following measurements.
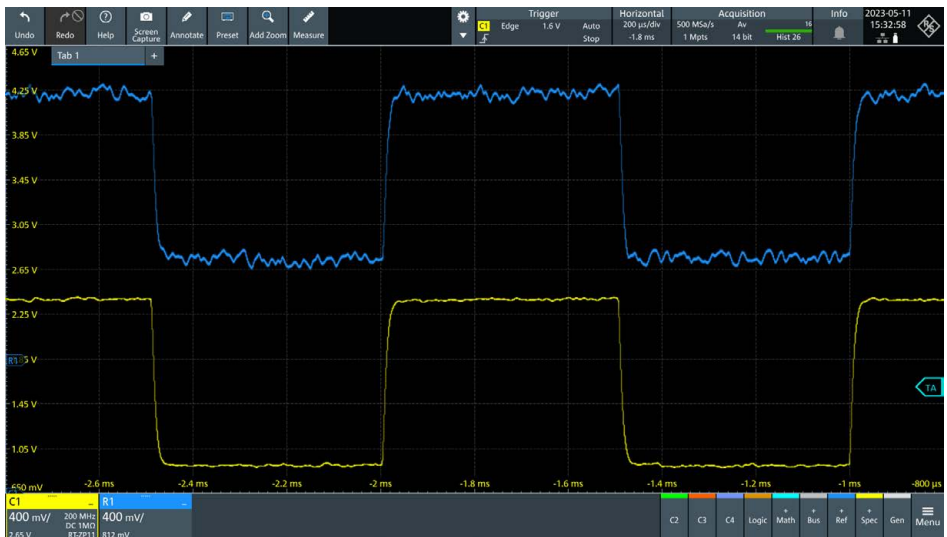
Screenshot: 1 kHz square wave with strong noise (blue reference: normal mode, yellow signal: average mode)

2.7.2 Peak detect

Depending on the horizontal scaling and oscilloscope sample rate settings, many samples will be recorded per screen column. However, not all samples can or shall always be stored and displayed at the maximum data rate. In these cases the data rate is reduced (downsampling).

In the "Peak Detect" acquisition mode, downsampling is performed by reducing to minimum and maximum values so that anomalies can be detected and short-term problem areas can be made more visible despite the lower data rate.

Please follow the steps below:
- Set Select 1 to "1" and Select 2 to "8" (200 Hz square wave).
- Measure the "DAC BUF" signal (J4) with one channel of your oscilloscope. If available, use a BNC cable or a probe with a 1:1 divider.
- Adjust the vertical and horizontal system and the trigger so that you see one or two full periods on the screen.
- Set the memory depth to the lowest value. For some oscilloscopes it is necessary to set the memory depth to the minimum in order to see the effect clearly based on the demo board signals. Depending on the oscilloscope manufacturer, the memory depth setting can usually be found in the "Acquire" menu.
- Set the "Acquire" mode to Peak Detect.
- The waveform should now be wider, depending on the oscilloscope. You may even see a "double line" as in the screenshot below. The top line shows the maximum values, the bottom line shows the minimum values.
- Switch back and forth between Normal and Peak Detect.
- For subsequent measurements, set the "Acquire" mode back to Normal and remove the memory limitation.
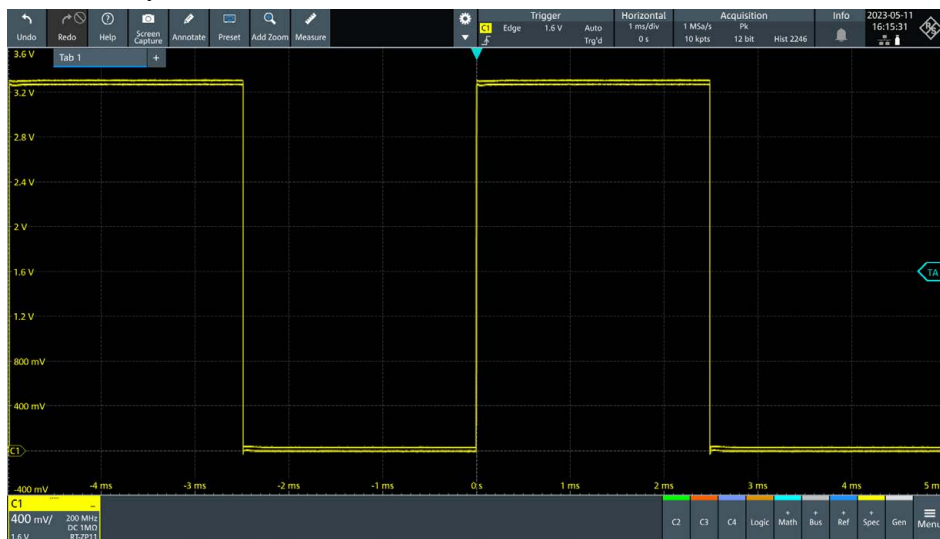
Screenshot: 200 Hz square wave in peak detect mode

2.8 Cursor

The cursors can be used to measure voltages, voltage differences and time intervals on the screen.

Please follow the steps below:
- Set Select 1 to "5" and Select 2 to "8" (pulse sequence with different high pulses).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Set the trigger to "Edge" and the trigger level to 2.8 V so that it triggers on the highest pulse and you see a still image.
- Adjust the vertical and horizontal system so that you see as full a cycle as possible on the screen.
- Enable the vertical cursors and use them to measure the voltage level of the medium-high pulse.
- Enable the horizontal cursors and use them to measure the width of a pulse.

Try out the other options your oscilloscope provides in the cursor menu.

Screenshot: Measuring a pulse with the cursor function

2.9 Measurements

The measurements allow a variety of automatic measurements to be made on the signal waveform. In addition to the numerous voltage and time measurements, combined measurements such as the rise time of a signal and measurements between signals (e.g. delay) are also possible.

Depending on the oscilloscope, several measurements can be displayed simultaneously. Further settings can be applied via the measurement menu, such as the display of measurement statistics (min, max, average, etc.). Measurement results are usually determined over the range visible on the screen.

Please follow the steps below:
- Set Select 1 to "1" and Select 2 to "F" (10 kHz square wave).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Adjust the vertical, horizontal and trigger system so that one or two waveforms are displayed almost full screen.
- Use the measurements to measure voltage amplitude (Vamp), peak-to-peak voltage (Vpp), frequency and rise time.
- Turn on the measurement statistics and get an overview of this evaluation of anomalies, averages and more.

Screenshot: Automatic measurement of various signal parameters

2.10 Persistence

The persistence setting allows oscilloscopes to "afterglow" a waveform for an adjustable time, or even permanently, while more recent waveforms are displayed. This setting is useful for making infrequent events/errors better visible.

Please follow the steps below:
- Set Select 1 to "C" and Select 2 to "6" (10 kHz sine wave with about ten noise peaks per second).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Adjust the vertical, horizontal and trigger system so that four to eight waveforms are displayed almost full screen.
- If necessary, reduce or increase Select 2, depending on the oscilloscope, until only occasional spurious peaks are visible.
- Set the persistence to one second. This setting is usually found in the "Display" menu, depending on the oscilloscope manufacturer.
- Any spikes that appear should be visible for about one second before disappearing.
- Now set the persistence to infinite. Once signals have been recorded, they will remain visible until a new measurement is started or the display is cleared by pressing the "Clear" button.
- Switch off the persistence display for subsequent measurements.
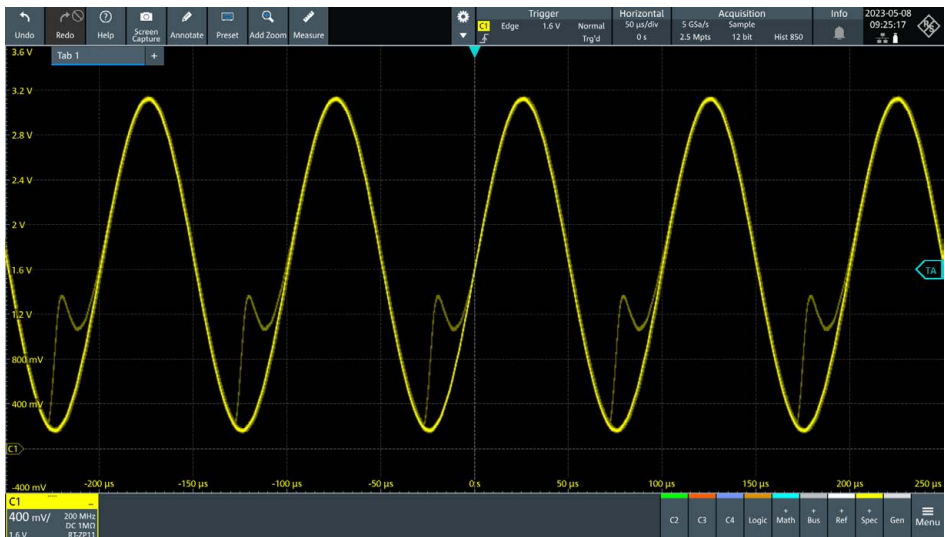
Screenshot: Better visibility of rare signal events by setting the display persistence

2.11 Mask tests

The mask test function allows a signal to be measured continuously and compared with an "ideal" waveform. If there is a deviation, various actions can be taken, such as stopping the measurement and outputting an audible signal or a control signal on the oscilloscope's aux output. The number of good and bad passes (pass/fail) can also be counted and an error rate calculated and displayed.

Please follow the steps below:
- Set Select 1 to "B" and Select 2 to "C" (10 kHz sine wave with occasional errors).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Adjust the vertical, horizontal and trigger system so that one or two waveforms are displayed almost full screen.
- Take a single measurement. Repeat if you have recorded an incorrect sine wave.
- Use the measurement as a pattern. Use the "Create Mask" function in the "Mask Test" menu of your oscilloscope.
- Start the measurement and mask test and watch the pass/fail counter.
- Adjust the error frequency using different settings on Select 2 on the demo board.
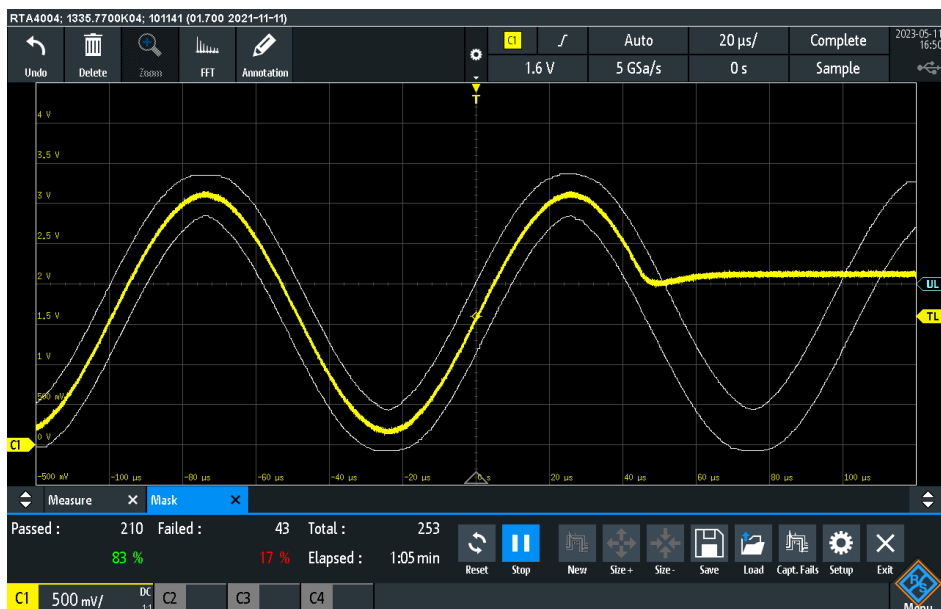
Screenshot: Mask test with a faulty signal and visible mask area

2.12 References

References can be used to display stored signals or mathematical operations in subsequent measurements without the need for the signal to be present on the oscilloscope. References can be used, for example, to make adjustments to the circuit under test and to check the resulting changes by comparing the signal before and after. Depending on the oscilloscope, a certain number of reference signals can be stored and displayed.

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "8" (sine signal).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Adjust the vertical, horizontal and trigger system so that one or two waveforms are displayed almost full screen.
- Create a reference of the waveform.
- Set the Select 1 switch from "0" to "1" (square wave signal).
- You can now see the previously measured waveform as a reference and the currently measured waveform at the same time.
- You can also use the vertical offset to move the reference or current signal slightly up or down. Now the two waveforms are no longer exactly on top of each other. This makes it easier to compare similar waveforms.
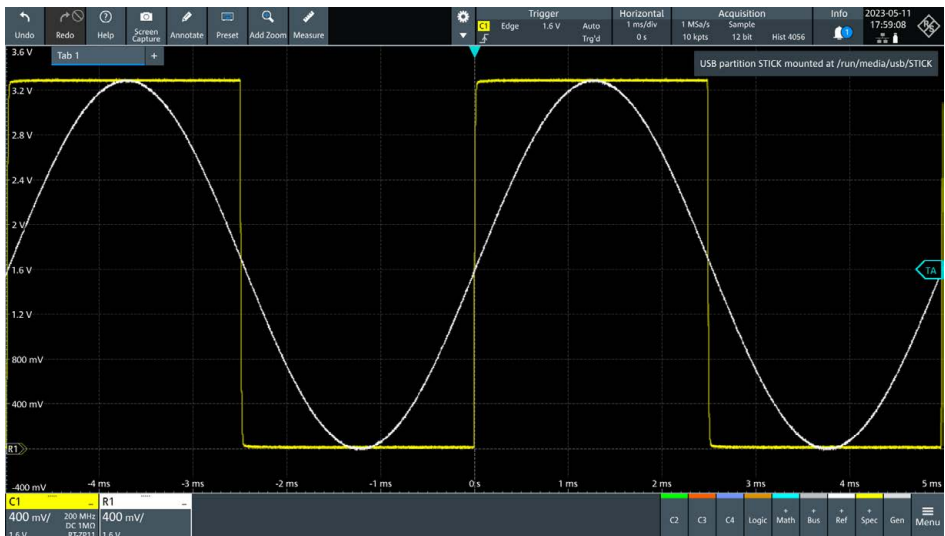
Screenshot: The reference function allows to compare a previously measured signal with a current signal.

2.13 Mathematics

Mathematical functions can be used to add, subtract, multiply or divide signals from multiple channels. Depending on the oscilloscope, even complex formulas are possible. The curve resulting from the math function is displayed on the screen.

For example, if the supply voltage of a circuit is measured on one channel and the current consumption is measured on a second channel using a current probe, the power consumption over time can be calculated by multiplying both channels and displayed on the oscilloscope screen.

Please follow the steps below:
• Set Select 1 to "F" and Select 2 to "4".
• Measure the "DAC BUF" (J4) and "DAC FLT" (J5) signals with a probe or connect each to an oscilloscope channel using a BNC cable.
• Set the trigger to "Edge", trigger on the "DAC BUF" signal (J4) and set the trigger level to 1.5 V.
• Adjust the vertical, horizontal and trigger system so that some periods are displayed in the lower half of the screen.
• Switch on a math function that multiplies both channels. If one channel measured voltage and the other current, you would see the power curve (screenshot: red signal curve).
• If your oscilloscope supports it, you can configure another math channel that forms the integral over both channels multiplied over time. As a result you will see the energy consumption within the active time range (screenshot: yellow signal curve).
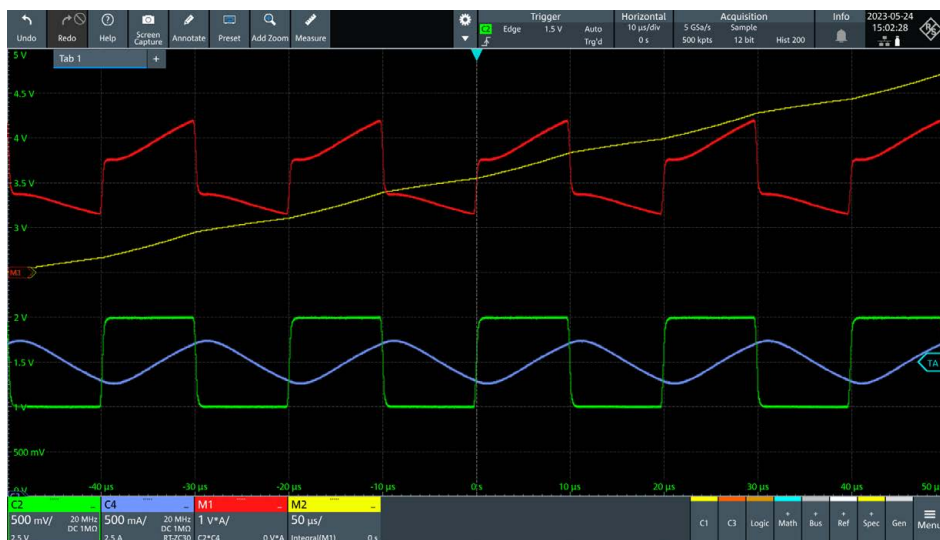
Screenshot: Using math functions to calculate power and energy consumption

2.14 FFT / Spectrum analysis

With the help of the fast Fourier transform (FFT) signals can be separated into and display based on their spectral components, leading to a significant gain in analysis capabilities. This functionality is ideal for identifying the frequency components in noise and deducing possible causes of the noise. FFT analysis can also be used to determine the quality of an amplifier and signal interference.

2.14.1 Harmonics of a square wave

In terms of signal theory, (ideal) square wave signals are made up of individual sinusoidal harmonics of multiples of the fundamental frequency, the odd harmonics. Please follow the steps below:

- Set Select 1 to "1" and Select 2 to "F" (square wave).
- Measure the "DAC BUF" signal (J4) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Use the vertical scaling and offset setting of your oscilloscope to display the signal on as large a vertical display area as possible.
- Set the time scale so that you can see many full periods of the signal on the screen, e.g. 500 µs/div.
- Turn on the FFT analysis. Depending on the manufacturer of the oscilloscope, you will find this under "FFT", "Spectrum" or in the "Math" menu.
- Set the FFT window function to "Hann" or "Hanning".
- Set the FFT display range to a "centre" frequency of 30 kHz and the span to 60 kHz (or 5 kHz/div).
- You should now be able to see the fundamental peak at 10 kHz and the harmonic peaks at 30 kHz and 50 kHz.
- Switch on the peak table display and set the threshold to -20 dBm.

Screenshot: FFT analysis of a square wave signal with clearly visible harmonics

2.14.2 Signal purity of a sine signal

In DAC or amplifier circuits, it is often important to generate or amplify a signal with as little distortion as possible. Coarse distortions, e.g. of a sinusoidal signal, can be easily seen on the screen in the time domain. Finer distortions are not visible to the eye, but can be measured and displayed very well in the frequency domain.

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "F" (sine wave).
- Measure the "DAC FLT" signal (J5) with a probe or connect it to an oscilloscope channel using a BNC cable.
- Use the vertical scaling and offset setting of your oscilloscope to display the signal on as large a vertical display area as possible.
- Set the time deflection so that you can see many full periods of the signal on the screen, e.g. 500 µs/div.
- Turn on the FFT analysis. Depending on the manufacturer of the oscilloscope, you will find this under "FFT", "Spectrum" or in the "Math" menu.
- Set the FFT window function to "Hann" or "Hanning".
- Set the FFT display range to a "center" frequency of 30 kHz and the "span" to 60 kHz (or 5 kHz/div).
- You should now be able to see the peak of the fundamental at 10 kHz, the third harmonic at 30 kHz and possibly the fifth harmonic at 50 kHz.
- The difference between the two levels in the screenshot below is 43.5 dB (+13.05 dBm minus -30.49 dBm).
- The greater the difference between the two values is, the "purer" is the sine wave signal. The 43.5 dB measured here corresponds to a factor of approximately 150 between the fundamental and the third harmonic.

Screenshot: FFT analysis of a sinusoidal signal with clear peaks at 30 kHz and 50 kHz (HD3, HD5)

2.14.3 Noise analysis of a DC voltage

FFT analysis is also ideal for identifying the dominant frequency components in a signal's noise, making it easier to find the source of the noise.

Please follow the steps below:
- Set Select 1 to "4" and Select 2 to "F" (3.2 V DC).
- If possible, measure the "DAC BUF" signal (J4) using a BNC cable or a probe with a 1:1 divider to keep the inherent noise of your oscilloscope low.
- Set the AC coupling for the channel.
- Use the vertical scaling and offset settings of your oscilloscope to display the signal on as large a vertical area of the screen as possible.
- Turn on the FFT analysis. Depending on the manufacturer of your oscilloscope, you will find this under "FFT", "Spectrum" or in the "Math" menu.
- Set the FFT window function to "Hann" or "Hanning".
- Set the FFT display range to a "center" frequency of 200 kHz and the "span" to 400 kHz (or 50 kHz/div).
- Enable the 20 MHz input filter of the channel in use (individual channel settings).
- You should see one or more peaks around 200 kHz (the sampling frequency of the DAC).
- Now connect the filtered "DAC FLT" signal (J5) and compare the amplitude of the noise floor.
- For subsequent measurements, switch the channel back to DC coupling and disable the 20 MHz input filter.

Screenshot: FFT noise analysis of a DC voltage from the function generator of the demo board

2.15 Frequency response analysis (FRA / Bode plot)

This function can be used to measure the frequency response of a system. A Bode plot shows the amplitude and phase response of a system as a function of (input) frequency.

First please ensure that your oscilloscope provides this analysis feature.

If this is the case, please follow the steps below:
- Insert the jumper at "Bode" as well as at the upper "50" (both J3).
- Connect the function generator output of your oscilloscope to the "TERM IN" input (J6) of the demo board.
- Place a probe on each of the "IN" and "BP" signals.
- Configure the Frequency Response Analysis (FRA) / Bode plot function on your oscilloscope. You need to set the input channel, the measurement channel and the frequency range. For the filter on the demo board, a start frequency of 200 Hz and a stop frequency of 10 MHz is suitable.
- Start the measurement and, if necessary, adjust the vertical scaling of the Bode plot so that you can see the waveform clearly.
- After the measurement run, which may take some time, you should see a curve similar to the one in the following screenshot. The blue line in the following screenshot shows the amplitude response, the red line corresponds the phase response of the bandpass filter on the demo board.

Screenshot: The Bode plot of the demonstration filter on the demo board

2.16 XY mode

The XY diagram shows the frequency, phase and amplitude relationships of two signals. As in the normal mode (YT mode), the vertical displacement is controlled by one signal. However, in the XY mode, the horizontal displacement is controlled by a second signal instead of time.

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "D" (5 kHz sine wave).
- Connect the BNC connectors "DAC FLT" (J5) and "TERM IN" (J6) to each other using a BNC cable.
- Insert the jumper at "Bode" and at the upper "50" (both J3).
- Measure the "DAC FLT" signal (J5) with a probe (e.g. clamping hook) on the right-angled wire going from the PCB to the BNC socket.
- Measure the "BP" signal (in the Bode area) with a second probe.
- Set the vertical scale of both channels to 200 mV/div and the horizontal scale to 1 ms/div.
- Switch on the oscilloscope's XY mode and set up the two channels for XY display.
- Change the frequency of the sine wave using the Select 2 switch and observe the effect on the XY display. The phase shift to the first channel and the amplitude of the BP (bandpass) output change significantly as the frequency is changed.

Screenshot: XY diagram of two signals

3 Serial and parallel digital interfaces

In many electronic circuits, components communicate with each other via serial or parallel interfaces. Depending on the interface, digital data and sometimes clock signals are in most cases transmitted over one to four lines. On the demo board, there are several standard interfaces (SPI, I2C, UART) with bidirectional communication, as well as a 7-bit digital bus that also supports CAN, CAN-FD and LIN communication.

The standard interfaces can be configured in a variety of ways using the Select 1 and Select 2 switches.

The Select 1 switch is used to select various interface settings (LSB/MSB first, number of bits / parity, etc.) and data patterns. For example, the transmitter sends the name of an EU country and the receiver responds with the capital. The selection and order of countries is based on the number of inhabitants.

The Select 2 switch determines the speed of the interfaces (bit or baud rate).
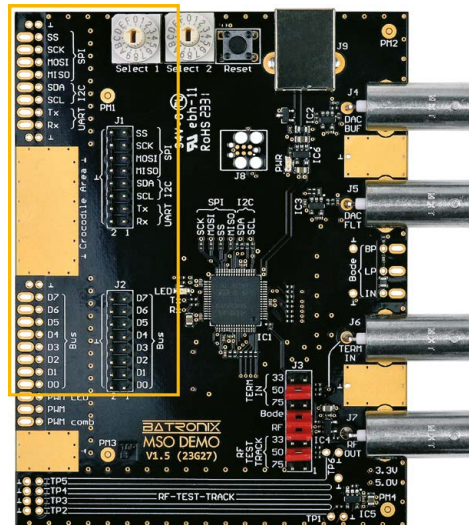
3.1 I²C interface (Inter-Integrated Circuit)

The I²C interface uses a master-slave architecture where a master device controls communication and can communicate with multiple slave devices.

I²C only needs two lines:
- SDA (Serial Data): Bidirectional data line through which both the master and the slaves can send and receive data.
- SCL (Serial Clock): Clock line for synchronising data, generated by the master.

The Select 1 switch is used to select different communications and Select 2 to select the speed:

Select 1	Communication
0	Listing of some EU countries and their capitals
1 to D	Country / Capital
E	"Batronix" with occasional errors / Counter
F	Joke

Select 2	0	1	2	3	4	5	6	7
Bit rate [kbit/s]	2	2.5	3	4	5	8	10	15

Select 2	8	9	A	B	C	D	E	F
Bit rate [kbit/s]	20	30	50	100	150	200	300	400

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "8".
- Measure the I²C signals "SDA" and "SCL" with one probe each.
- First set the trigger mode to "Edge" on one of the two channels and move the trigger level to the centre of the signal amplitude.
- Set up the vertical and horizontal systems of your oscilloscope so that the channels use a large part of the vertical screen size. The horizontal setting should be such that a data word is completely visible. The individual data words are separated by pauses, making them easy to identify.
- Use the "Single" function of your oscilloscope to display a single record.
- Set up your oscilloscope's decoder. In addition to selecting the interface and assigning the corresponding channels used to SCL and SDA, you should also pay attention to the threshold setting. Set it somewhere in the middle of the signal level, e.g. 1.6 V. Set the display data format to "ASCII".
- Start a new single recording.
- If you see the name of an EU country or capital, you have done everything correctly. If you do not see any decoding information, or if you see unexpected or incorrect decoding information, check the above points again and, if necessary, compare the selected decoder settings with your oscilloscope's manual.

3.1.1 Decoder table

- Apply the settings as described in 3.1.
- Set Select 1 to "F" and Select 2 to "8".
- Activate the decoder table.
- Set the decoder display to ASCII.
- Zoom out further in time by setting the horizontal setting to e.g. 10 ms/div.
- Read the words/sentences that the transmitter (Write) and receiver (Read) send to each other.

3.1.2 Protocol trigger

Once you have set up a decoder in the appropriate way, you can also trigger on data words and on events.

Please follow the steps below:
- Apply the settings as described in 3.1.
- Set Select 1 to "0" and Select 2 to "8".
- Set the trigger to "Serial" / "Decode" or "I²C".
- Set the trigger to the ASCII character "B" or the hex code 42h.
- Start the measurement in progress ("Run" key).
- Switch from auto trigger mode to normal trigger mode.
- Triggering should now take place on the following cities or countries: "Berlin", "Bucharest", "Belgium", "Brussels" and "Budapest".
- Try different decoder trigger conditions. Depending on your oscilloscope, you may be able to narrow down the conditions even further, for example triggering only on a data word at a particular point in a sequence.

Screenshot: I²C interface decoding and decoder table

3.2 SPI interface (Serial Peripheral Interface)

The SPI interface uses a master-slave architecture where a master controls the communication and one or more slaves can receive and respond to the data. The interface uses the following lines:
- SS/CS (Slave Select or Chip Select): Used to select the slave when there are several slaves, each of which usually has its own SS/CS line
- SCK (Serial Clock): Clock line for data synchronisation, generated by the master
- MOSI (Master Output, Slave Input): Data line from the master to the slaves
- MISO (Master Input, Slave Output): Data line from the slaves to the master

Select 1	Communication (MOSI/MISO)	Clock edge	Clock idle	Word size	Bit order
0	Listing of some EU countries and their capitals	↥	0	8 Bit	MSB
1	Country / Capital	↥	0	8 Bit	MSB
2	Country / Capital	↧	0	8 Bit	MSB
3	Country / Capital	↥	0	8 Bit	LSB
4	Country / Capital	↧	0	8 Bit	LSB
5	Country / Capital	↥	1	8 Bit	MSB
6	Country / Capital	↧	1	8 Bit	MSB
7	Country / Capital	↥	1	8 Bit	LSB
8	Country / Capital	↧	1	8 Bit	LSB
9	Country / Capital	↥	0	9 Bit	MSB
A	Country / Capital	↧	0	9 Bit	MSB
B	Country / Capital	↥	0	9 Bit	LSB
C	Country / Capital	↧	0	9 Bit	LSB
D	Country / Capital	↥	1	9 Bit	MSB
E	"Batronix" with occasional errors / Counter	↥	0	8 Bit	MSB
F	Joke	↥	0	8 Bit	MSB

Select 2	0	1	2	3	4	5	6	7
Bit rate [kbit/s]	10	15	20	30	50	80	100	150

Select 2	8	9	A	B	C	D	E	F
Bit rate [kbit/s]	200	300	500	800	1000	1500	2000	3000

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "8".
- If you have a 4-channel oscilloscope, measure each of the signals "SCK", "MOSI", "MISO" and "SS/CS" using a probe. If you have a 2-channel oscilloscope, measure "SCK" and "MOSI".
- Set the trigger mode "Edge" to the "SS/CS" signal (4 channels) or to "SCK" (2 channels) and move the trigger level to the middle of the signal amplitude.
- Set up the vertical and horizontal systems of your oscilloscope. The horizontal setting should be such that a data word is fully visible. The individual data words are separated by pauses, making them easy to identify.
- Use the "Single" function of your oscilloscope to display a single waveform.
- Set up your oscilloscope's decoder. In addition to selecting the interface and assigning the corresponding channels used to "SCK", "MOSI", "MISO" and "SS/CS", you should also pay attention to the threshold setting. Set this to about the middle of the signal level, e.g. 1.6 V. Set the "Clock Polarity" to rising edge, the word width to 8 bits, the bit order to MSB first and the display data format to "ASCII".
- On a 2-channel oscilloscope, set the clock timeout to 100 µs as a "substitute" for the unmeasured "SS/CS" signal.
- Start a new "Single" recording or leave the oscilloscope in "Run" mode.
- If you now see the name of an EU country (MOSI signal) or the name of an EU capital (MISO signal), you have done everything correctly. If you do not see any decoding information, or if you see unexpected or incorrect decoding information, check the above points again and, if necessary, compare the selected decoder settings with your oscilloscope's manual.

Screenshot: SPI interface decoding and decoder table

3.3 UART interface (Universal Asynchronous Receiver / Transmitter)

The UART interface uses an asynchronous communication architecture where data is sent (transmitted) and received between two devices without a separate clock line. Instead start and stop bits are used to mark the beginning and end of a data word.

The interface uses the following lines:
 • TX (Transmit): Data line, transmit direction
 • RX (Receive): Data line, receive direction

Select 1	Communication (Tx/Rx)	Parity	Stop bits	Data bits	Bit Order
0	Listing of some EU countries and their capitals	none	1 bit	8 bit	LSB
1	Country / Capital	none	1 bit	8 bit	LSB
2	Country / Capital	even	1 bit	8 bit	LSB
3	Country / Capital	odd	1 bit	8 bit	LSB
4	Country / Capital	none	2 bit	8 bit	LSB
5	Country / Capital	even	2 bit	8 bit	LSB
6	Country / Capital	odd	2 bit	8 bit	LSB
7	Country / Capital	none	1 bit	7 bit	LSB
8	Country / Capital	even	1 bit	7 bit	LSB
9	Country / Capital	odd	2 bit	7 bit	LSB
A	Country / Capital	none	1 bit	9 bit	LSB
B	Country / Capital	even	1 bit	9 bit	LSB
C	Country / Capital	odd	2 bit	9 bit	LSB
D	Country / Capital	none	1 bit	8 bit	MSB
E	"Batronix" with occasional errors / Counter	none	1 bit	8 bit	LSB
F	Joke	none	1 bit	8 bit	LSB

Select 2	0	1	2	3	4	5	6	7
Bit rate [kbit/s]	9.6	12.5	14.4	19.2	28.8	38.4	56	57.6

Select 2	8	9	A	B	C	D	E	F
Bit rate [kbit/s]	115.2	128	230.4	256	460.8	500	512	1000

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "0".
- Measure the UART signals "Tx" and "Rx" with one probe each.
- First set the trigger mode to Edge on one of the two channels and move the trigger level to the centre of the signal amplitude.
- Set up the vertical and horizontal systems of your oscilloscope so that the channels use a large part of the vertical of the vertical screen size. The horizontal setting should be such that a data word is completely visible. The individual data words are separated by pauses, making them easy to identify.
- Use the "Single" function of your oscilloscope to display a single record.
- Set up your oscilloscope's decoder. In addition to selecting the interface and assigning the UART channels "Tx" and "Rx", you should pay attention to setting the threshold level, bit rate and other UART decoder settings. The correct parameters for the above selector switch settings are as follows: Bit rate (Baud rate) 9.6 kbit/s, bit order LSB first, 8 data bits, 1 stop bit, no parity bits and idle level high (if configurable). Set the display data format to "ASCII".
- Start a new "Single" capture or leave the oscilloscope in run mode.
- If you now see the name of an EU country (Tx) or the name of an EU capital (Rx), you have done everything correctly. If you do not see any decoding information, or if you see unexpected or incorrect decoding information, check the above points again and, if necessary, compare the selected decoder settings with your oscilloscope's manual.
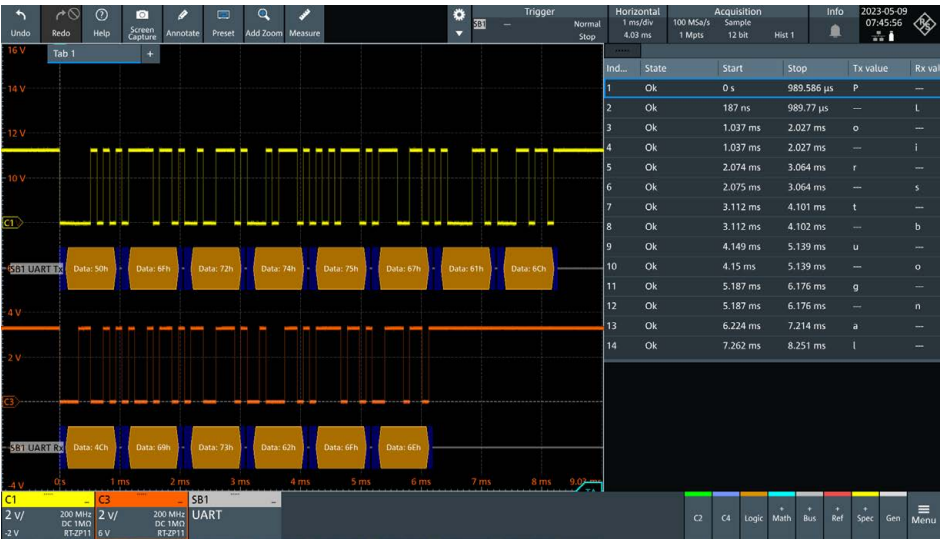

Screenshot: UART interface decoding and decoder table

3.4 Parallel bus and CAN/CAN-FD/LIN communication

The parallel bus and CAN/CAN-FD/LIN/Manchester communication share lines D0 to D7. Selection is made using the selector switch Select 1.

Select 1	Communication	Interface
0	"Batronix"	CAN / CAN-FD / LIN / Manchester
1 to D	Country / Capital	Parallel bus 7 Bit + Clock on D7
E	"Batronix" with occasional errors / Counter	Parallel bus 7 Bit + Clock on D7
F	Joke	Parallel bus 7 Bit + Clock on D7
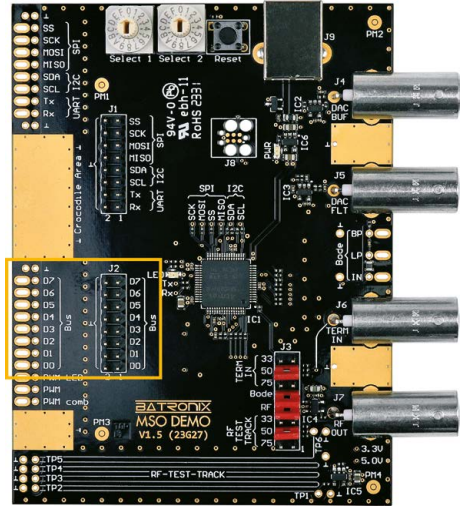
3.5 Parallel bus interface

Unlike serial interfaces such as SPI, I²C or UART, where data is transmitted bit by bit, parallel buses transmit data simultaneously on multiple lines.

Select 2	0	1	2	3	4	5	6	7
Bit rate [kbit/s]	2	2.5	4	5	6	7.5	10	12.5

Select 2	8	9	A	B	C	D	E	F
Bit rate [kbit/s]	15	20	25	30	35	40	45	50

Data lines D0 to D7 of the parallel bus can be measured at connector J2 as well as at the tapped holes on the left side of the board as long as the selector switch Select 1 is set to "1" to "F". If the selector switch is set to "0", CAN, CAN-FD, LIN and Manchester communication is available (see chapter 3.6).

The parallel bus interface on the demo board has seven data lines (D0 to D6) and a clock signal on D7. Data is updated on the falling edge of D7 and must be sampled on the rising edge of D7.

Please follow the steps below:
 • Set Select 1 to "E" and Select 2 to "A".
 • Measure data lines D0 to D7. Due to the number of lines, you will need a mixed-signal oscilloscope (MSO), an oscilloscope with eight analog channels or a logic analyzer with at least eight channels.
 • First set the trigger mode to edge and trigger on D7.
 • The horizontal setting should be such that a communication is fully visible. The individual communications are separated by pauses, making them easy to identify.
 • Use the "Single" function of your oscilloscope to display a single record.
 • Set up the decoder on your oscilloscope. The clock line is on line D7.

Screenshot: Parallel bus interface decoding and decoder table

3.6 CAN/CAN-FD/LIN/Manchester communication

CAN, CAN-FD, LIN and Manchester communication on the demo board is performed in a simplified manner using 3.3 V CMOS voltage levels instead of the levels and differential lines defined in the respective interface standards.

A pulse of the length of the respective communication output is provided on data line D0, which may be used, for example, as a triggering aid.

Please note:
- Select 1 must be set to "0".
- The CAN and CAN-FD signals all correspond to CAN_H.
- The data rate for CAN-FD remains constant (the data rate set with Select 2 also applies to the "flex" range).
- Manchester configuration: MSB first, 8 bits per transmission, polarity falling edge = 0, idle = high, no preamble (sync/header) bits, start with first edge.
- With Manchester communication, the bit rate is only half that shown in the table below, due to the combination with the clock signal.

Data line	Interface	Communication
D7	CAN	"Batronix"
D6	CAN	"Batronix" with occasional errors
D5	CAN-FD	"Batronix"
D4	CAN-FD	"Batronix" with occasional errors
D3	LIN	"Hi"
D2	LIN	"Hi" with occasional errors
D1	Manchester	"Batronix"
D0	Puls	-

Select 2	0	1	2	3	4	5	6	7
Bit rate [kbit/s]	10	10	10	10	10	10	10	12

Select 2	8	9	A	B	C	D	E	F
Bit rate [kbit/s]	15	18	20	22	25	30	35	40

CAN, CAN-FD, LIN and Manchester communication can be selected on the parallel bus data lines D0 to D7. The corresponding signals can be measured at connector J2 as well as at the taps on the left side of the board as long as the selector switch Select 1 is set to "0". If this selector is set to "1" to "F", the 7-bit bus is used (see chapter 3.5).

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "A" (CAN, etc. at 20 kbit/s).
- Measure the signal "D7" (CAN_H) with a probe.
- First set the trigger mode to "Edge" and move the trigger level to the centre of the signal amplitude.
- The horizontal setting should be such that a communication is fully visible. The individual communications are separated by pauses, making them easy to identify.
- Use the "Single" function of your oscilloscope to display a single record.
- Set up the decoder on your oscilloscope. In addition to selecting the CAN interface and assigning the channel as "CAN_H", you should pay attention to the setting of the threshold level and the bit rate (here 20 kbit/s).
- Also try the other lines, e.g. CAN-FD on D5 and LIN on D3.

Screenshot: CAN interface decoding and decoder table

4 RF signals and terminations

The demo board contains two fast logic gates with rise and fall times of less than 450 ps each. This allows a number of RF effects to be demonstrated, as well as the need for good probe grounding and proper termination.

Component	Function
"Bode"	Signal taps of the Bode plot demo filter ・ IN: Filter input signal ・ LP: Low-pass filter output ・ BP: Band-pass filter output The signal from the generator is connected to "TERM IN" (J6) and the jumper "Bode" must be set (J3).
"TERM IN" J6	BNC input socket with adjustable termination (33/50/75/open), also input for the Bode plot demo filter
"RF OUT" J7	BNC output socket for the RF signal
"RF-TEST-TRACK"	RF test track for demonstration of RF effects and terminations
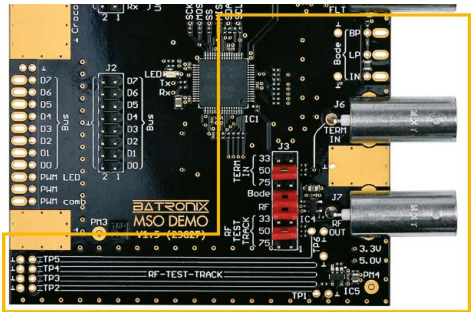
Jumper connector J3		Funktion
"TERM IN"	33	33 Ω termination of "TERM IN" (J6)
	50	50 Ω termination of "TERM IN" (J6)
	75	75 Ω termination of "TERM IN" (J6)
"Bode"		Connecting the demo filter and the Bode pins IN, LP, BP to the BNC socket "TERM IN" (J6)
"RF"		Activation of the RF signal on the RF test track and the BNC socket "RF OUT" (J7)
"RF TEST TRACK"	33	33 Ω termination of the RF test track
	50	50 Ω termination of the RF test track
	75	75 Ω termination of the RF test track

4.1 Probe ground connection

With fast signal edges, a good and as short as possible ground connection of the probe is essential to avoid too much distortion of the signal during the measurement.
 • Set Select 1 to "0" and Select 2 to "8".
 • Set the jumpers to "RF" and both to "50" (J3).
 • Contact test point TP1 in the RF test track area with a probe.
 • Set the vertical scaling to 500 mV/div and the horizontal scaling to 20 ns/div. Set the vertical position (channel offset) near the bottom edge of the display area.
 • For the time being, leave the probe's ground terminal unconnected and look at the measured signal on the screen. You will see a square wave signal with strong overshoots and post-edge oscillations (screenshot: red signal curve).
 • Now connect the ground cable to your probe and the alligator clip of the cable to the ground of the demo board (e.g. to one of the BNC sockets or to a gold ground plane). The overshoots will become smaller, but will still be visible (screenshot: blue signal curve).
 • Now use the grounding spring (if present) of your probe instead of the ground cable and connect it to the ground test point directly to the right of test point "TP1". The measurement result should now look much better and more like the real signal (screenshot: orange signal curve).
 • If available, you can now try an active probe. These have a much smaller input capacitance and therefore load (or change) the signal much less (screenshot: green waveform).

Screenshot: The effect of different probe ground connections at fast edges

4.2 Signal influence by probes

The effect of a probe on a signal can be significant on fast edges and, in some circuits, can even cause the circuit to fail while measuring.

Please follow the steps below:
- Set the jumpers to "RF" and both to "50" (J3).
- Connect a probe with the best possible earth connection (with earth spring if available) to test point TP1 (screenshot: green signal curve).
- Now connect a second probe to the same test point (TP1). Establish an optimal earth connection there too. You can see how the connection of the second probe loads your signal curve at the first probe (screenshot: blue signal curve).
- Observe how the signal from the first probe changes when you connect and disconnect the second probe.
- Now connect the second probe to test point TP3. The impact on the signal at TP1 can now be encountered at a later point in time (relative to the edge). (screenshot: yellow signal curve).
- Try the other test points (TP2-TP6) and see the effect.
- If you have active probes, you can also perform the experiment with them. The effect will be much smaller, depending on the input capacitance of the probes.

Screenshot: Signal influence due to loading with a probe

4.3 Termination and conductor impedances

As long as the lines are short enough for the speed of the signal, Ohm's law states that the current flow is equal to the voltage divided by the resistance. But if, for example, an IC switches its output very quickly, there is a very fast voltage change and if the resistor is at the end of a "long" line, the current flow does not immediately match this resistor. With this fast signal edge, the IC output driver cannot "see" the termination at the end of the trace.

This situation, where the trace length is more than a tenth of the wavelength, is called electrically "long". The current flow in an electrically "long" interconnection is initially determined by the geometric properties of the trace, which determine the trace impedance.

For RF signals, the trace geometry on printed circuit boards is usually designed to obtain a trace impedance of 50 Ω. The decision to use 50 Ω was a compromise between the lowest attenuation (which occurs at 77 Ω) and the highest power transmission (at 30 Ω) of coaxial cable. In satellite/antenna technology, 75 Ω is common because the emphasis is on lowest attenuation.

Back to RF signals on printed circuit boards: Here, for example, the output signal of a fast operational amplifier is fed through a 50 Ω output resistor and then terminated at the destination with 50 Ω to ground. This means that only half the output voltage reaches the receiver, but if done correctly it is "clean" with little or no overshoot and reflections.

A calculation example: For example, when an IC output changes from 0 V to 3 V, 30 mA flows into this line (3 V / (50 Ω output resistance + 50 Ω trace impedance)). At the end of the line, this 30 mA flows at 1.5 V through the 50 Ω termination resistor to ground, and the voltage is 1.5 V as expected (3 V at the voltage divider of two 50 Ω resistors).

On the other hand, if the trace impedance and the termination do not match, reflections will occur after the signal edge arrives at the termination resistor, resulting in a returning wave. The voltage adapts to the divider ratios of the two "real" resistors. If there is no terminating resistor at the end and only a high impedance IC input, the 30 mA current that has flowed into the line cannot flow out and flows back to the output. This can also be seen by measuring the voltage with a fast oscilloscope. The voltage of 1.5 V at the beginning of the line rises to 3 V shortly afterwards with the returning wave.

If, on the other hand, the terminating resistor is smaller than the trace impedance, say 33 Ω, the 30 mA will flow through the terminating resistor and the voltage will drop to about 1.2 V (3 V / 83 Ω * 33 Ω) with the returning wave.

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "8".
- Use a ground spring (if available) and connect it to the ground measuring point directly to the right of test point "TP1". An alligator clip earth cable can also be used, but the effect will be less obvious due to overshoot and post-edge oscillation.
- Set the jumpers to "RF" and the lower "50" (both J3).
- Place a probe on test point "TP1" in the "RF-Test-Track" area (screenshot: green signal curve).
- Now remove the jumper connected to the lower "50". This will remove the termination and you should see a voltage rise to approx. 4.8 V about 8 ns after the signal edge (from 0 V to approx. 2.4 V) (screenshot: red signal curve).
- Now set the jumper to the lower "75". The termination is now 75 Ω. The measured voltage now rises a few nanoseconds after the signal edge from about 2.4 V to about 2.9 V (screenshot: blue signal curve).
- Now set the jumper to the lower "33". The termination is now 33 Ω. The measured voltage now drops from approx. 2.4 V to approx. 1.9 V a few nanoseconds after the signal edge (screenshot: yellow signal curve).
- You can also place a second probe at a different test point (TP2-TP6) and examine the curve at different points on the trace.
- At the last test point (TP6), the leading edge of the signal and the trailing adaptation appear practically simultaneously, so you can only see the jump to the "matching" voltage here.
- Set the jumper back to "50" for subsequent tests.

Screenshot: The effect of correct termination for "electrically long" signals

4.4 Cables and stubs

The following experiment will demonstrate the wave delay using a stub line.

Please follow the steps below:
- Set Select 1 to "0" and Select 2 to "8".
- Set the jumper to "RF".
- Connect a BNC tee adapter to one of the inputs of your oscilloscope.
- Activate the 50 Ω termination for this input on your oscilloscope if it is available. If not, proceed without making any adjustments.
- Connect the "RF OUT" signal (J7) using a BNC cable to the BNC tee adapter (screenshot: green signal trace).
- Adjust the vertical, horizontal and trigger system so that it looks similar to the following screenshot (green signal curve).
- Connect another BNC cable to the BNC tee adapter, leaving the other end of the cable open. The length of the cable determines the time it takes for the wave to travel back and forth. In the screenshot below, a 2m cable has been connected (screenshot: yellow signal trace).

Keep experimenting, the sky's the limit when it comes to being creative:
- Use longer or shorter BNC cables.
- For example, the free end of the cable can be routed back to the demo board and terminated with 75 Ω at "TERM IN" (J6).
- Additional BNC tee adapters can be used to add more spur lines and study their effects.

Screenshot: Reflections on non-terminated lines

5 Glossary and list of abbreviations

Term	Description / Explanation
AC	AC (alternating current) stands for alternating current or alternating voltage.
ADC	An analog-to-digital converter (ADC) is a device that converts analog input signals into digital data that can be processed by subsequent digital devices (e.g. microcontrollers, processors, FPGAs). In oscilloscopes, for example, ADCs are used to convert analog signal voltages into discrete digital values.
AM	AM (amplitude modulation) is a modulation technique in which the amplitude of a high-frequency carrier signal is varied according to the signal to be transmitted. The information is encoded in the amplitude of the carrier signal.
BNC	BNC (Bayonet Neill-Concelman) is a special type of connector often used in electronics and telecommunications. It provides a fast and secure connection for coaxial cables and is used in measurement technology, among other things.
DAC	A DAC (digital-to-analog converter) is a device that converts digital data into analog output signals. Digital components (e.g. microcontrollers, processors, FPGAs) can generate a data stream that is converted to an analog voltage by the DAC. Function generators, among others, use DACs to convert digital data into analog signal voltages.
DC	DC (direct current) stands for direct current or direct voltage.
GND	GND (ground) is the reference point in an electrical system or circuit and is used as the reference potential for other voltages in the system.
IC	An IC (Integrated Circuit) is a device that combines a large number of electronic components such as transistors, resistors and capacitors on a small silicon chip. ICs are used to control, process and amplify electrical signals.
PWM	PWM (pulse-width modulation) is a modulation technique in which the width of a periodic signal, usually a square wave, is varied to control the average power or voltage. It is used, among other things, to control the speed of motors or the luminosity of LEDs, or to digitally represent analog signals.
RF	RF (radio frequency) refers to high frequencies or fast signal edges.