



Instruction Manual

Trigger, Decode,
Measure/Graph
Eye Diagrams

for:

USB2 and USB2-HSIC



USB2 and USB2-HSIC TDME Software Instruction Manual

© 2016 Teledyne LeCroy, Inc. All rights reserved.

Unauthorized duplication of Teledyne LeCroy, Inc. documentation materials other than for internal sales and distribution purposes is strictly prohibited. However, clients are encouraged to duplicate and distribute Teledyne LeCroy, Inc. documentation for their own internal educational purposes.

Teledyne LeCroy is a trademark of Teledyne LeCroy, Inc., Inc. Other product or brand names are trademarks or requested trademarks of their respective holders. Information in this publication supersedes all earlier versions. Specifications are subject to change without notice.

922295 Rev C
September 2016

Contents

About the Options	1
About the USB2bus Option	1
About the USB2-HSICbus Option	1
Serial Decode	2
Bit-level Decoding	2
Logical Decoding	2
Message Decoding	2
User Interaction	2
Decoding Workflow	3
Decoder Set Up	3
Failure to Decode	5
Serial Decode Dialog	5
Reading Waveform Annotations	6
Serial Decode Result Table	7
Searching Decoded Waveforms	14
Decoding in Sequence Mode	15
Serial Trigger	16
Requirements	16
Restrictions	16
Linking Trigger and Decoder	16
USB2 Trigger Setup	17
Using the Decoder with the Trigger	21
Saving Trigger Data	22
Measure/Graph	23
Serial Data Measurements	23
Graphing Measurements	24
Measure/Graph Setup Dialog	24
Filtering Measurements	25
Eye Diagram Tests	27
Eye Diagram Setup Dialog	27
Mask Failure Locator Dialog	28
Technical Support	29
Live Support	29
Resources	29
Service Centers	29

About This Manual

Teledyne LeCroy offers a wide array of toolsets for decoding and debugging serial data streams. These toolsets may be purchased as optional software packages, or are provided standard with some oscilloscopes.

This manual explains the basic procedures for using serial data decoder and trigger software options. There are also sections pertaining to the measure and graphing capabilities and eye diagram tests.

It is assumed that:

- You have purchased and activated one of the serial data products described in this manual.
- You have a basic understanding of the serial data standard physical and protocol layer specifications, and know how these standards are used in controllers.
- You have a basic understanding of how to use an oscilloscope, and specifically the Teledyne LeCroy oscilloscope on which the option is installed. Only features directly related to serial data triggering and decoding are explained in this manual.

Teledyne LeCroy is constantly expanding coverage of serial data standards and updating software. Some capabilities described in this documentation may only be available with the latest version of our firmware. You can download the free firmware update from:

teledynelecroy.com/support/softwaredownload

While some of the images in this manual may not exactly match what is on your oscilloscope display—or may show an example taken from another standard—be assured that the functionality is identical, as much functionality is shared. Product-specific exceptions will be noted in the text.

About the Options

Decode

Teledyne LeCroy decoders apply software algorithms to extract serial data information from physical layer waveforms measured on your oscilloscope. The extracted information is displayed over the actual physical layer waveforms, color-coded to provide fast, intuitive understanding of the relationship between message frames and other, time synchronous events.

Trigger

Trigger *and* decode (-TD) options enable you to trigger the oscilloscope acquisition upon finding specific message frames, data patterns, or errors in serial data streams. Conditional filtering at different levels enables you to target the trigger to a single message or a range of matching data.

Measure/Graph and Eye Diagrams

The installation of any -DME or -TDME option adds a set of measurements designed for serial data analysis and protocol-specific eye diagram tests to the standard trigger and decoder capabilities. See [Measuring](#) for instructions on using the measure and graphing capabilities. See [Eye Diagram Tests](#) for instructions on using the eye diagram tests.

About the USB2bus Option

USB 2.0 is a version of the Universal Serial Bus specification, which is used to connect a host controller (typically a PC) to various devices. The 2.0 version brought a higher maximum bandwidth (480 Mb/s) and features such as the Mini-B Connector, Battery Charging, and Micro-USB Cables and Connectors specifications, to name a few.

USB standards are maintained by the non-profit USB Implementers Forum, Inc. (USB-IF). Additional information, including specifications for specific versions, can be found on their website at www.usb.org/.

The USB2bus TD and TDME options provide comprehensive USB1.1x and 2.0 (low, full and high speed) protocol decoding and triggering using a true hardware trigger for most efficient debug.

About the USB2-HSICbus Option

High-Speed Inter-Chip (HSIC) technology communicates between microchips in an embedded sub-system and leverages the traditional host-device topology of USB, utilizing a large amount of existing USB code. This makes HSIC extremely lucrative in areas demanding fast time-to-market solutions, such as the mobile industry including both smart phones and tablets.

The USB2-HSICbus D option extracts transaction-level data from a USB2-HSIC signal to show significant transitions in the signal.

Serial Decode

The algorithms described here at a high level are used by all Teledyne LeCroy serial decoders sold for oscilloscopes. They differ slightly between serial data signals that have a clock embedded in data and those with separate clock and data signals.

Bit-level Decoding

The first software algorithm examines the embedded clock for each message based on a default or user-specified vertical threshold level. Once the clock signal is extracted or known, the algorithm examines the corresponding data signal at the predetermined vertical level to determine whether a data bit is high or low. The default vertical level is set to 50% and is determined from a measurement of peak amplitude of the signals acquired by the oscilloscope. For most decoders, it can also be set to an absolute voltage level, if desired. The algorithm intelligently applies a hysteresis to the rising and falling edge of the serial data signal to minimize the chance of perturbations or ringing on the edge affecting the data bit decoding.



Note: Although the decoding algorithm is based on a clock extraction software algorithm using a vertical level, the results returned are the same as those from a traditional protocol analyzer using sampling point-based decode.

Logical Decoding

After determining individual data bit values, another algorithm performs a decoding of the serial data message after separation of the underlying data bits into logical groups specific to the protocol (Header/ID, Address Labels, Data Length Codes, Data, CRC, Parity Bits, Start Bits, Stop Bits, Delimiters, Idle Segments, etc.).

Message Decoding

Finally, another algorithm applies a color overlay with annotations to the decoded waveform to mark the transitions in the signal. Decoded message data is displayed in tabular form below the grid. Various compaction schemes are utilized to show the data during a long acquisition (many hundreds or thousands of serial data messages) or a short acquisition (one serial data message acquisition). In the case of the longest acquisition, only the most important information is highlighted, whereas in the case of the shortest acquisition, all information is displayed with additional highlighting of the complete message frame.

User Interaction

Your interaction with the software in many ways mirrors the order of the algorithms. You will:

- Assign a protocol/encoding scheme, an input source, and a clock source (if necessary) to one of the four decoder panels using the Serial Data and Decode Setup dialogs.
- Complete the remaining dialogs required by the protocol/encoding scheme.
- Work with the decoded waveform, result table, and measurements to analyze the decoding.

Decoding Workflow

We recommend the following workflow for effective decoding:

1. Connect your data and strobe/clock lines (if used) to the oscilloscope.
2. Set up the decoder using the lowest level decoding mode available (e.g., Bits).
3. Acquire a sufficient burst of relevant data. The data burst should be reasonably well centered on screen, in both directions, with generous idle segments on both sides.



Note: See [Failure to Decode](#) for more information about the required acquisition settings. A burst might contain at most 100000 transitions, or 32000 bits/1000 words, whichever occurs first. This is more a safety limit for software engineering reasons than a limit based on any protocol. We recommend starting with much smaller bursts.

4. Stop the acquisition, then run the decoder.
5. Use the various decoder tools to verify that transitions are being correctly decoded. Tune the decoder settings as needed.
6. Once you know you are correctly decoding transitions in one mode, continue making small acquisitions of five to eight bursts and running the decoder in higher level modes (e.g., Words). The decoder settings you verify on a few bursts will be reused when handling many packets.
7. Run the decoder on acquisitions of the desired length.

When you are satisfied the decoder is working properly, you can disable/enable the decoder as desired without having to repeat this set up and tuning process, provided the basic signal characteristics do not change.

Decoder Set Up

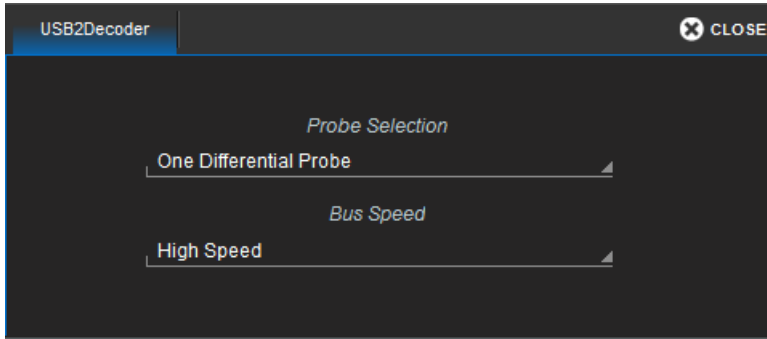
Use the Decode Setup dialog and its protocol-related subdialogs to preset decoders for future use. Each decoder can use different protocols and data sources, or have other variations, giving you maximum flexibility to compare different signals or view the same signal from multiple perspectives.

1. Touch the **Front Panel Serial Decode button** (if available on your oscilloscope), or choose **Analysis > Serial Decode** from the oscilloscope menu bar. Open the **Decode Setup** dialog.
2. From the buttons at the left, select the **Decode #** to set up.
3. Select the **data source (Src 1)** to be decoded and the **Protocol** to decode.
4. If required by the protocol, also select the **Strobe** or **Clock** source. (These controls will simply not appear if not relevant.)
5. Define the bit- and protocol-level decoding on the subdialogs next to the Decode Setup dialog.



Tip: After completing setup for one decoder, you can quickly start setup for the other decoders by using the buttons at the left of the Decode Setup dialog to change the Decode # .

USB2 Decoder Settings

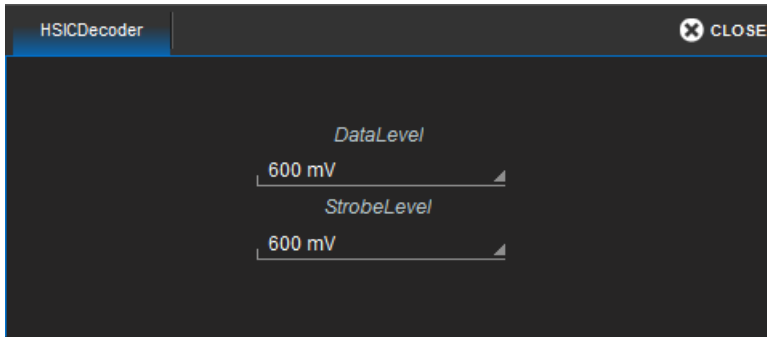


Make a **Probe Selection** matching your physical setup: One Differential Probe, Two Single Ended Probes or One Single Ended Probe.

If using Two Single Ended Probes, enter your **Dp** (Data positive) and **Dn** (Data negative) source channels separately on the Decode Setup dialog to the left.

Bus Speed - select **Low**, **Full**, or **High**.

USB2-HSIC Decoder Settings



DataLevel - Enter the crossing level for the data waveform in Volts. The DataLevel appears as a dotted horizontal line across the oscilloscope grid. If initial decoding indicates that there are a number of error messages, make sure that the level is set to a reasonable value.

StrobeLevel - Enter the crossing level for the strobe waveform in Volts.

Failure to Decode

Three conditions in particular may cause a decoder to fail, in which case a failure message will appear in the first row of the summary result table, instead of in the message bar as usual.

All decoders will test for the condition **Too small amplitude**. If the signal's amplitude is too small with respect to the full ADC range, the message "Decrease V/Div" will appear. The required amplitude to allow decoding is usually one vertical division.

If the decoder incorporates a user-defined bit rate (usually these are protocols that do not utilize a dedicated clock/strobe line), the following two conditions are also tested:

- **Under sampled.** If the sampling rate (SR) is insufficient to resolve the signal adequately based on the bit rate (BR) setup or clock frequency, the message "Under Sampled" will appear. The minimum SR:BR ratio required is 4:1. It is suggested that you use a slightly higher SR:BR ratio if possible, and use significantly higher SR:BR ratios if you want to also view perturbations or other anomalies on your serial data analog signal.
- **Too short acquisition.** If the acquisition window is too short to allow any meaningful decoding, the message "Too Short Acquisition" will appear. The minimum number of bits required varies from one protocol to another, but is usually between 5 and 50.

In all the above cases, the decoding is turned off to protect you from incorrect data. Adjust your acquisition settings accordingly, then re-enable the decoder.



Note: It is possible that several conditions are present, but you will only see the first relevant message in the table. If you continue to experience failures, try adjusting the other settings.

Serial Decode Dialog

To first set up a decoder, go to the [Decode Setup dialog](#). Once decoders have been configured, use the Serial Decode dialog to quickly turn on/off a decoder or make minor modifications to the settings.

To turn on decoders:

1. On the same row as the **Decode #**, check **On** to enable the decoder.

As long as On is checked (and there is a valid acquisition), a [result table](#) and [decoded waveform](#) appear. The number of rows of data displayed will depend on the **Table#Rows** setting (on the Decode Setup dialog).

2. Optionally, modify the:
 - **Protocol** associated with the decoder.
 - **Data (Source)** to be decoded.
3. Check [Link To Trigger On](#) to tie this decoder setup to a serial trigger setup.

To turn off decoders: deselect the On boxes individually, or touch **Turn All Off**.

Reading Waveform Annotations

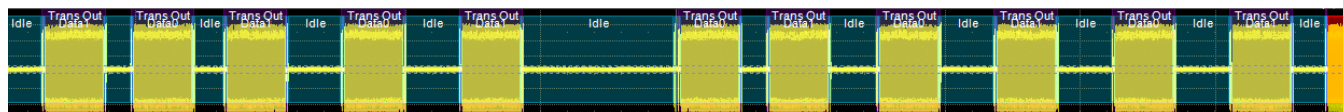
When a decoder is enabled, an annotated waveform appears on the oscilloscope display, allowing you to quickly see the relationship between the protocol decoding and the physical layer. A colored overlay marks significant bit-sequences in the source signal: Header/ID, Address, Labels, Data Length Codes, Data, CRC, Parity Bits, Start Bits, Stop Bits, Delimiters, Idle segments, etc. Annotations are customized to the protocol or encoding scheme.

The amount of information shown on an annotation is affected by the width of the rectangles in the overlay, which is determined by the magnification (scale) of the trace and the length of the acquisition. Zooming a portion of the decoder trace will reveal the detailed annotations.

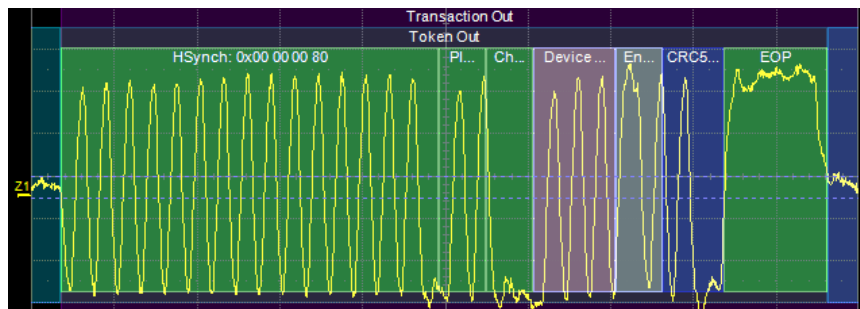
These overlays appear on decoded USB2 and USB2-HSIC waveforms and their zoom traces.

Annotation	Overlay Color (1)	Text (2)
Transaction	Dark Purple (behind all fields)	Transaction <Type> <In Out>
Packet Type	Charcoal (behind other fields)	<Handshake Token Data> <In Out>
Control Sequences	Green	<Synch PID Check EOP> bits
Device Address	Light Purple	Device Address=<value>
Endpoint	Grey	Endpoint
Payload Data	Green	Data=<value>
Cyclic Redundancy Check	Royal Blue	CRC<value>
Inter-packet Idle	Royal Blue	Idle
Inter-transaction Idle	Blue-Green	Idle

1. Combined overlays affect the appearance of colors.
2. Text in brackets < > is variable. The amount of text shown depends on your zoom factors.



Decoded USB2 waveform. At this resolution, little information appears on the overlay.



Zoom of transaction, showing annotation details.

Serial Decode Result Table

By default, a table summarizing the decoder results appears below the grids whenever a decoder is enabled. The result table provides a view of data as decoded during the most recent acquisition, even when the number of bursts are too many to allow legible annotation on the waveform trace.

The table is displayed only when the **View Decode** checkbox is marked on the Decode Setup Dialog *and* a source signal has been decoded using that protocol.

You can [export result table data](#) to a .CSV file.

Table Rows

Each row of the table represents one index of data found within the acquisition, numbered sequentially. Exactly what this represents depends on the protocol and how you have chosen to "packetize" the data stream when configuring the decoder (frame, message, packet, etc.).



Note: For some decoders, it is even possible to turn off packetization, in which case all the decoded data appears on one row of the table.

When multiple decoders are run at once, the index rows are combined in a summary table, ordered according to their acquisition time. The Protocol column is colorized to match the input source that resulted in that index.

You can [change the number of rows](#) displayed on the table at one time. The default is five rows.

Swipe the table up/down or use the scrollbar at the far right to navigate the table. See [Using the Result Table](#) for more information about how to interact with the table rows to view the decoding.

Table Columns

When a single decoder is enabled, the result table shows the protocol-specific details of the decoding. This **detailed result table** may be [customized](#) to show only those columns you want displayed.

A top-level **summary result table** combining results from two decoders will show these columns of data. It cannot be customized.

Column	Extracted or Computed Data
Index	Number of the line in the table
Time	Time elapsed from start of acquisition to start of message
Protocol	Protocol being decoded
Message	Message identifier bits
Data	Data payload
CRC	Cyclic Redundancy Check sequence bits
Status	Any decoder messages; content may vary by protocol

Index	Time	Protocol	Message	Data	CRC	Status
▶ 1	-1.096 μs	USB2	Idle			
▶ 2	-1.096 μs	USB2	Idle			
▶ 3	-994 ns	USB2	Idle	Handshake: ACK		
▶ 4	-994 ns	USB2	Setup		CRC5: 0...	
▶ 5	-994 ns	USB2	Setup	Handshake: ACK		
▶ 6	-994 ns	USB2	Setup		CRC5: 0...	▲ 24
▶ 7	-859 ns	USB2	Idle			

Example summary result table, with results from two decoders combined on one table.

When you select the Index number from the summary result table, the detailed results for that index drop-in below it.

Index	Time	Protocol	Message	Data	CRC	Status	
▶ 1	-1.096 μs	USB2	Idle				
▶ 2	-1.096 μs	USB2	Idle				
▲ 3	-994 ns	USB2	Setup	Handshake: ACK			
		Time(us)	TransType	ItemType	DevAddr	Endpoint	Details
		-0.994339	Setup		0x01	0x0	Handshake: ACK
▶ 4	-994 ns	USB2	Setup				CRC5: 0...
▶ 5	-994 ns	USB2	Setup	Handshake: ACK			

Example summary result table showing drop-in detailed result table.

This extracted USB2 data appears on the detailed result table.

Column	Extracted or Computed Data
Idx (always shown)	Number of the line in the result table
TransIdx	Number of the transaction in the signal, labeled Tx. Packets and idles belonging to this transaction are all prefixed with this number.
Time(μ)	Time in microseconds elapsed from start of acquisition to start of transaction
TransType	Transaction type
ItemType	Packet type
DevAddr	Device Address
Endpoint	Endpoint
Details	Extracted values

USB2	TransIdx	Time(us)	TransType	ItemType	DevAddr	Endpoint	Details
0		-1.096065		Idle			
1	T0	-0.994339	Setup		0x01	0x0	Handshake: ACK
2	0-0	-0.994339		Setup	0x01	0x0	
3	0-1	-0.858728		Idle			
4	0-2	-0.577709		Data0			[Data Len: 8], Data: 02 01 00 00 88 00 00 00
5	0-3	-0.30721		Idle			
6	0-4	-0.104541		ACK			

Section of typical USB2 result table.

Using the Result Table

Besides displaying the decoded serial data, the result table helps you to inspect the acquisition.

Zoom & Search

Touching any cell of the table opens a zoom centered around the part of the waveform corresponding to the index. The Zx dialog opens to allow you to rescale the zoom, or to [Search](#) the acquisition. This is a quick way to navigate to events of interest in the acquisition.



Tip: When in a summary table, touch any data cell *other than* Index and Protocol to zoom.

The table rows corresponding to the zoomed area are highlighted, as is the zoomed area of the source waveform. The highlight color reflects the zoom that it relates to (Z1 yellow, Z2 pink, etc.). As you adjust the zoom scale, the highlighted area may expand to several rows of the table, or fade to indicate that only a part of that Index is shown in the zoom.

When there are multiple decoders running, each can have its own zoom of the decoding highlighted on the summary table at the same time.



Note: The zoom number is no longer tied to the decoder number. The software tries to match the numbers, but if it cannot it uses the next zoom that is not yet turned on.

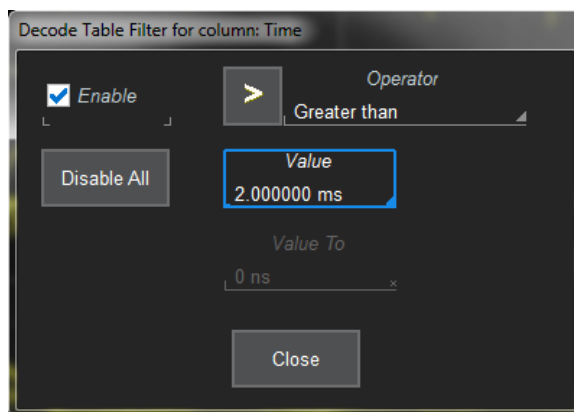
Index	Time	Protocol	Message	Data	CRC	Status
1	-1.096 μs	USB2	Idle			
2	-1.096 μs	USB2	Idle			
3	-994 ns	USB2	Handshake: ACK	Handshake: ACK		
4	-994 ns	USB2	Setup		CRC5: 0...	
5	-994 ns	USB2	Handshake: ACK	Handshake: ACK		
6	-994 ns	USB2	Setup		CRC5: 0...	24
7	-859 ns	USB2	Idle			

Example multi-decoder summary table, both zoomed indexes highlighted.

Filter Results

Those columns of data that have a drop-down arrow in the header cell can be filtered: **Time**

Touch the **header cell** to open the Decode Table Filter dialog.



Select a filter **Operator** and enter a **Value** that satisfies the filter condition.

Operators	Data Types	Returns
=, ≠	Numeric or Text	Exact matches only
>, ≥, <, ≤	Numeric	All data that satisfies the operator
In Range, Out Range	Numeric	All data within/without range limits
Equals Any (on List), Does Not Equal Any (on List)	Text	All data that is/is not an exact match to any full value on the list. Enter a comma-delimited list of values, no spaces before or after the comma, although there may be spaces within the strings.
Contains, Does Not Contain	Text	All data that contains or does not contain the string

Customizing the Result Table

Follow these steps to change what data appears on the detailed result table:

1. Press the Front Panel **Serial Decode button** or choose **Analysis > Serial Decode**, then open the **Decode Setup tab**.
2. Touch the **Configure Table** button.
3. On the **View Columns** pop-up dialog, mark the columns you want to appear and clear those you wish to remove. Only those columns selected will appear on the oscilloscope display.



Note: If a selected column is not relevant to the decoding selections, the column will not appear in the table.

To return to the preset display, touch **Default**.

4. Touch the **Close** button when finished.

On some decoders, you may also use the View Columns pop-up to set a **Bit Rate Tolerance** percentage. When implemented, the tolerance is used to flag out-of-tolerance messages (messages outside the user-defined bitrate +/- tolerance) by colorizing in red the Bitrate shown in the table.

You may customize the size of the result table by changing the **Table # Rows** setting on the Decode Setup dialog. Keep in mind that the deeper the table, the more compressed the waveform display on the grid, especially if there are also measurements turned on.

Exporting Result Table Data

You can manually export the detailed result table data to a .CSV file:

1. Press the Front Panel **Serial Decode button**, or choose **Analysis > Serial Decode**, then open the **Decode Setup tab**.
2. Optionally, touch **Browse** and enter a new **File Name** and output folder.
3. Touch the **Export Table** button.

Export files are by default created in the D:\Applications\



Note: When a summary table is exported, a combined file is saved in D:\Applications\Serial Decode. Separate files for each decoder are saved in D:\Applications\

In addition, the oscilloscope Save Table feature will automatically create tabular data files with each acquisition trigger. The file names are automatically incremented so that data is not lost. Choose **File > Save Table** from the oscilloscope menu bar and select **Decodex** as the source. Make other file format and storage selections as you wish.

Using ColToValue and MsgToValue

These serial decode Measure features are available if you have installed ProtoBus MAG or a Measure (DME/TDME) package. See your oscilloscope *Operator's Manual* for more information about using measure, math, and analysis features.

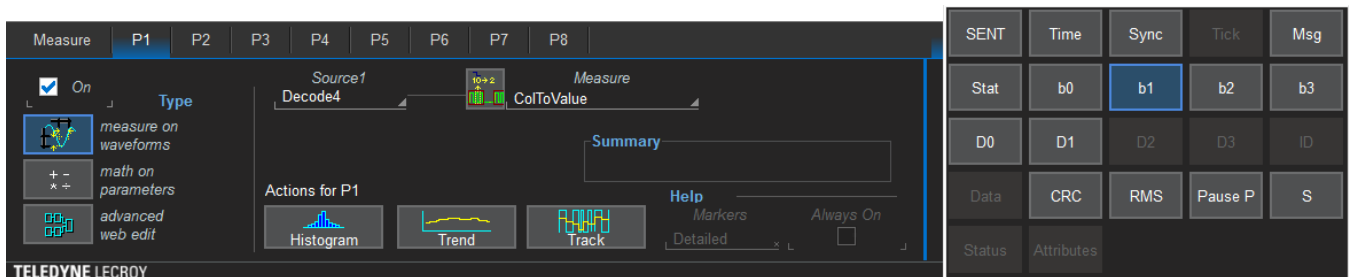
Column to Value

ColToValue acts as a special "pass thru" of one column of decoder table values, allowing you to graph or run Pass/Fail tests on the values via a parameter that is configured with this measurement.

ColToValue acts upon any columns in the table (except those with text values), but does not perform any filtering or extraction of the results. There is no special setup other than the column selection. It is very similar to using Excel to graph a column of data. ColToValue works best if all the lines have similar content. When they do not, use MsgToValue to first filter lines with similar content.

ColToValue can only be selected from the standard measurement set up dialogs under the Serial Decode group, not from the Measure/Graph Setup dialog that is activated with Measure options. There are two keys to using this parameter successfully:

- Because it is not set up like other serial decode measurements, the parameter source is not automatically set on a decoder. You must be sure to select the correct decoder (Decode1 - Decode4), rather than an input channel, as the parameter source. This will expose the decoder table columns for selection on the ColToValue subdialog.
- Be sure there is an active decoding. If there is no data in the result table, the Select Column field on the ColToValue subdialog shows "undef", and the selector is inactive. As long as there is data in the table, you should see a selection of those columns populated by the decoder.

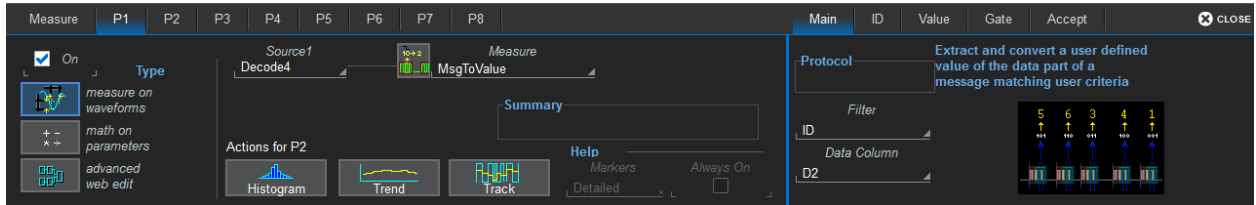


Message to Value

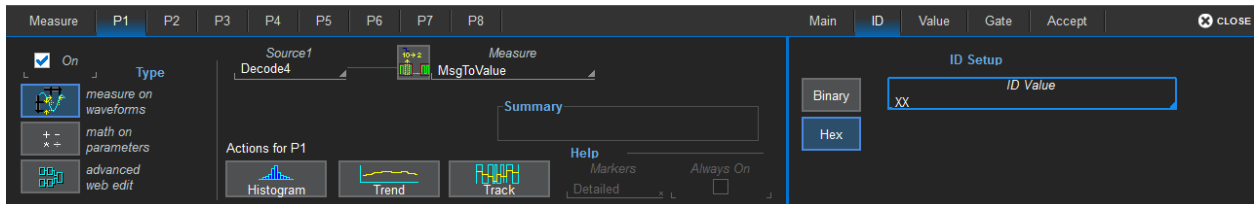
MsgToValue also enables you to apply oscilloscope features to a subset of the result table, but is aimed at protocols with addressed packets containing varying types of data, like CAN, LIN, MIL1553 and many others. You can first filter the table results by a particular ID, as well as extract and convert specific data values, prior to any other math or measurement processing.

MsgToValue requires several selections from the parameter set up subdialogs:

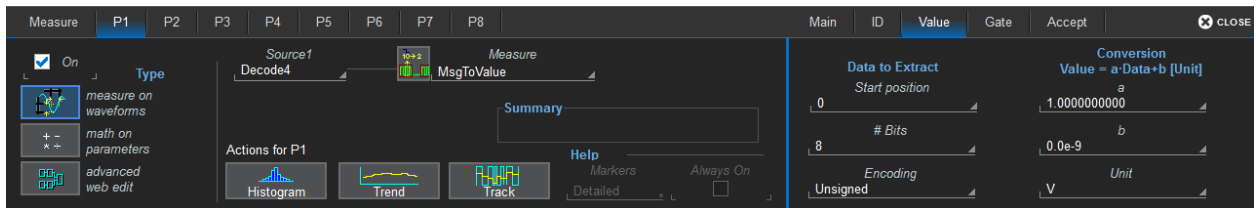
- Choose whether or not you wish to **Filter** by ID or accept **Any** packets.



- If you are filtering by ID, enter the desired ID on the ID subdialog.



- On the Value subdialog, enter the **Data to Extract** and any **Conversion** to be made.



Setting Value Conditions

Follow these steps to define the values to extract:

- Under Data to Extract, begin by entering the **Start position** and the **# Bits** to extract.
- Choose the **Encoding** type if the signal uses encoding, otherwise leave it Unsigned.
- Under Conversion, enter the **a. Coefficient** and **b. Term** that satisfy the formula:
 $Value = Coefficient * Raw Value + Term$.
- Optionally, enter a **Unit** for the extracted decimal value.

Searching Decoded Waveforms

Touching the Action toolbar **Search button** button on the Decode Setup dialog creates a 10:1 zoom of the center of the decoder source trace and opens the Search subdialog.

Touching the **any cell** of the result table similarly creates a zoom and opens Search, but of only that part of the waveform corresponding to the index (plus any padding).



Tip: In summary table mode, touch any cell *other than* Index and Protocol to create the zoom.

Basic Search

On the Search subdialog, select what type of data element to **Search for**. These basic criteria vary by protocol, but generally correspond to the columns of data displayed on the detailed decoder result table.

Optionally:

- Check **Use Value** and enter the **Value** to find in that column. If you do not enter a Value, Search goes to the beginning of the next data element of that type found in the acquisition.
- Enter a **Left/Right Pad**, the percentage of horizontal division around matching data to display on the zoom.
- Check **Show Frame** to mark on the overlay the frame in which the event was found.

After entering the Search criteria, use the **Prev** and **Next** buttons to navigate to the matching data in the table, simultaneously shifting the zoom to the portion of the waveform that corresponds to the match.

The touch screen message bar shows details about the table row and column where the matching data was found.

i Idx = 15 (decimal) found at Row 55 Column 0 going Left

Advanced Search

Advanced Search allows you to create complex criteria by using Boolean AND/OR logic to combine up-to-three different searches. On the Advanced dialog, choose the **Col(umns) to Search 1 - 3** and the **Value** to find just as you would a basic search, then choose the **Operator(s)** that represent the relationship between them.

Decoding in Sequence Mode

Decoders can be applied to Sequence Mode acquisitions. In this case, the index numbers on the result table are followed by the segment in which the index was found and the number of the sample within that segment: *index (segment-sample)*.

CAN Std	Time	Format	ID	IDE	RTR	DLC	Data
2 (2-1)	9.72882 ms	Std	0x400	0	0	2	6a 6b
3 (3-1)	19.7527 ms	Std	0x400	0	0	2	6a 6b
4 (4-1)	30.2558 ms	Std	0x400	0	0	2	6a 6b
5 (5-1)	40.1663 ms	Std	0x400	0	0	2	6a 6b
6 (6-1)	49.8284 ms	Std	0x400	0	0	2	6a 6b
7 (7-1)	59.8595 ms	Std	0x400	0	0	2	6a 6b
8 (8-1)	69.8913 ms	Std	0x400	0	0	2	6a 6b
9 (9-1)	80.4032 ms	Std	0x400	0	0	2	6a 6b
10 (10-1)	89.9384 ms	Std	0x400	0	0	2	6a 6b
11 (11-1)	99.9688 ms	Std	0x400	0	0	2	6a 6b

Example filtered result table for a sequence mode acquisition.

In the example above, each segment was triggered on the occurrence of ID 0x400, which occurred only once per segment, so there is only one sample per segment.

Otherwise, the results are the same as for other types of acquisitions and can be zoomed, filtered, searched, or used to navigate. When a Sequence Mode table is filtered, the waveform annotation appears on only those segments and samples corresponding to the filtered results.



Note: Annotations can only be shown when the Sequence Display Mode is Adjacent.

Multiple decoders can be run on Sequence Mode acquisitions, but in a summary table, each decoder will have a first segment, second segment, etc., and there may be any number of samples in each. As in any summary table, the samples will be interleaved and indexed according to their actual acquisition time. So, you may find (3-2) of one decoder before (1-1) of another. Filter on the Protocol column to see the sequential results for only one decoder.

Also, waveform annotations are not adjusted when a Sequence Mode summary table is filtered, only the table data.

Serial Trigger

TD options provide advanced serial data triggering in addition to decoding. Serial data triggering is implemented directly within the hardware of the oscilloscope acquisition system. The serial data trigger scrutinises the data stream in real time to recognise "on-the-fly" the user-defined serial data conditions. When the desired pattern is recognised, the oscilloscope takes a real-time acquisition of all input signals as configured in the instrument's acquisition settings. This allows decode and analysis of the signal being triggered on, as well as concomittant data streams and analog signals.

The serial trigger supports fairly simple conditions, such as "trigger at the begining of any packet," but the conditions can be made more restrictive depending on the protocol and the available filters, such as "trigger on packets with ID = 0x456". The most complex triggers incorporate a double condition on the ID and data, for example "trigger on packets with ID = 0x456 and when data in position 27 exceeds 1000".

The trigger and decode systems are independant, although they are seamlessly coordinated in the user interface and the architecture. It is therefore possible to trigger without decoding and decode without triggering.

Requirements

Serial trigger options require the appropriate hardware (please consult support), an installed option key, and the latest firmware release.

Restrictions

The serial trigger only operates on one protocol at a time. It is therefore impossible to express a condition such as "trigger on CAN frames with ID = 0x456 followed by LIN packet with Adress 0xEBC."

Linking Trigger and Decoder

A quick way to set up a serial trigger is to link it to a decoder by checking the **Link to Trigger** ("On") box on the Serial Decode dialog. Linking trigger and decoder allows you to configure the trigger with the exact same values that are used for decoding the signal (in particular the bit rate), saving the extra effort needed to re-enter values on the serial trigger set up dialogs.

While the decoder and the trigger have distinct sets of controls, when the link is active, a change to the bit rate in the decoder will immediately propagate to the trigger and vice-versa.

USB2 Trigger Setup



Note: The USB2 serial trigger is not supported on LabMaster and WaveMaster oscilloscopes, or their SDA/DDA equivalents.

USB2 triggers fall into four basic types of Packet, Protocol Error, Transaction, and Bus Event. The setups for each type of trigger differ significantly and so are described in separate procedures below. Only the first group of settings describing the source Setup is utilized by all the trigger types.

To access the serial trigger dialogs:

- Touch the Trigger descriptor box, or choose **Trigger > Trigger Setup** from the Menu Bar.
- Touch the **Serial** Type button, and the **USB2** Standard button.

Then, working from left to right, make the desired selections from the USB2 - <trigger type> dialogs.

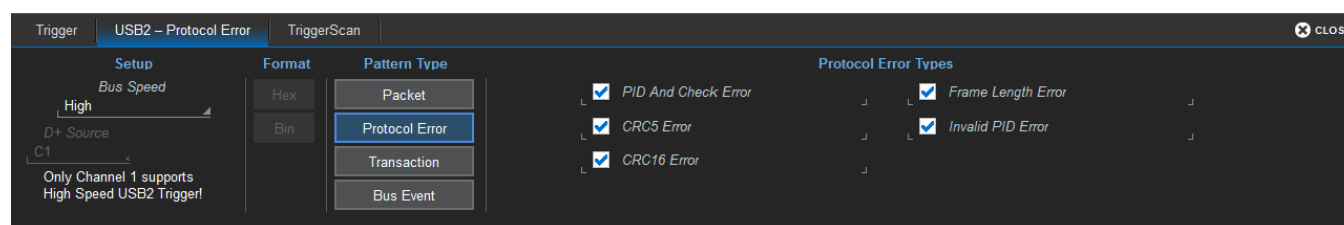
Setup

Bus Speed - Select the signaling rate of the trigger source, then make any other entries required by your selection:

- **Low** and **Full** - Enter the D+ (Data positive) and D- (Data negative) source channels and waveform crossing levels. These may be the same or different channels depending on whether your trigger setup utilizes a differential probe or two single-ended probes.
- **High** - High speed USB triggering is only available on Channel 4, which is preselected for you when you choose High. Be sure your input is connected to this channel.

Error Trigger

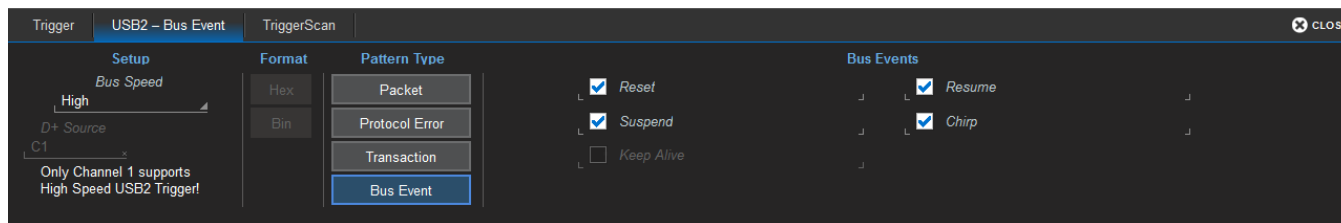
This will trigger upon encountering the first error of the type(s) you have selected.



1. Choose Pattern Type **Protocol Error**.
2. Select as many of the error types as you wish.

Bus Event Trigger

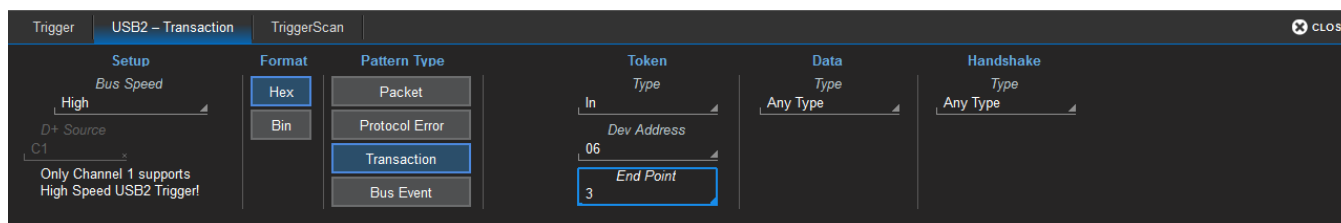
This will trigger upon encountering the first of the selected type(s) of bus events. Options vary depending on the bus speed.



1. Choose Pattern Type **Bus Event**.
2. Select as many of the event types as you wish.

Transaction Trigger

This will trigger upon finding transactions that meet all the Token, Data, and Handshake packet criteria. Options vary depending on the bus speed. The principal difference between a Transaction trigger and a Packet trigger is that for a Transaction trigger, the specified packet types simply need to be found in the transaction, whereas for a Packet trigger, the packet types and values must meet the criteria.



1. Choose Pattern Type **Transaction**.
2. Choose to enter values in **Binary** or **Hexadecimal (Hex)** format. The selection propagates throughout the entire trigger setup. Toggling between formats does not result in loss of information, but will transform the appearance of values.
3. Select the **Token Type**:
 - Any Type triggers on any Token packet that meets the remaining criteria.
 - In, Out, Setup, and Ping (High Speed only) trigger only when one of these Token types meet the remaining criteria.
4. Enter the Token **DevAddress** (Device Address) and **Endpoint** values.
5. To add a Data criteria to the trigger, choose the **Data Type**, otherwise leave Any Type selected. Not Exist will trigger when there is no Data packet in the transaction.
6. To add a Handshake criteria to the trigger, choose the **Handshake Type**, otherwise leave Any Type selected. Not Exist will trigger when there is no Handshake packet in the transaction.

Packet Trigger

This type of trigger can be set on complex patterns in any packet field—token, data, or handshake—or on a 1-16 byte user-defined data pattern. Different controls for specifying the pattern will be activated depending on the Category selection.



Tip: When entering patterns, use "X" as a wildcard for any positions where you don't care about the value.

Trigger on Any USB2 Packet

Choose Pattern Type **Packet** and Category **Any Sync**.

Trigger on Token Pattern



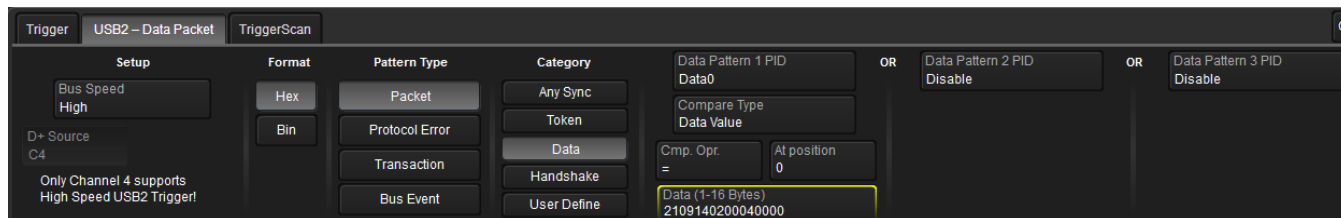
Note: Pattern types vary by bus speed. Enter values for any additional controls that appear. If you only want to define one or two patterns, choose Disable in Token 2 or 3.

The screenshot shows the configuration interface for a USB2 - Token Packet trigger. The interface is divided into several sections:

- Trigger:** USB2 - Token Packet, TriggerScan
- Setup:** Bus Speed: High; D+ Source: C4; Note: Only Channel 4 supports High Speed USB2 Trigger!
- Format:** Hex (selected), Bin
- Pattern Type:** Packet (selected), Protocol Error, Transaction, Bus Event
- Category:** Any Sync (selected), Token, Data, Handshake, User Define
- Token 1 PID Split:** Hub Address: 01, SC: 0, Port: 04; S: X, E: X, ET: 0
- Token 2 PID:** Disable
- Token 3 PID:** Disable

1. Choose Pattern Type **Packet** and Category **Token**.
2. Choose to enter values in **Binary** or **Hexadecimal (Hex)** format. The selection propagates throughout the entire trigger setup. Toggling between formats does not result in loss of information, but will transform the appearance of values.
3. Use the three **Token # PID** controls to define the first, second, and third token patterns that will produce a trigger.
 - Any (Token 1 PID) triggers upon finding any token packet. If Any is selected, the remaining Token # PID controls are disabled, as the first token encountered will cause a trigger.
 - In, Out, Setup, and Ping trigger when a token of the that type contains the values entered in **DevAddress** and **EndPoint**.
 - Split (High Speed only) triggers when a split contains the values entered in **Hub Address**, **SC**, **Port number** and **S**, **E**, or **ET**.
 - SOF (Full and High Speed only) triggers when Start of Frame value matches the **Frame Number** entered.
 - Preamble (Full Speed only) triggers when the token contains a preamble.

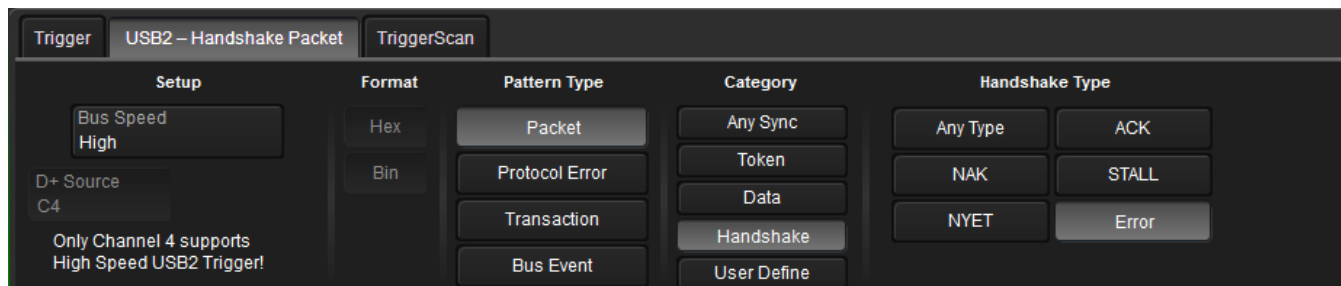
Trigger on Data Pattern



1. Choose Pattern Type **Packet** and Category **Data**.
2. Choose to enter values in **Binary** or **Hexadecimal (Hex)** format. The selection propagates throughout the entire trigger setup. Toggling between formats does not result in loss of information, but will transform the appearance of values.
3. Use the three **Data Pattern # PID** controls to define up-to-three data patterns that will produce a trigger.
 - Any Type triggers upon finding any data packet that satisfies the remaining criteria.
 - Data0, Data1, Data2, and MData will trigger only when that type of data packet matches the criteria.
4. In **Compare Type**, choose to compare Data Values or Data Lengths to find a match.
5. If you chose to compare lengths, enter the **Cmp. Opr.** (operator) and **Length** (in bytes) that define the trigger criteria (e.g., Trigger if Data0 length is greater than 4 bytes).

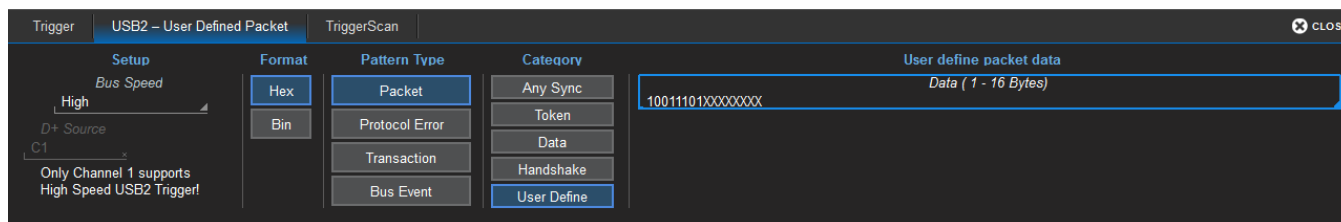
If you chose to compare values, enter the **Cmp. Opr.** (operator), **At position**, and a reference **Data (1-16 Bytes)** pattern that define the trigger criteria (e.g., Trigger if the value at position x in Data0 is equal to y).

Trigger on Handshake Type



1. Choose Pattern Type **Packet** and Category **Handshake**.
2. Select the handshake type that will cause a trigger: **Any Type**, **ACK**, **NAK**, **STALL**, **NYET** (High Speed only), or **Error** (High Speed only).

Trigger on User-defined Pattern



1. Choose Pattern Type **Packet** and Category **User Define**.
2. Choose to enter values in **Binary** or **Hexadecimal (Hex)** format. The selection propagates throughout the entire trigger setup. Toggling between formats does not result in loss of information, but will transform the appearance of values.
3. In **Data (1-16 Bytes)**, enter the pattern that will cause a trigger.

Using the Decoder with the Trigger

A key feature of Teledyne LeCroy trigger and decode options is the integration of the decoder functionality with the trigger. While you may not be interested in the decoded data per se, using the decoded waveform can help with understanding and tuning the trigger.

Stop and Look

Decoding with repetitive triggers can be very dynamic. Stop the acquisition and use the decoder tools such as [Search](#), or oscilloscope tools such as TriggerScan, to inspect the waveform for events of interest. Touch and drag the paused trace to show time pre- or post-trigger.

Optimize the Grid

The initial decoding may be very compressed and impossible to read. Try the following:

- Increase the height of the trace by *decreasing* the gain setting (V/Div) of the decoder source channel. This causes the trace to occupy more of the available grid.
- Change your Display settings to turn off unnecessary grids. The Auto Grid feature automatically closes unused grids. On many oscilloscopes, you can manually move traces to consolidate grids.
- Close setup dialogs.

Use Zoom

The default trigger point is at zero (center), marked by a small triangle of the same color as the input channel at the bottom of the grid. Zoom small areas around the trigger point. The zoom will automatically expand to fit the width of the screen on a new grid. This will help you to see that your trigger is occurring on the bits you specified.

If you drag a trace too far left or right of the trigger point, the message decoding may disappear from the grid. You can prevent "losing" the decode by creating a zoom of whatever portion of the decode interests you. The zoom trace will not disappear when dragged and will show much more detail.

Saving Trigger Data

The message decoding and the result table are dynamic and will continue to change as long as there are new trigger events. As there may be many trigger events in long acquisitions or repetitive waveforms, it can be difficult (if not impossible) to actually read the results on screen unless you stop the acquisition. You can preserve data concurrent with the trigger by using the **AutoSave** feature.

- AutoSave Waveform creates a .trc file that copies the waveform at each trigger point. These files can be recalled to the oscilloscope for later viewing. Choose **File > Save Waveform** and an Auto Save setting of **Wrap** (overwrite when drive full) or **Fill** (stop when drive full). The files are saved in D:\Waveforms.
- AutoSave Table creates a .csv file of the result table data at each trigger point. Choose **File > Save Table** and an Auto Save setting of **Wrap** or **Fill**. The files are saved in D:\Tables.



Caution: If you have frequent triggers, it is possible you will eventually run out of hard drive space. Choose Wrap only if you're not concerned about files persisting on the instrument. If you choose Fill, plan to periodically delete or move files out of the directory.

Measure/Graph

The installation of the Measure/Graph package (included with any -DME or -TDME option) adds a set of measurements and plots designed for serial data analysis to the oscilloscope's standard measurement capabilities. Measurements can be quickly applied without having to leave the waveform or tabular views of the decoding.



Note: This functionality was formerly offered as part of -TDM options and the ProtoBus MAG software option. The features described in this section should be present if you have either of these installed on your oscilloscope.

Serial Data Measurements

These measurements designed for debugging serial data streams can be applied to the decoded waveform. Measurements appear in a tabular readout below the grid (the same as for any other measurements) and are in addition to the [result table](#) that shows the decoded data. You can set up as many measurements as your oscilloscope has parameter locations.



Note: Measurements appear in the Serial Decode sub-menu of the Measure Setup menu and may have slightly different names. For example, the CAN sub-menu has measurements for CANtoValue instead of MsgToValue, etc. The measurements are the same.

Measurement	Filters	Description
AnalogToMsg	ID, Data, Analog	Computes time from crossing threshold on an analog signal to start of first message that meets conditions. If the message condition precedes the analog condition, no measurement is performed.
BusLoad	ID, Data	Computes the load of selected messages on the bus (as a percent).
ColToValue	Column	Extracts the data in a single column of the result table to a measurement parameter location, with no transformation of value.
DeltaMsg	ID, Data	Computes time difference between two messages on a single decoded line.
MsgBitrate	ID, Data	Computes the bitrate of selected messages within the decoded stream.
MsgToAnalog	ID, Data, Analog	Computes time from start of first message that meets conditions to crossing threshold on an analog signal. If the analog condition precedes the message condition, no measurement is performed.
MsgToMsg	ID, Data	Computes time from start of first message that meets conditions to start of the next message that meets conditions.
MsgToValue	ID, Value	Extracts a selected portion of the data to a measurement parameter location, with optional conversion of value. Data may be selected by ID and/or data field position.
NumMessages	ID, Data	Computes the total number of messages in the decoding that meet conditions.
Time@Msg	ID, Data	Computes time from trigger to start of each message that meets conditions.

Graphing Measurements

The Measure/Graph package include simplified methods for plotting measurement values as:

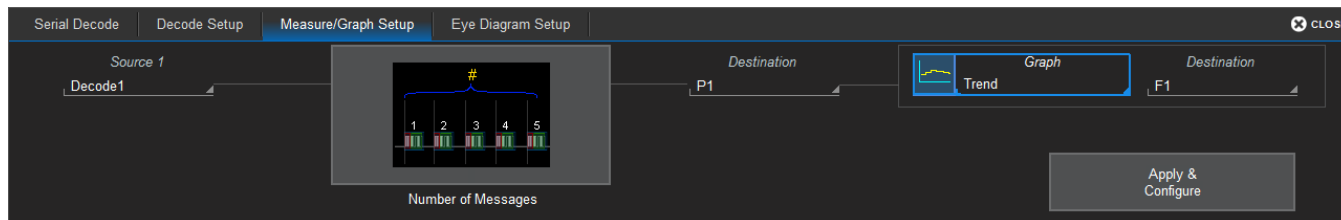
- **Histogram** - a bar chart of the number of data points that fall into statistically significant intervals or bins. Bar height relates to the frequency at which data points fall into each interval/bin. Histogram is helpful to understand the modality of a parameter and to debug excessive variation.
- **Trend** - a plot of the evolution of a parameter over time. The graph's vertical axis is the value of the parameter; its horizontal axis is the order in which the values were acquired. Trending data can be accumulated over many acquisitions. It is analogous to a chart recorder.
- **Track** - a time-correlated accumulation of values for a single acquisition. Tracks are time synchronous and clear with each new acquisition. Track can be used to plot data values and compare them to a corresponding analog signal, or to observe changes in timing. A parameter tracked over a long acquisition could provide information about the modulation of the parameter.

These plots effectively perform a digital-to-analog conversion that can be viewed right next to the decoded waveform.

To graph a measurement, just select the plot type from the Measure/Graph dialog when setting up the measurement. All plots are created as Math functions that open along side the decoding in a separate grid.

Measure/Graph Setup Dialog

Use the Measure/Graph Setup dialog to apply serial data measurement parameters to the decoded waveform and simultaneously graph the results. This dialog appears behind the Decode Setup dialog and is active when measurements are supported.



1. Select the **Measurement** to apply and the **Destination** parameter (Px) to which to assign it.
2. The active decoder is preselected in **Source 1**, indicating the measurement will be applied to the decoder results; change it if necessary. If the measurement requires it, also select an appropriate Source 2 (such as an analog waveform for comparison).
3. Optionally:
 - Touch [Graph](#) to select a plot type. Also select a **Destination** function (Fx) for the plot.
 - Touch **Apply & Configure** to set a filter, gate or other qualifiers on the measurement.

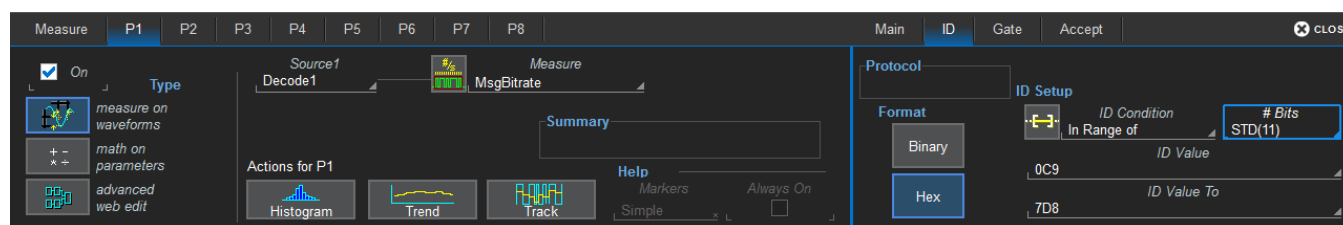
Filtering Measurements

Certain serial decode measurements can be filtered to include only the results from specified IDs or specific data patterns. As with all measurements, you can set a gate to restrict measurements to a horizontal range of the grid corresponding to a specific time segment of the acquisition.

After creating a measurement on the Measure/Graph Setup dialog, touch **Apply&Configure**. The touch screen display will switch to the standard Measure setup dialogs for the parameter you selected. Set filter conditions on the right-hand subdialogs that appear next to the Px dialogs.

ID Filter

This filter restricts the measurement to only frames/packets with a specific ID value. Settings on this dialog may change depending on the protocol.



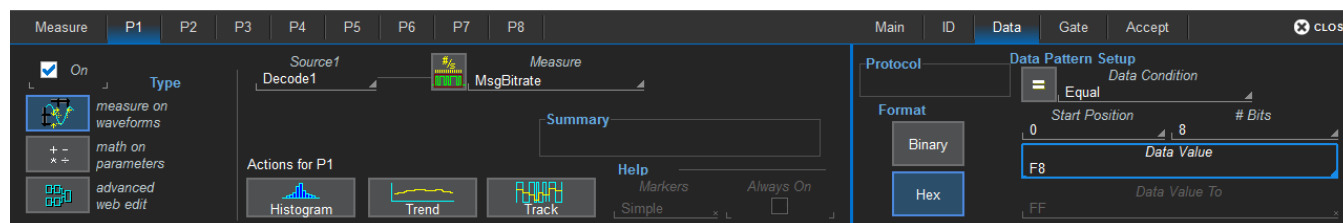
1. On the Main subdialog, choose to **Filter** by **ID** or **ID + Data**.
2. On the **ID subdialog**, choose to enter the ID in **Binary** or **Hex**(adecimal) format.
3. If the field appears, select the **# Bits** used to define the frame ID. (This will change the ID Value field length.)
4. Using the **ID Condition** and **ID Value** controls, create a condition statement that describes the IDs you want included in the measurement. To set a range of values, also enter the **ID Value To**.



Tip: On the value entry pop-up: use the arrow keys to position the cursor; use Back to clear the previous character (like Backspace); use Clear to clear all characters.

Data Filter

This restricts measurements to only frames containing extracted data that matches the filter condition. It can be combined with a Frame ID filter by choosing **ID+Data** on the Main subdialog.



Use the same procedure as above to create a condition describing the **Data Value(s)** to include in the measurement. Use "X" as a wild card ("Don't Care") in any position where the value doesn't matter.

Optionally, enter a **Start Position** within the data field byte to begin seeking the pattern, and the **# Bits** in the data pattern. The remaining data fields positions will autofill with "X".



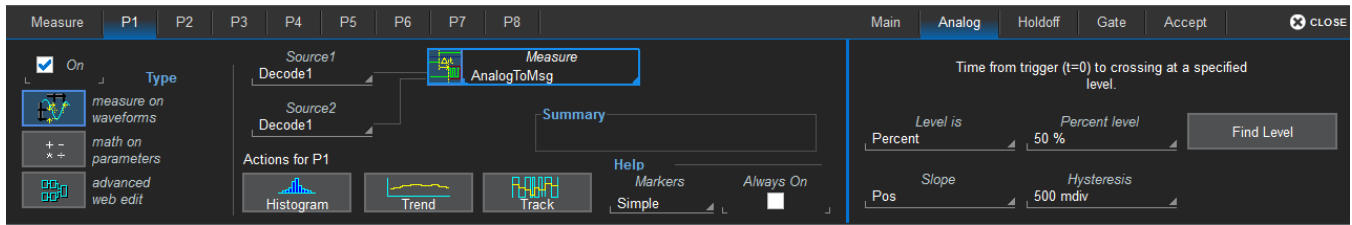
Note: For MsgtoMsg measurements, the data condition is entered twice: first for the Start Message and then for the End Message. The measurement computes the time to find a match to each set of conditions.

Analog Settings

The measurements AnalogToMsg and MsgToAnalog allow you to use crossing level and slope to define the event in the Analog waveform that is to be used as the reference for the measurement.

As with the decoder, Level may be set as a percentage of amplitude (default), or as an absolute voltage level by changing **Level Is** to Absolute. You can also use **Find Level** to allow the oscilloscope to set the level to the mean top-base amplitude.

A **Slope** and **Hysteresis** selection is also offered. The width of the Hysteresis band is specified in milli-divisions. See [Setting Level and Hysteresis](#) for more information on using these controls.



Eye Diagram Tests

The -DME and -TDME options provide easy [eye diagram setup](#) and eye mask testing.

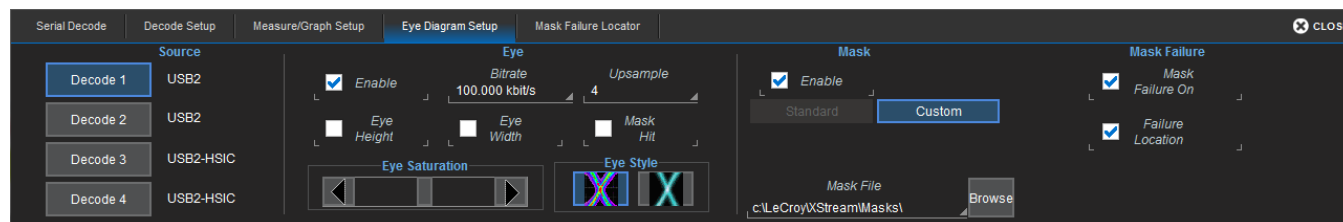
Eye diagrams are a key component of serial data analysis. They are used both quantitatively and qualitatively to understand the quality of the signal communications path. Signal integrity effects such as intersymbol interference, loss, crosstalk and EMI can be identified by viewing eye diagrams, such that the eye is typically viewed prior to performing any further analysis.

Each pixel in the eye takes on a color that indicates how frequently a signal has passed through the time and voltage specified for that pixel. The eye diagram shows all values a digital signal takes on during a bit period. A bit period (also referred to as unit interval, or UI) is defined by the data clock, whether explicit or extrapolated depending on the protocol.

Eye diagrams show the acquired signal that is currently being shown on the decoder result table. They are not persistent, as are eye diagrams generated in some other serial data analysis software; the eye will change from one acquisition to the next and when the result table is filtered. Our recommended approach for using the eye diagrams is to:

- Make single shot acquisitions with decoder and eye diagram enabled to check that both are working correctly.
- Run a normal acquisition with Mask Testing and Stop On Failure enabled in the [Mask Failure Locator](#), or with a Pass/Fail test set on one of the eye parameters.

Eye Diagram Setup Dialog



Create Eye Diagram



Open the **Eye Diagram Setup** dialog and select the **Decode** for which to create an eye diagram.

Under Eye, check **Enable** to display the eye diagram.

The **Bitrate** is automatically read from the decoder setup. This value is linked to the decoder bit rate setting, and changing it in either place will update both settings.

The **Upsample** factor increases the number of sample points used to compose the eye diagram. Increase from 1 to a higher number (e.g. 5) to fill in gaps. Gaps can occur when the bitrate is extremely close to a submultiple of the sampling rate, such that the sampling of the waveform does not move throughout the entire unit interval. Gaps can also occur when using a record length that does not sample a sufficiently large number of unit intervals.

The **Eye Style** may utilize color-graded or analog persistence:

- With **color-graded** persistence , pixels are given a color based on the pixel's relative population and the selected Eye Saturation. The color palette ranges from violet to red.
- With **analog** persistence , the color used mimicks the relative intensity that would be seen on an analog oscilloscope.

Use the **Eye Saturation** slider to adjust the color grading or intensity. Slide to the left to reduce the threshold required to reach saturation.

Choose to display the **Eye Height**, **Eye Width**, or **Mask Hit(s)** measurement parameters. These are added to the Measure table in the first open parameter slots.

Eye Mask Test

Under Mask, check **Enable** to turn on eye mask testing.

Select to use either a **Standard** or **Custom** mask, then either select the **Standard Mask** or **Browse** to and select your custom **Mask File**.



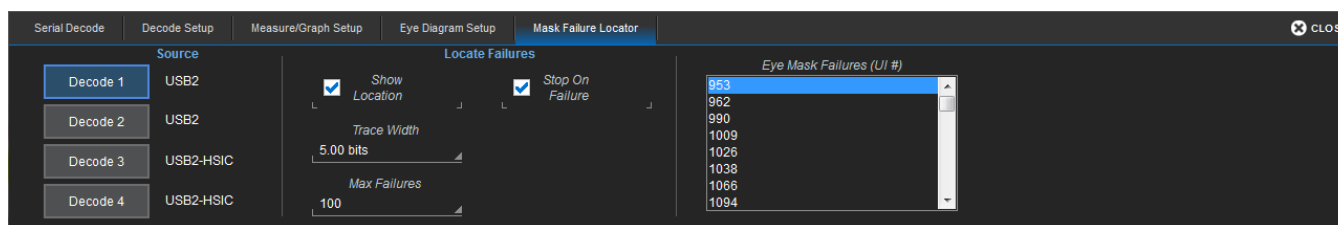
Tip: Masks previously created on the instrument are stored in D:\Masks. For ease of selection, copy other .msk files to this location.

Check **Mask Failure On** to mark the parts of the eye diagram that fail the mask test. Mask violations appear as red failure indicators where the eye diagram intersects the mask.

Check **Failure Location** to display the [Mask Failure Locator](#) dialog.

Mask Failure Locator Dialog

Use this dialog to quickly search the acquisition for eye diagram mask test failures.



In **Trace Width**, enter the number of UIs surrounding the mask violation to display as "padding."

Check **Stop On Failure** to stop acquisition whenever an eye mask failure occurs.

Enter the **Max Failures** to retain in the Eye Mask Failure list.

Select from the **Eye Mask Failure** list to mark and zoom to the location of that failure. Yellow circles appear over the red failure indicators to show the location of the failure.

Technical Support

Live Support

Registered users can contact their local Teledyne LeCroy service center at the number listed on our website.

You can also submit Technical Support requests via the website at:

teledynelecroy.com/support/techhelp

Resources

Teledyne LeCroy publishes a free Technical Library on its website. Manuals, tutorials, application notes, white papers, and videos are available to help you get the most out of your Teledyne LeCroy products.

The Datasheet published on the product page contains the detailed product specifications.

Service Centers

For a complete list of offices by country, including our sales & distribution partners, visit:

teledynelecroy.com/support/contact

Teledyne LeCroy
700 Chestnut Ridge Road
Chestnut Ridge, NY, 10977, USA
teledynelecroy.com

Sales and Service:

Ph: 800-553-2769 / 845-425-2000
FAX: 845-578-5985
contact.corp@teledynelecroy.com

Support:

Ph: 800-553-2769
customersupport@teledynelecroy.com



700 Chestnut Ridge Road
Chestnut Ridge, NY 10977
USA

teledynelecroy.com