# NI ELVIS III Using Your Instruments

2023-09-18

# Contents

# Using the Instruments

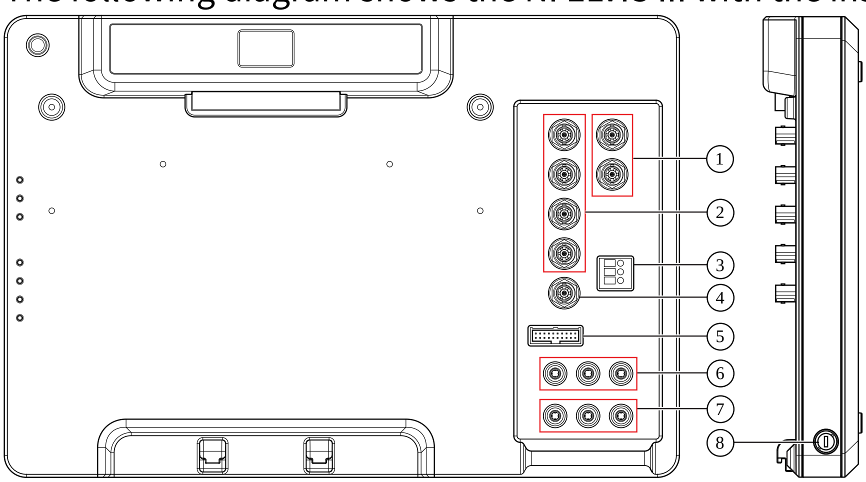
The NI ELVIS III combines a suite of instruments that offers the same functionality as a set of stand-alone benchtop instruments commonly used in electronics laboratories.

You can use the Soft Front Panels (SFPs) to interact with the instruments on the NI ELVIS III without programming. You can also program the instruments after you install the NI ELVIS III Software Bundle. The NI ELVIS III Software Bundle includes the LabVIEW ELVIS III Toolkit, which provides the Instruments and Control I/O palettes that contain VIs for controlling the instrumentation I/O and control I/O channels respectively on the NI ELVIS III.

The instruments leverage two types of I/O on the NI ELVIS III: Instrumentation I/O and Control I/O.

## Instrumentation I/O

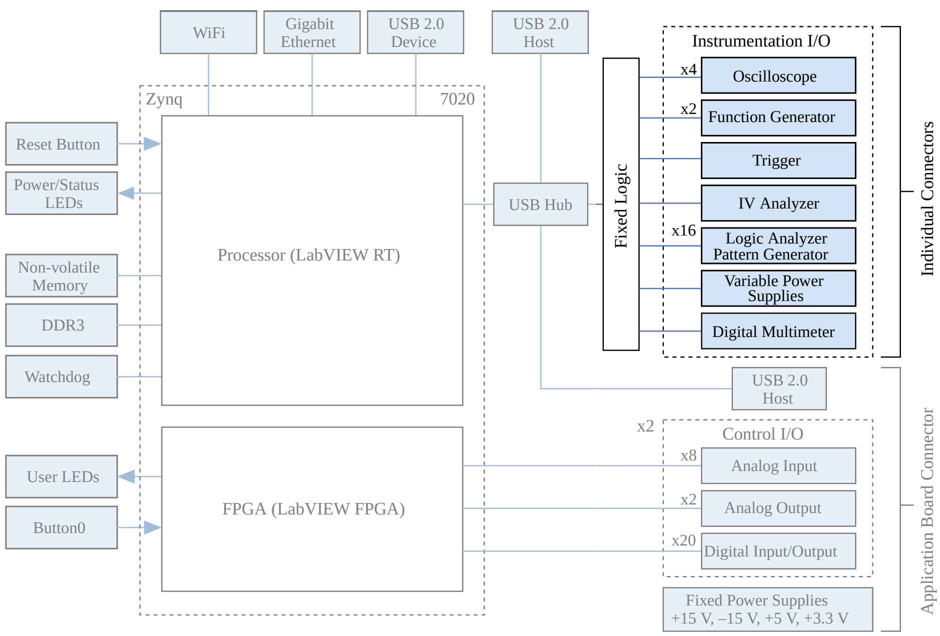
The following diagram shows the NI ELVIS III with the instrumentation I/O labeled.




1. Function Generator
2. Oscilloscope
3. IV Analyzer
4. Trigger
5. Logic Analyzer/Pattern Generator

6. Variable Power Supplies
7. Digital Multimeter
8. Digital Multimeter Fuse

The following diagram shows the NI ELVIS III hardware architecture with the instrumentation I/O highlighted.




You may encounter resource conflicts if you run certain instrument circuitry simultaneously. The following table shows all the instrumentation resource conflicts. To use the information in the following table, find the instrument you want to use in the left column. The rows from that left column lists all the functions that are resource conflicts. If the intersecting box contains an X, you cannot use those functions simultaneously; otherwise you may encounter resource conflicts. If the intersecting box is empty, you can use those functions simultaneously.

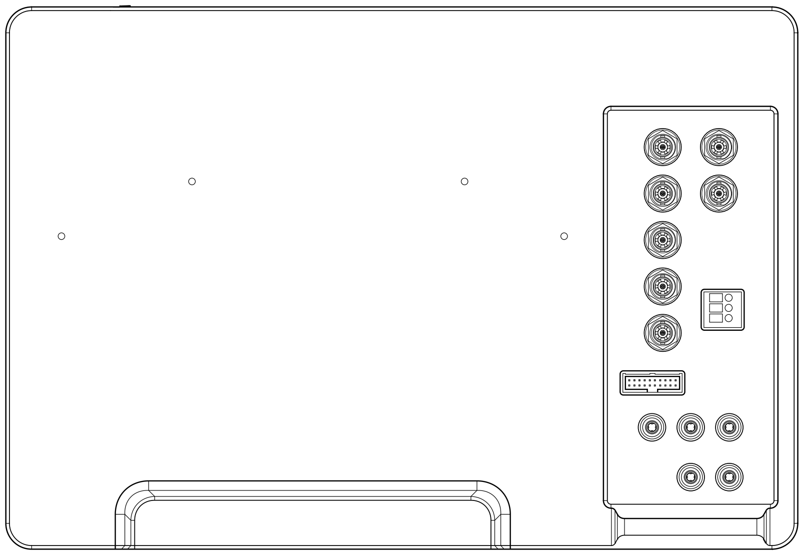
**Table 1.** Instrumentation Resource Conflicts

| | Oscilloscope | Function Generator | Digital Multimeter (Capacita | Digital Multimeter (Other) | Variable Power Supplies | Logic Analyzer and Pattern | IV Analyzer | Bode Analyzer |
|---|---|---|---|---|---|---|---|---|

| | | | nce/ Inductance) | | | Generator | | |
|---|---|---|---|---|---|---|---|---|
| Oscilloscope | - | | X | | | | X | X |
| Function Generator | | - | X | | | | X | X |
| Digital Multimeter (Capacitance/ Inductance) | X | X | - | | | | X | X |
| Digital Multimeter (Other) | | | | - | | | | |
| Variable Power Supplies | | | | | - | | | |
| Logic Analyzer and Pattern Generator | | | | | | - | | |
| IV Analyzer | X | X | X | | | | - | X |
| Bode Analyzer | X | X | X | | | | X | - |

## Control I/O

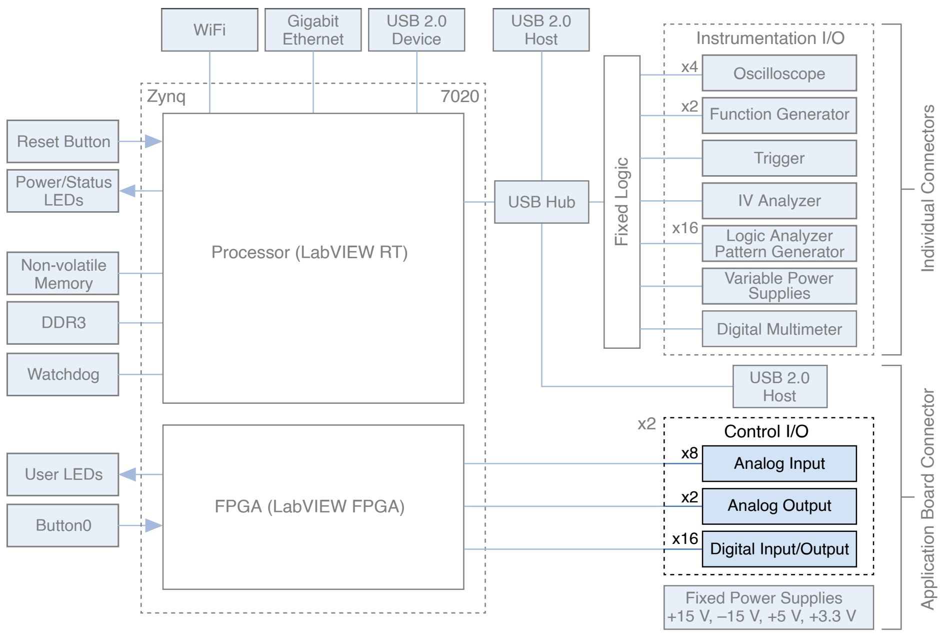
The following diagram shows the NI ELVIS III with most control I/O terminals labeled. The labeled terminals are used by the instruments.



1. Analog input terminals used by the Data Logger
2. Analog output terminals used by the Data Generator
3. Digital input/output terminals used by the Digital I/O

**Note** You can use the NI ELVIS III Prototyping Board to access the control I/O terminals.

The following diagram shows the NI ELVIS III hardware architecture with the control I/O highlighted.




**Note** The Digital I/O uses only 16 out of the 20 digital input/output channels.

Refer to Programming the Control I/O to learn how to program the control I/O with LabVIEW. When running the instruments that use the control I/O channels, you may encounter resource conflicts in one of the following three conditions:

- LabVIEW is running a custom FPGA bitfile.
- LabVIEW and Measurements Live are running different versions of the Control I/O VIs.
- LabVIEW and Measurements Live are using the same control I/O channel(s) simultaneously.

You can close the related VIs in LabVIEW to resolve the resource conflict.

## Related information:

- Installing the NI ELVIS III Software Bundle
- LabVIEW ELVIS III Toolkit Help
- Launching the Soft Front Panels

## Simulating the Instruments

Learn to use the major instruments on the NI ELVIS III device before connecting to a real device.

The following two typical pairs of instruments are pre-connected and configured for you to simulate, covering the common use cases in labs, such as generating, acquiring, and analyzing waveforms.

- Oscilloscope & Function and Arbitrary Waveform Generator
- Variable Power Supply & Digital Multimeter

Complete the following steps to simulate the instruments.

1. Go directly to measurementslive.ni.com.
2. Click **DEVICE SIMULATION** to connect Measurements Live to a simulated NI ELVIS III device.
3. On the top right corner, choose a pair of instruments to simulate by clicking a checkbox.
   You launch the corresponding Soft Front Panels (SFPs) and the SFPs run immediately.
4. Observe the measurements on the SFPs.
5. Adjust the configuration options as needed.

Some configuration options are dimmed because they do not work on a simulated device. Refer to Channel Connections and Supported Features for Simulated Instruments to learn what tasks you can perform by simulating the instruments. To experience full functionality of the NI ELVIS III, connect to a real device!

## Channel Connections and Supported Features for Simulated Instruments

Each pair of instruments is pre-connected and configured for you to simulate.

Learn the channel connections and supported features for each pair of instruments in the following table.

| Instrument pair | Channel connection | Supported feature |
| --- | --- | --- |
| Oscilloscope & Function Generator and Arbitrary Waveform Generator (FGen) | ▪ The Oscilloscope channel (CH) 1 is connected to the FGen CH1.<br>▪ The Oscilloscope CH 2 is connected to the FGen CH 2. | Oscilloscope:<br>▪ Acquire and display the waveforms with most functions available.<br>▪ Perform FFT or math operations on the waveform.<br>▪ Compare waveforms using the reference channels.<br>FGen:<br>▪ Generate standard waveforms. |
| Variable Power Supply (VPS) & Digital Multimeter (DMM) | ▪ The VPS "+" terminal is connected to the DMM VΩ terminal.<br>▪ The VPS ground terminal is connected to the DMM COM terminal. | VPS:<br>▪ Provide positive DC voltages.<br>DMM:<br>▪ Measure positive DC voltages. |

# Choosing the Instruments

You can choose from various instruments based on the kind of measurement you want to take.

## Oscilloscope vs Data Logger

Refer to the following table to determine which instrument you want to use to measure analog signals.

| | Oscilloscope | Data Logger |
|---|---|---|
| Use cases | ▪ Acquire, display, analyze, and record waveforms.<br>▪ Perform fast Fourier transform or mathematical operations on the signal.<br>▪ Compare the acquired signal with the signal on a reference channel. | Acquire, display, record, and store analog signals to a file. |
| Hardware terminals | Instrumentation I/O | Control I/O |
| Number of channels | 4 | 24 (single-ended mode and differential mode) |
| Input range | ±50 V | ±10 V, ±5 V, ±2 V, ±1 V |
| Accuracy | 2% of input + 1% of full scale | ▪ Typical condition (25 °C ± 5 °C): 0.064% of reading + 0.004% of range<br>▪ Maximum (10 °C to 35 °C): 0.397% of reading + 0.054% of range |
| Trigger types | Two types of hardware triggers:<br>▪ Analog edge<br>▪ Digital edge | Various types of software triggers:<br>▪ Start trigger: Immediate, Delay, Level, and Range.<br>▪ Stop trigger: No trigger, Duration, Level, Range, and Number of samples. |
| Maximum sample rate | ▪ 400 MS/s with repetitive sampling enabled | ▪ Single channel: 100 kS/s<br>▪ Multichannel: 100 kS/s (aggregate) |

| | Oscilloscope | Data Logger |
|---|---|---|
| | ▪ 100 MS/s without repetitive sampling enabled | |
| Continuously acquire data? | No | Yes |
| Automatically save data to the local computer? | No | Yes |
| Export data? | Yes | Yes |

## Function and Arbitrary Waveform Generator vs Data Generator

Refer to the following table to determine which instrument you want to use to generate analog signals.

| | Function and Arbitrary Waveform Generator | Data Generator |
|---|---|---|
| Use cases | Generate standard or arbitrary waveforms. | Generate standard waveforms. |
| Hardware terminals | Instrumentation I/O | Control I/O |
| Number of channels | 2 | 4 |
| Maximum frequency | 15 MHz | 50 kHz |
| Channel mode | Static, Sweep, and Custom | Static |
| Can generate a phase shift? | Yes | Yes |

### Related concepts:

- Oscilloscope
- Data Logger
- Function and Arbitrary Waveform Generator
- Data Generator

# Oscilloscope

The Oscilloscope is a standard digital storage oscilloscope.
You can use the Oscilloscope to complete the following tasks:

- Acquire, display, and analyze the input signal.

- Perform fast Fourier transform or mathematical operations on the signal.
- Compare the acquired signal with the signal on a reference channel.

Find an introductory video about using the instrument on ni.com.
Find out more about the Oscilloscope in the following topics:

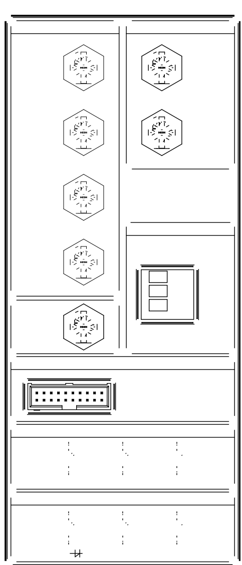
- Connecting Signals to the Oscilloscope
- Measuring a Waveform
- Configuration Options (Oscilloscope)
- TDMS File Format (Oscilloscope)
- Programming the Oscilloscope
- Oscilloscope Concepts
- Oscilloscope Specifications

**Related reference:**

- Choosing the Instruments

## Connecting Signals to the Oscilloscope

The NI ELVIS III instrumentation panel provides four channels for oscilloscope input signals.
The Oscilloscope uses dedicated analog-to-digital converters to acquire up to 100 MS/s per channel with 14-bit resolution.

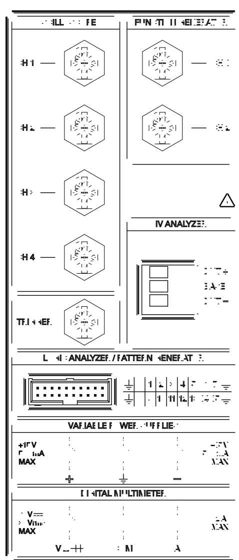


## Measuring a Waveform

This example illustrates how to use the Function and Arbitrary Waveform Generator (FGen/Arb) to generate a waveform and use the Oscilloscope to measure the waveform.

The procedure for measuring other types of waveforms is similar. Refer to the Configuration Options (Oscilloscope) section to learn about other functionalities of the Oscilloscope.

1. Connect a BNC - BNC cable between Oscilloscope CH 1 and Function Generator CH1 on the NI ELVIS III, as shown in the following diagram:

   Refer to Connecting Signals to the Oscilloscope for more details about connecting to the Oscilloscope.
2. Launch the FGen/Arb Soft Front Panel (SFP).
3. Configure the FGen/Arb to generate a sine wave on **Channel 1**.
4. Click **Run** to start generating a sine wave.
5. Launch the Oscilloscope Soft Front Panel (SFP).
6. Configure the Oscilloscope to measure the input signal on **Channel 1**.
7. Click **Run** to start measuring the input signal.
   A sine wave appears in the display window.

8. Adjust the display settings as needed. You can also select **Auto setup** if you want the Oscilloscope to choose some settings automatically.
9. If necessary, adjust the trigger settings to stabilize the waveform in the display window.

> **Note** You may hear clicking sounds from the NI ELVIS III while operating the SFP. This is expected behavior and is due to the switching of internal relays.

**Related information:**

- Launching the Soft Front Panels

## Configuration Options (Oscilloscope)

Use configuration options on the Oscilloscope Soft Front Panel (SFP) to control how to acquire, display, and analyze the waveforms. For example, you can adjust the sample rate.

- Toolbar Controls
- Trigger Settings
- Horizontal & Acquisition Settings
- Channels Settings
- Additional Channels Settings
- Display Window

> **Note** Click the expand arrow near the upper right corner of the SFP to expand the configuration pane if it is collapsed by default.

### Toolbar Controls (Oscilloscope)

Use the toolbar controls to perform common tasks for the Oscilloscope. The toolbar is located at the top of the SFP.

The layout of the controls may vary depending on the device from which you launch the SFP. Some controls are hidden by default when you resize the SFP. Click the ellipsis on the toolbar to access hidden controls.

| Control | Description |
|---|---|
| Drag and drop | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| Down triangle | Click to open the current SFP in a separate browser window. To restore it to the original window, close the SFP by clicking and launch it again. |
| Run/Stop | Click **Run** to start data acquisition. Click **Stop** to stop data acquisition. |
| Auto setup/Default | When you select **Auto setup**, the instrument automatically chooses some settings, including channel and trigger settings, that are best suited to the input signal. When you select **Default**, the settings fall back to default values. |
| Single | Takes a single-sweep measurement. |
| Auto | The Oscilloscope waits momentarily for any occurrence of trigger condition before automatically triggering. |
| Normal | The Oscilloscope waits indefinitely for any occurrence of trigger condition. |
| Force | Manually triggers the Oscilloscope. |
| Collaboration | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the instrument. For more information, refer to Collaborating with Your Team Members. |
| Collapse/Expand | Collapses or expands the configuration pane on the right side of the panel. |

## Trigger Settings (Oscilloscope)

You can configure the trigger by using the trigger settings.

| Setting | Description |
| --- | --- |
| **Type** | Type of trigger to start the acquisition. Refer to the Oscilloscope Concepts section to learn about different trigger types.<br>▪ **Analog edge**—A trigger occurs when the signal you select in **Source** crosses the trigger threshold that you define in **Level**.<br>▪ **Digital edge**—A trigger occurs on a rising edge of an external digital signal. To use a digital edge trigger, you must first connect a TTL-level signal to the Trigger channel on the NI ELVIS III. |
| **Source** | Source of the trigger that starts the data acquisition.<br>▪ **Channel N**—The instrument uses Channel N as the trigger signal. You can select **Channel N** only when **Type** is **Analog edge**.<br>▪ **TRIG**—The instrument uses the Trigger channel on the NI ELVIS III as the trigger signal. You can select **TRIG** only when **Type** is **Digital edge**. |
| **Slope** | Place at which a trigger occurs.<br>▪ **Rising**—The trigger occurs at a rising edge for digital signals, or a rising slope for analog signals.<br>▪ **Falling**—The trigger occurs at a falling slope for analog signals. This option is currently unavailable for digital signals.<br>▪ **Either**—The trigger occurs in either case. This option is currently unavailable for digital signals. |

| Setting | Description |
| --- | --- |
| **Level** | Voltage at which a trigger occurs. This setting is enabled only when **Type** is **Analog edge**. **Level** must be between -50 V and 50 V when the probe attenuation of the trigger signal is **1x** and between -500 V and 500 V when the probe attenuation of the trigger signal is **10x**.<br><br>**Note** The Oscilloscope uses hysteresis to reduce false triggering due to noise or jitter in the signal, by adding a window above or below **Level**. The hysteresis is 25% of **Volts/div**. You can adjust the value of hysteresis by configuring **Volts/div** in Channel Settings. Refer to Hysteresis to learn more about how hysteresis works. |
| **Set 50%** | Sets the trigger level as 50% of the peak-to-peak voltage of the measured signal. This setting is available only when **Type** is **Analog edge**. |
| **Acquisition delay** | Decides whether the instrument waits for a certain period of time after a trigger is detected before it starts sampling. Enable **Acquisition delay** when there is a delay between when the trigger occurs and when the desired data is generated, or when you need a high sampling rate for the desired acquisition window. |
| **Position** | The time between the trigger position and the middle of the display pane. The value of **Position** changes when you move the trigger horizontally. |

## Horizontal & Acquisition Settings (Oscilloscope)

You can configure the display and the data acquisition by using the **Horizontal & Acquisition** settings.

| Setting | Description |
|---|---|
| **Time/div** | The horizontal scale of the display. |
| **Shrink** | Contracts the trace by increasing **Time/div**. |
| **Expand** | Expands the trace by decreasing **Time/div**. |
| **Acquisition** | Filter you apply to the analog-to-digital (AD) conversion results.<br>- **Decimate**—The Oscilloscope records only the N AD conversion result, where N is predefined by the Oscilloscope.<br>- **Average**—The Oscilloscope calculates each sample as the average of the AD conversion results.<br>- **Min/Max**—For each two samples, the Oscilloscope calculates one as the minimum value of the conversion results and the other as the maximum value of the conversion results. In other words, the maximum value and the minimum value appear by turns. |
| **Repetitive sampling** | Specifies whether to enable repetitive sampling on the trigger signal.<br>- **Off**—Disables repetitive sampling on the trigger signal.<br>- **Auto**—The Oscilloscope automatically detects if all repetitive sampling conditions are met and decides whether to enable repetitive sampling on the trigger signal.<br>To enable repetitive sampling, you must configure the Oscilloscope as follows:<br>- The trigger type must be **Analog edge**, and the trigger signal must be enabled. |

| Setting | Description |
| --- | --- |
| | ▪ The trigger mode must be **Single** or **Normal**.<br>▪ The trigger level must be within the range of the input signal. When repetitive sampling is enabled, you cannot manually trigger the instrument, and the **Force** button is dimmed.<br>▪ **Sample rate** must be 100 MS/s. You can increase **Sample rate** by reducing **Time/div** or the absolute value of the **Position**. However, that is not applicable when **Acquisition delay** is enabled and the value of **Position** is negative.<br><br>**Note** The trigger signal must be periodic and with low noise. Otherwise, the graph on the display may be distorted. You can configure **Type**, **Mode**, **Acquisition delay**, and **Position** in Trigger Settings. Refer to Repetitive Sampling for more information. |

## Channels Settings (Oscilloscope)

You can configure channels by using the channels settings.

| Setting | Description | Operational Limits |
| --- | --- | --- |
| **Status** | Shows/Hides the channel's trace on graph. | - |
| **Volts/div** | Vertical scale of the display.<br><br>**Note** The Oscilloscope uses hysteresis to reduce false triggering due to noise or jitter in | - |

| Setting | Description | Operational Limits |
|---|---|---|
| | the signal. The hysteresis is 25% of **Volts/div**. By increasing **Volts/div**, you increase the value of hysteresis, which in turn stabilizes the signal in the display window. However, when hysteresis is too high, the Oscilloscope may never be triggered. Adjust **Volts/div** until you get a stable signal on the display. | |
| **Shrink** | Contracts the trace by increasing **Volts/div**. | - |
| **Expand** | Expands the trace by decreasing **Volts/div**. | - |
| **Vertical position** | Vertical positioning of the waveform on the graph. Use **Vertical position** or **Vertical offset** to move the waveform up and down the graph. The Oscilloscope does not apply **Vertical position** to the input signal. | Range: -100 V to 100 V |
| **Probe attenuation** | Amount of attenuation provided by the probe connected to the channel. Select **1X** if the probe does not attenuate the input signal. Select **10X** if the probe attenuates the input signal by a | - |

| Setting | Description | Operational Limits |
|---|---|---|
| | factor of ten. **Probe attenuation** must match the setting of the probe connected to the channel. | |
| **Coupling** | Specifies whether to remove the DC component from the signal. Select **DC** if you want to measure the entire signal. Select **AC** if you want to remove the DC component from the signal. | - |
| **Vertical offset** | Voltage on which the vertical range is centered. You can use **Vertical position** or **Vertical offset** to move the waveform up and down the graph. The Oscilloscope applies **Vertical offset** to the input signal. | Range: -25 V to 25 V when **Probe attenuation** is **1x** and -250 V to 250 V when **Probe attenuation** is **10x** |
| **Vertical range** | Indicates the current input range. You can change **Vertical range** by adjusting **Vertical offset**.<br><br>**Caution** Protect yourself with safety equipment when measuring high voltages. You must use a 10x probe to protect the hardware. | - |

## Additional Channels Settings (Oscilloscope)

Use configuration options in the **Additional Channels** section to perform a 16K-point fast Fourier transform (FFT) or mathematical operations on the signal, or to configure a reference channel.

FFT Channel

You can configure the fast Fourier transform (FFT) channel to perform a 16K-point fast Fourier transform on the signal.

Use the following settings to configure the FFT channel.

| Setting | Description |
|---|---|
| **Status** | Enables or disables fast Fourier transform on the source channel. |
| **Source channel** | Source channel on which you want to perform fast Fourier transform. |
| **Window** | Time-domain window to apply to the signal. |
| **Start frequency** | Frequency at the extreme left of the horizontal axis. |
| **Stop frequency** | Frequency at the extreme right of the horizontal axis. |

| Display | Description |
|---|---|
| **Center frequency** | Frequency that is at the center of the horizontal axis. |
| **Span** | Range between the start frequency and the stop frequency. |
| **Reference level** | Maximum expected power of the input signal. The Oscilloscope automatically sets **Reference level** based on the value of **Volts/Div**. |
| **Resolution bandwidth** | The horizontal scale. |
| **Record length** | Range of the X axis on the time domain graph. |

Math Channel

You can configure the mathematical channel to perform mathematical operations on the signal.

Use the following settings to configure the math channel.

| Setting | Description |
|---|---|
| **Status** | Enables or disables math operation on the source channels. |
| **Volts/div** | Vertical scale of the display. |
| **Shrink** | Contracts the trace by increasing **Volts/div**. |
| **Expand** | Expands the trace by decreasing **Volts/div**. |
| **Vertical position** | Vertical positioning of the waveform on the graph. Use **Vertical position** to move the waveform up and down the graph. The Oscilloscope does not apply **Vertical position** to the input signal. The range of **Vertical position** is -100 V to 100 V. |
| **Function** | Math operation you want to perform on the channels. |
| **Source A** | First source channel. |
| **Source B** | Second source channel. |

## Digital Filtering

The Oscilloscope may acquire noisy signals. Use digital filtering to remove unwanted parts of the captured waveform such as noise within the bandwidth of the instrument. You can set **Function** to **Filter** to enable digital filtering. Use the following settings to configure the math channel when digital filtering is enabled:

| Setting | Description |
|---|---|
| **Source** | Source channel on which you want to perform digital filtering. |
| **Typology** | The design type of the filter.<br>▪ **Butterworth filter**—Rolls off slowly around the cutoff frequency.<br>▪ **Chebyshev filter**—Rolls off moderately around the cutoff frequency. |

| Setting | Description |
|---|---|
| **Type** | The passband of the filter. Only low pass filter is available. |
| **Order** | The maximum delay, in samples, used in creating each output sample. The range of **Order** is between 1 and 16. |
| **Cutoff frequency** | A frequency characterizing a boundary between a passband and a stopband. The value of **Cutoff frequency** must be greater than 0. |

Reference Channel

Use the following settings to configure a reference channel.

| Setting | Description |
|---|---|
| **Status** | Enables or disables sourcing data from another channel. |
| **Volts/div** | Vertical scale of the display. |
| **Shrink** | Contracts the trace by increasing **Volts/div**. |
| **Expand** | Expands the trace by decreasing **Volts/div**. |
| **Vertical position** | Vertical positioning of the waveform on the graph. Use **Vertical position** to move the waveform up and down the graph. The Oscilloscope does not apply **Vertical position** to the input signal. The range of **Vertical position** is -100 V to 100 V. |
| **Mode** | Data sources for the reference channels.<br>• When you select **Load from file**, the Oscilloscope loads data from a data file. You specify the path to the file in **Source file**.<br>• When you select **Stream from Multisim Live**, the Oscilloscope loads simulated data streamed from Multisim Live. |

## Load from file

When you select **Load from file**, the Oscilloscope loads data from a data file. The following settings are available only when you select **Load from file** for **Mode**.

| Setting | Description |
| --- | --- |
| **Source file** | Path to the data file. You can load CSV files from Multisim and Multisim Live and TDMS files from LabVIEW. The TDMS files must contain only analog waveform data. You can also load CSV or TDMS files exported from the Oscilloscope. |
| **Source file channel** | Channel you want to import. If the source data file contains data of multiple channels, you can use **Source file channel** to select which channel of data you want to import to the Oscilloscope. |

## Stream from Multisim Live

When you select **Stream from Multisim Live**, the Oscilloscope loads simulated data streamed from Multisim Live. The following settings are available only when you select **Stream from Multisim Live** for **Mode**.

| Setting | Description |
| --- | --- |
| **Source circuit** | Name of the circuit you want to use as the source for the reference channel. |
| **Source probe** | Name of the probe you want to use as the source for the reference channel. |
| **Sync trigger settings** | Select the checkbox to synchronize your simulated signals with real signals. **Sync trigger settings** means applying the physical channel trigger settings to this reference channel and aligning simulated time points on all reference channels with the physical channel trigger settings. You can apply the trigger settings to only one reference channel at one |

| Setting | Description |
|---|---|
| | time. By default, the checkbox does not contain a checkmark. |

Display Window (Oscilloscope)

The Oscilloscope displays the acquired signals on the left side of the panel.

## Time Pane

The **Time** pane displays the time-domain data.

Use cursors to measure a given point on a trace, or compare the differences between two points. You can select one of the three modes for a cursor:

| Cursor Mode | Description |
|---|---|
| **Off** | Turns off the cursors. |
| **Track** | Cursor positions are restricted to the acquired data points on a trace. |
| **Manual** | Cursors positions are not restricted to the acquired data points on a trace. |

When **Cursors** is set to **Manual** or **Track**, a table appears above the display with the following data:

| Column | Description |
|---|---|
| **C1** | Displays the time and amplitude readings for Cursor 1. |
| **C2** | Displays the time and amplitude readings for Cursor 2. |
| **\|Δ Value\|** | Displays the absolute time and amplitude differences between Cursor 1 and Cursor 2. |
| **1/\|Δ Value\|** | Displays the reciprocal of the absolute time difference between Cursor 1 and Cursor 2. |

The **Time** pane displays the Oscilloscope's acquisition status and sample rate(s) in the upper right corner of the graph. The Oscilloscope has the following acquisition statuses:

- **Waiting for trigger**—The instrument is waiting for trigger.

- **Triggered**—The instrument is triggered.
- **Auto Triggered**—The instrument is triggered by an automatic trigger.
- **Force Triggered**—The instrument is triggered by a manual trigger.
- **Stopped**—The instrument is stopped.

When repetitive sampling is enabled, the pane displays both the repetitive sampling rate of the trigger signal and the normal sampling rate of the remaining signals.

## Measurements Pane

The **Measurements** pane displays the measurements of the channels, including the peak-to-peak voltage, root mean square (RMS), frequency, and period.

## FFT Pane

The **FFT** pane displays the frequency-domain data. The **FFT** pane appears only when you enable the FFT channel.

Use cursors to measure a given point on a trace, or compare the differences between two points. You can select one of the three modes for a cursor:

| Cursor Mode | Description |
|---|---|
| **Off** | Turns off the cursors. |
| **Track** | Cursor positions are restricted to the acquired data points on a trace. |
| **Manual** | Cursors positions are not restricted to the acquired data points on a trace. |

When **Cursors** is set to **Manual** or **Track**, a table appears above the display with the following data:

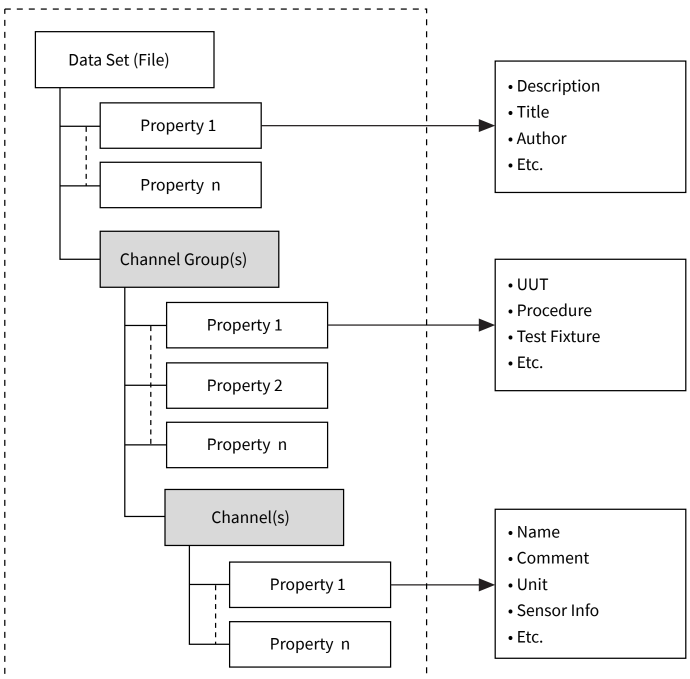
| Column | Description |
|---|---|
| **C1** | Displays the frequency and gain readings for Cursor 1. |
| **C2** | Displays the frequency and gain readings for Cursor 2. |

| Column | Description |
|---|---|
| **\|Δ Value\|** | Displays the absolute frequency and gain differences between Cursor 1 and Cursor 2. |

Click the configuration button on the top right of the pane to access the **Gain mapping** control. You can choose **Logarithmic** or **Linear** for the **Gain** axis.

## TDMS File Format (Oscilloscope)

You can export the data acquired by the Oscilloscope to TDMS files.
The TDMS file format is supported by the TDM data model. The TDM data model arranges the data in three hierarchical levels: file, channel groups, and channels. The following figure illustrates the TDM data model:




The TDMS files that you export from the SFPs contain some custom properties, for example, **product-name**. For all TDMS files that you generate using the NI ELVIS III instruments, the value of **product-name** is always NI ELVIS III. These TDMS files also have designated names for the channel group and channels.
The TDMS files you export from the Oscilloscope have the following data structure:

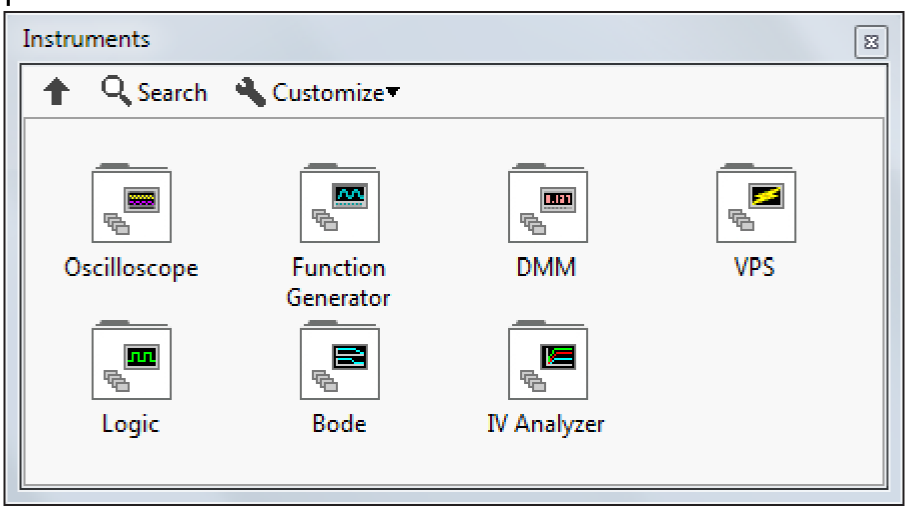
- **channel_group**: Scope

- **channel**: CH1
- **channel**: CH2
- **channel**: MATH
- ...

The TDMS files list enabled channels only. The order of the channels in the TDMS files matches the top-to-bottom order of the channels on the configuration panel on the right side of the SFP.

## Programming the Oscilloscope

You can use the Oscilloscope VIs installed with the LabVIEW ELVIS III Toolkit to program the Oscilloscope instrument on the NI ELVIS III. Find the Oscilloscope VIs on the **Academic I/O** » **Instruments** » **Oscilloscope** subpalette on the Functions palette.




### Related information:

- Oscilloscope VIs

## Streaming Data from Multisim Live

Streaming data from Multisim Live to Measurements Live allows you to compare your circuit design with real measurements.

You can simulate the theoretical concepts in Multisim Live, prototype the actual circuit with NI ELVIS III, and stream your circuit design from Multisim Live to Measurements Live. Afterwards, compare the simulation with real-world measurements inside the Measurements Live environment using the NI ELVIS III

Oscilloscope. Refer to the Multisim Live Tutorial for a complete workflow of streaming data from Multisim Live.

Multisim Live is an online SPICE circuit simulator that enables you to create, interactively simulate, learn, and share circuits in a web browser.

## Oscilloscope Concepts

This section introduces some of the important concepts about the Oscilloscope.

- Automatic Setup
- Analog Edge Trigger
- Digital Edge Trigger
- Hysteresis
- Reference Channel
- Repetitive Sampling

### Automatic Setup

When you select **Auto setup** for the Oscilloscope, the instrument senses the input signal and automatically chooses many of the instrument settings that best suit the input signal.

If a signal is detected on an analog channel, the Oscilloscope chooses the smallest available vertical range that is larger than the signal range, and an offset to center the waveform within that range. For example, if the signal is a 1.2 $V_{pk-pk}$ sine wave, the Oscilloscope will choose the 2 V vertical range for that channel. An analog channel is considered to have a signal present if the signal is at least 10% of the smallest vertical range available for that channel and vertical offset.

If a signal is detected on at least one analog channel, the lowest-numbered analog channel with a signal will be used as the **Analog edge** trigger, and the sample rate and acquisition time will be optimized for that channel.

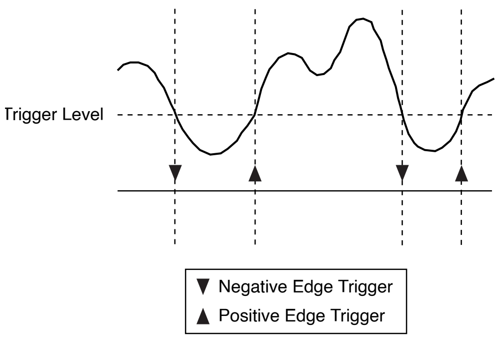
If no signal is detected on any analog channel, all settings will be configured to default values, and a warning occurs.

### Analog Edge Trigger

An analog edge trigger occurs when an analog signal crosses a trigger threshold that you specify.

A positive edge trigger occurs on the rising slope of the source signal, and a negative edge trigger occurs on the falling slope of the source signal. Analog edge triggering is possible on all of the Oscilloscope's analog channels.

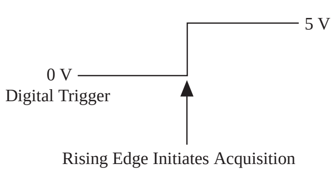
The following figure shows where analog edge triggers occur:




## Digital Edge Trigger

A digital edge trigger occurs when a digital signal transitions from the low level to the high level, also known as a rising-edge trigger.

The following figure shows a rising-edge trigger:




## Hysteresis

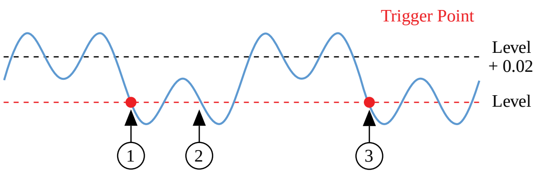
Hysteresis, also known as window size, adds a window above or below the trigger level. Hysteresis reduces false triggering caused by noise or jitter in the signal. The Oscilloscope uses a hysteresis which is 25% of **Volts/div**. For example, when you set **Volts/div** to 0.08 V, the hysteresis is 0.02 V. The way this hysteresis works depends on the value you specify for **Slope** in trigger settings, as described in the following table:

| Slope | Description |
| --- | --- |
| **Rising** | The Oscilloscope detects a trigger when the signal starts or drops below **Level** minus hysteresis and then rises above **Level**. |
| **Falling** | The Oscilloscope detects a trigger when the signal starts or rises above **Level** plus hysteresis and then drops below **Level**. |

| Slope | Description |
|---|---|
| **Either** | The Oscilloscope detects a trigger when the signal starts or drops below **Level** minus hysteresis/2 and then rises above **Level**, or when the signal starts or rises above **Level** plus hysteresis/2 and then drops below **Level**. |

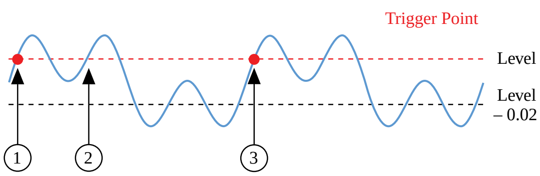
## How the Oscilloscope Uses Hysteresis to Avoid False Triggering

The following figure explains how the Oscilloscope uses hysteresis to avoid false triggering when **Slope** is **Falling**:



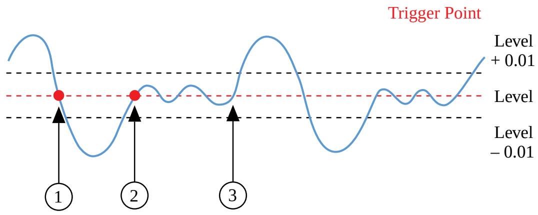

1. The Oscilloscope detects a trigger when the signal first rises above the trigger level plus hysteresis and then drops below the trigger level.
2. The Oscilloscope does not detect any trigger, because the signal stays below the trigger level plus hysteresis since the Oscilloscope last detected a trigger.
3. The Oscilloscope detects another trigger.

The following figure explains how the Oscilloscope uses hysteresis to avoid false triggering when **Slope** is **Rising**:




1. The Oscilloscope detects a trigger when the signal first drops below the trigger level minus hysteresis and then rises above the trigger level.
2. The Oscilloscope does not detect any trigger, because the signal stays above the trigger level minus hysteresis since the Oscilloscope last detected a trigger.
3. The Oscilloscope detects another trigger .

The following figure explains how the Oscilloscope uses hysteresis to avoid false triggering when **Slope** is **Either**:




1. The Oscilloscope detects a trigger when the signal first rises above the trigger level plus hysteresis/2 and then drops below the trigger level.
2. The Oscilloscope detects another trigger when the signal first drops below the trigger level minus hysteresis/2 and then rises above the trigger level.
3. The Oscilloscope does not detect any trigger, because the signal stays above the trigger level minus hysteresis/2 since the Oscilloscope last detected a trigger.

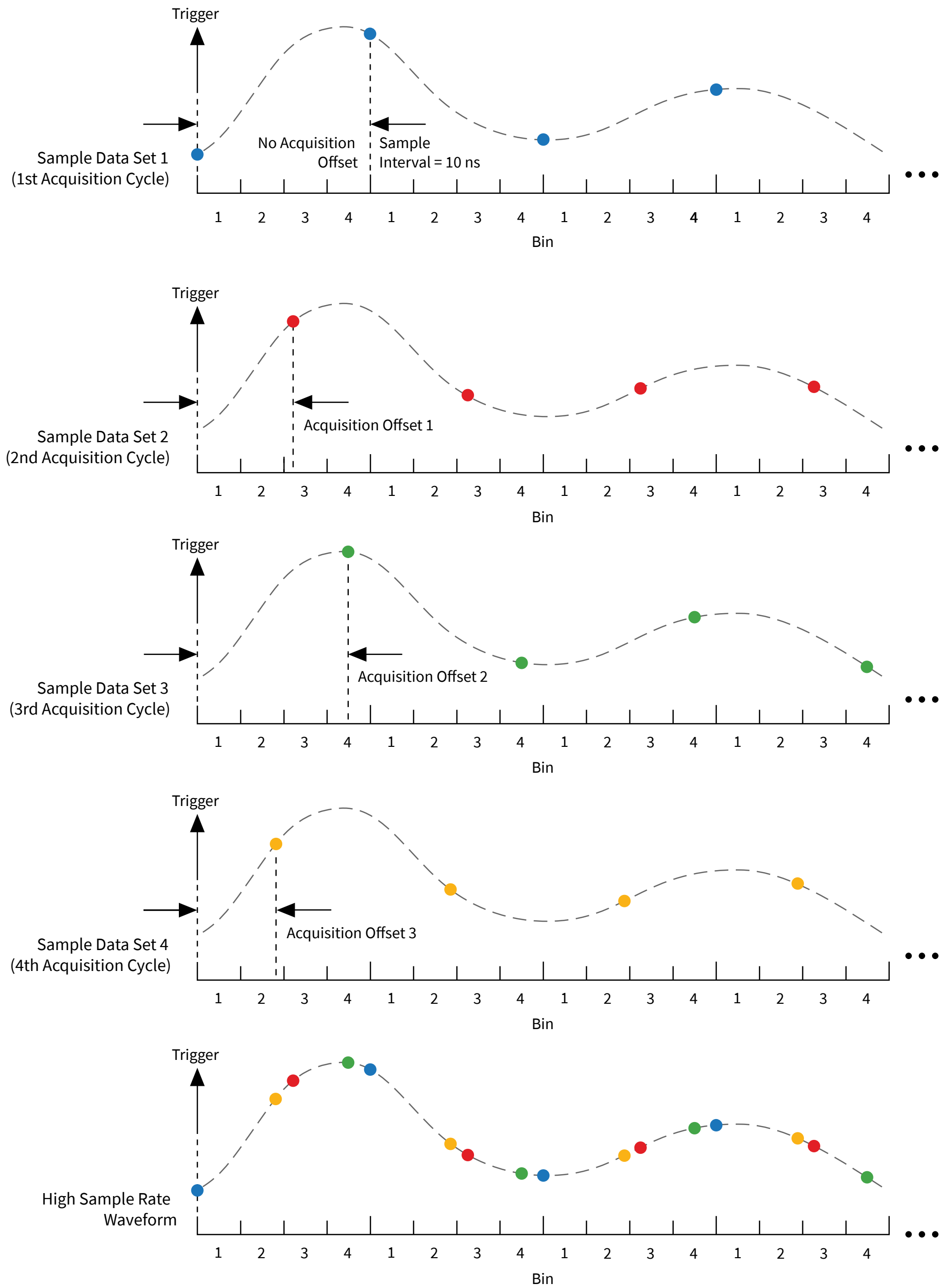
## Reference Channel

A reference channel is a snapshot of the state of an analog channel. A reference channel acts just like other analog channels but its data does not change after the snapshot is taken. Measurements and cursors can be used on reference channels. You can use data files exported from the Oscilloscope as the source for a reference channel.

## Repetitive Sampling

Repetitive sampling increases the sample rate of the Oscilloscope by shifting the acquisition time for different acquisition cycles and interleaving multiple acquisition results.

The following figure explains how the Oscilloscope achieves a repetitive sampling rate that is four times the configured sample rate.




## Multiple Acquisition Cycles

When you set the sample rate to 100 MS/s, the Oscilloscope samples the signal every 10 ns, known as the sample interval.

To achieve a repetitive sampling rate of 400 MS/s, the Oscilloscope divides the sample interval into four bins, and ensures that each bin is filled with a sample data point. To do so, the Oscilloscope starts sampling at a different time during each acquisition by inserting a time lapse, known as acquisition offset, between when the trigger occurs and when the Oscilloscope starts sampling. The acquisition offset ranges from 0 to 10 ns.

To ensure that each bin is filled with a sample date point, at least four acquisition cycles are needed because of the randomness of the acquisition offset. The previous figure shows an ideal situation with just four acquisition cycles.

## High Sample Rate Waveform

When all four bins are filled with sample data points, the Oscilloscope interleaves the sample data sets to form a single, high sample rate waveform.

The high sample rate waveform in the previous figure illustrates the concept of ideal repetitive waveform acquisition. In the actual case, the Oscilloscope samples for at least 16 acquisition cycles to generate a high sample rate waveform. The generated data points, which are uniformly distributed in each bin, are averages of four sample data points in the corresponding bin.

## Limitations

Because the Oscilloscope creates the waveform from multiple samples acquired at different times with respect to the trigger, the repetitive sampling method has certain limitations. The input signal must be periodic and low-noise.

## Oscilloscope Specifications

Specifications are **Typical** unless otherwise noted.

| | |
|---|---|
| Number of channels | 4 |
| **Maximum sampling rate (per channel)** | |
| with repetitive sampling enabled | 400 MS/s |
| without repetitive sampling enabled | 100 MS/s |
| Resolution | 14 bits |

| | |
|---|---|
| Bandwidth | 50 MHz at -3 dB |
| Input impedance | 1 MΩ ‖ 15 pF |
| Input coupling | AC, DC |
| AC coupling cut-off frequency | 12 Hz at -3 dB |
| Overvoltage protection | ±50 V |
| Accuracy | 2% of input + 1% of full scale |

**Table 2.** Input Range

| Range | Full Scale | Offset | Offset Accuracy |
|---|---|---|---|
| High gain (≤200 mV/div) | 2 V peak-to-peak | ±1 V | ±25 mV |
| Low gain (>200 mV/div) | 50 V peak-to-peak | ±25 V | ±625 mV |

**Note** Input voltages should not exceed 50 V DC or 30 V RMS.

## Function and Arbitrary Waveform Generator

The Function and Arbitrary Waveform Generator (FGen/Arb) generates standard or arbitrary waveforms by using Function Generator CH 1 and CH 2 on the NI ELVIS III.
Find an introductory video about using the instrument on ni.com.
Find out more about the Function and Arbitrary Waveform Generator in the following topics:

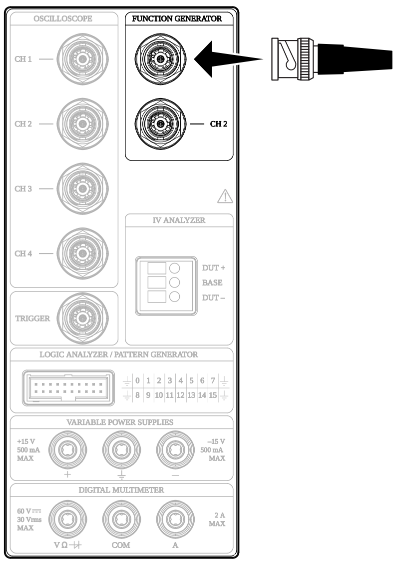
- Connecting Signals to the Function and Waveform Arbitrary Generator
- Generating a Waveform
- Configuration Options (FGen/Arb)
- Programming the Function and Arbitrary Waveform Generator
- Frequency Sweep
- Function and Arbitrary Waveform Generator Specifications

### Related reference:

- Choosing the Instruments

## Connecting Signals to the Function and Waveform Arbitrary Generator

The NI ELVIS III instrumentation panel provides two channels for function and waveform generator output signals.

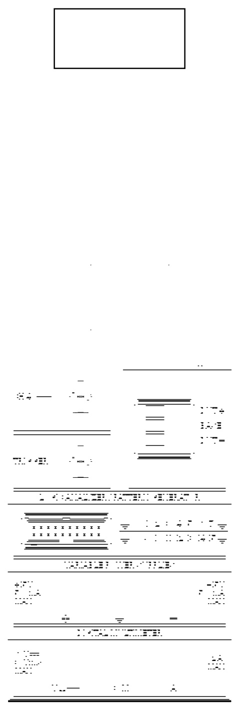



## Generating a Waveform

This example illustrates how to generate a swept sine wave by using the Function and Arbitrary Waveform Generator (FGen/Arb).

The procedure for generating other types of waveforms is similar. Refer to the Configuration Options (FGen/Arb) section to learn about other generation modes.

1. Connect a BNC cable between Function Generator CH 1 and the circuit under test, as shown in the following diagram:



Refer to Connecting Signals to the Function and Waveform Arbitrary Generator for more details about connecting to the FGen/Arb.

2. Launch the FGen/Arb Soft Front Panel (SFP).
3. Select **Sweep** as the channel mode and **Sine** as the waveform type for Channel 1.
4. Select the desired waveform settings.
5. Click **Run** to start signal generation. While the instrument is running, you can modify the waveform in real time by adjusting the settings.

> **Note** You may hear clicking sounds from the NI ELVIS III while operating the SFP. This is expected behavior and is due to the switching of internal relays.

**Related information:**

- Launching the Soft Front Panels

## Configuration Options (FGen/Arb)

Use configuration options on the Function and Arbitrary Waveform Generator (FGen/Arb) Soft Front Panel (SFP) to control the type and parameters of the waveform you want to generate. For example, you can configure symmetry to generate symmetric waveforms.

## Toolbar Controls

Use the toolbar controls to perform common tasks for the FGen. The toolbar is located at the top of the SFP.

| Control | Description |
|---|---|
| **Drag and drop** | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| **Dropdown** | Click to open the current SFP in a separate browser window. To restore it to the original window, close the SFP by clicking and launch it again. |
| **Run/Stop** | Click **Run** to start data acquisition or generation. Click **Stop** to stop data acquisition or generation. |
| **Collaboration** | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the instrument. For more information, refer to Collaborating with Your Team Members. |

## Static Mode

The following settings are available when you select the **Static** mode:

| Setting | Description |
| --- | --- |
| **Waveform** | Type of waveform you want to generate. |
| **Frequency** | Frequency of the waveform. This option is available only if you select **Sine**, **Square**, or **Triangle** as the waveform type. |
| **Amplitude** | Peak-to-peak voltage of the waveform. This option is available only if you select **Sine**, **Square**, or **Triangle** as the waveform type. |
| **DC offset** | Mean amplitude displacement from zero. This option is available only if you select **Sine**, **Square**, or **Triangle** as the waveform type. |
| **Phase** | Initial phase, in degrees, of the waveform. This option is available only if you select **Sine**, **Square**, or **Triangle** as the waveform type.<br>**Note** If you want to generate phase-shifted waveforms, you can open multiple channels and set different phases for the waveforms. If you want a stable phase shift, ensure that the waveforms are synchronized. To achieve that, the |

| Setting | Description |
|---|---|
| | waveforms must have the same frequency and you are not recommended to change their frequencies when the SFP is running. |
| **Duty cycle** | Percentage of the pulse width in the total period of the waveform. This option is available only if you select **Square** as the waveform type. |
| **Symmetry** | Projection of the line segment between the start and the peak divided by the width of a waveform cycle. The default is 50%. This option is available only if you select **Triangle** as the waveform type. |
| **Voltage** | Amplitude displacement from zero. This option is available only if you select **DC** as the waveform type. |

## Sweep Mode

The following settings are available when you select the **Sweep** mode:

| Setting | Description | Operational Limits |
|---|---|---|
| **Waveform** | Type of waveform you want to generate. | |
| **Start frequency** | Frequency at which the waveform frequency sweep starts. **Start frequency** can be | Range: 200 mHz to 15 MHz<br>Resolution: 0.001 mHz |

| Setting | Description | Operational Limits |
|---|---|---|
| | either greater than or less than **Stop frequency**. | |
| **Stop frequency** | Frequency at which the waveform frequency sweep stops. **Start frequency** can be either greater than or less than **Stop frequency**. | Range: 200 mHz to 15 MHz<br>Resolution: 0.001 mHz |
| **Step** | Frequency interval during a frequency sweep.<br>**Note** The number of steps in one sweep equals the rounded up value of \|**Start frequency** - **Stop frequency**\| / **Step**. The instrument stores the steps in the buffer, which can hold no more than 32,768 steps. When the number of steps exceeds 32,768, a buffer overflow error occurs. | Range: 200 mHz to 15 MHz<br>Resolution: 0.001 mHz |
| **Step interval** | Time interval between each waveform generation during a frequency sweep. | Range: 1 ms to 86400 s |
| **Generation mode** | Mode of the waveform generation. Select **Run once** if you only want to generate the waveform once. Select **Loop** if you want to generate the waveform continuously. | - |

| Setting | Description | Operational Limits |
|---|---|---|
| **Amplitude** | Peak-to-peak voltage of the waveform. | - |
| **DC offset** | Mean amplitude displacement from zero. | - |
| **Duty cycle** | Percentage of the pulse width in the total period of the waveform. This setting is available only if you select **Square** as the waveform type. | - |
| **Symmetry** | Projection of the line segment between the start and the peak divided by the width of a waveform cycle. The default is 50%. This option is available only if you select **Triangle** as the waveform type. | - |

## Custom Mode

The following settings are available when you select the **Custom** mode. The **Custom** mode allows you to generate an arbitrary waveform stored in a data file.

| Setting | Description |
|---|---|
| **Source file** | Data file you want to import. You can load CSV files from Multisim and Multisim Live and TDMS files from LabVIEW. The TDMS files must contain only analog waveform data. You can also load CSV or TDMS files exported from the Oscilloscope. The imported channel must contain 32,768 samples or less. |
| **Source channel** | Channel you want to import. If **Source file** contains data of multiple channels, you can use **Source channel** to select which channel of data you want to import to the FGen/Arb. |
| **Trigger source** | Trigger signal. |

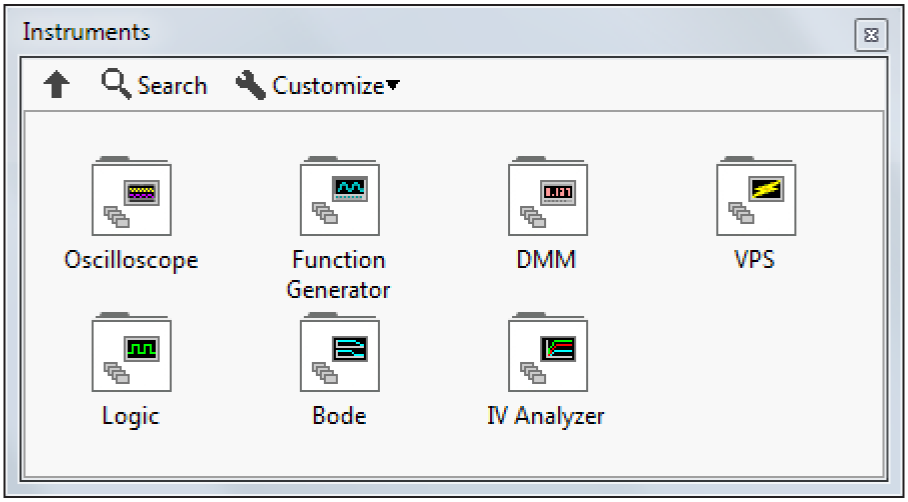
| Setting | Description |
|---|---|
| | ▪ **Immediate**—The signal generation begins immediately without any internal or external trigger source.<br>▪ **TRIG**—The instrument uses the TRIGGER channel on the NI ELVIS III as the trigger signal. |
| **Gain** | Scaling factor that the instrument applies to the amplitude of the loaded waveform. The range of the output amplitude is -10 V to 10 V. |
| **Update rate** | Number of samples that the instrument generates per second. The range of **Update rate** is 0 MS/s to 100 MS/s. The resolution of **Update rate** is 0.001 S/s. |
| **Generation mode** | Mode of the waveform generation. Select **Run Once** if you only want to generate the waveform once. Select **Loop** if you want to generate the waveform continuously. |
| **Preview graph** | Displays the waveform stored in the data file. It may differ from the real waveform generated by the instrument. |

### Related concepts:

- Frequency Sweep

## Programming the Function and Arbitrary Waveform Generator

You can use the Function and Arbitrary Waveform Generator VIs installed with the LabVIEW ELVIS III Toolkit to program the Function and Arbitrary Waveform Generator (FGen/Arb) instrument on the NI ELVIS III. Find the Function and Arbitrary Waveform Generator VIs on the **Academic I/O** » **Instruments** » **Function and Arbitrary Waveform Generator** subpalette on the Functions palette.




### Related information:

- Function and Arbitrary Waveform Generator VIs

## Frequency Sweep

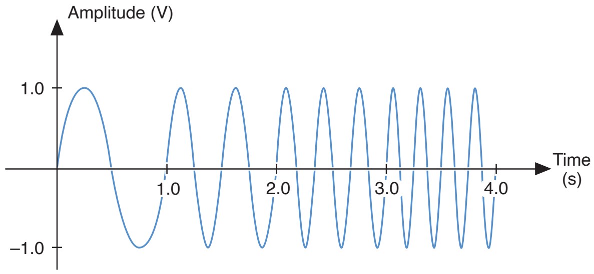
You can generate sweep signals with the Function and Arbitrary Waveform Generator (FGen/Arb). During a frequency sweep, the frequency of the sweep signal changes continuously from the minimum to the maximum or vice versa, at the interval you specify.

The following table shows an example of configuring the FGen/Arb to generate a swept sine wave.

| Control | Value |
|---|---|
| **Mode** | **Sweep** |
| **Waveform** | **Sine** |
| **Start frequency** | 1.0 Hz |
| **Stop frequency** | 4.0 Hz |
| **Step** | 1.0 Hz |
| **Step interval** | 1.0 s |
| **Generation mode** | **Run once** |

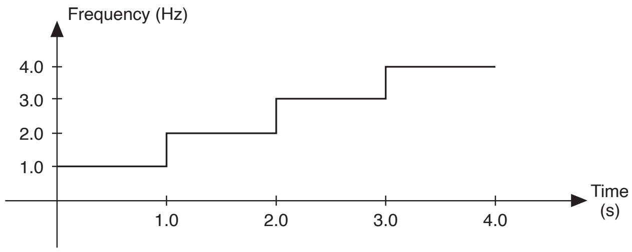
When you configure the FGen/Arb as in the previous table, the instrument generates a waveform as follows:

**Figure 1.** Output waveform when **Generation mode** is **Run Once**




The following figure explains how the frequency increases from 1.0 Hz to 4.0 Hz at the interval of 1.0 Hz. The frequency changes every one second, as specified in **Step interval**.

**Figure 2.** Frequency of the output waveform when **Generation mode** is **Run Once**




If **Generation mode** is set to **Loop**, the instrument generates the waveform continuously. The frequency first increases from 1.0 Hz to 4.0 Hz, and then goes back to 1.0 Hz and increases to 4.0 Hz again. The sweep continues until the instrument stops running.

**Figure 3.** Output waveform when **Generation mode** is **Loop**

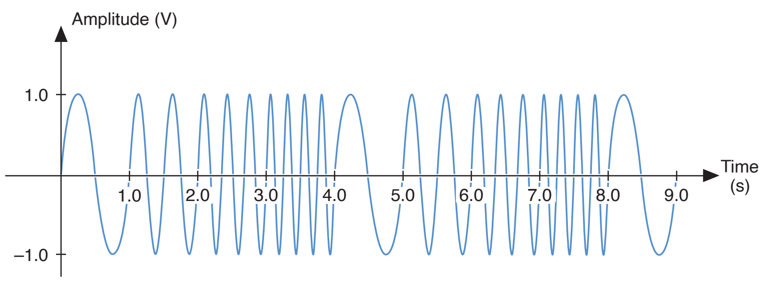
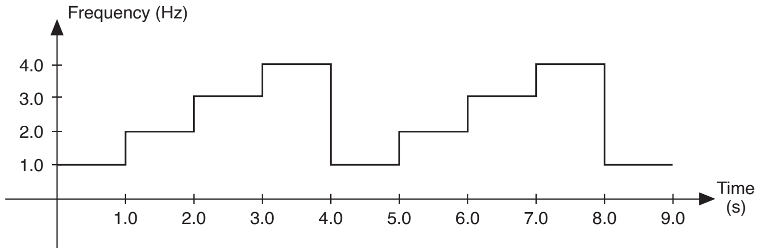


**Figure 4.** Frequency of the output waveform when **Generation mode** is **Loop**




## Related reference:

- Configuration Options (FGen/Arb)

# Function and Arbitrary Waveform Generator Specifications

Specifications are **Typical** unless otherwise noted.

| | |
|---|---|
| Number of channels | 2 |
| Maximum update rate (per channel) | 100 MS/s |
| Resolution | 14 bits |
| Slew rate | 188 V/µs |
| Small signal bandwidth (-3 dB) | 15 MHz with no load |

**Figure 5.** Function Generator Maximum Amplitude vs Frequency

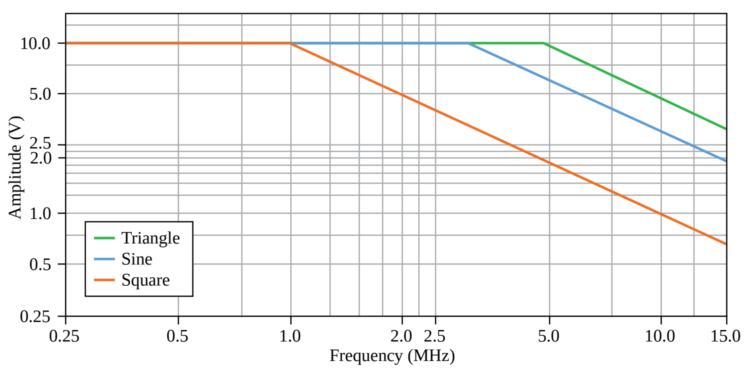


**Table 3.** Output Range

| Gain Setting | AC Amplitude Range | DC Offset Range | Resolution | Amplitude Error | DC Offset Error | Total Output Range |
|---|---|---|---|---|---|---|
| High gain | ±10 V | ±10 V | 1.25 mV/LSB | ±0.5% | ±50 mV | ±10 V |
| Low gain | ±2.5 V | ±10 V | 0.3 mV/LSB | ±0.5% | ±20 mV | ±10 V |

| | |
|---|---|
| Output impedance | 50 Ω |
| DC current drive | 30 mA maximum |
| Overvoltage protection (per channel) | ±10 V, short-circuit to ground |
| Power-on state | High Impedance |

# Data Generator

The Data Generator generates analog signals by using the analog output on the NI ELVIS III.

Find out more about the Data Generator in the following topics:

- Connecting Signals to Analog Output
- Generating a Waveform
- Configuration Options
- Programming Analog Output with LabVIEW
- Analog Output Specifications

### Related reference:

- Choosing the Instruments

## Generating a Waveform (Data Generator)

This example illustrates how to generate a static triangle waveform by using the Data Generator (Data Gen).

1. Install the NI ELVIS III prototyping board onto the NI ELVIS III workstation.
2. Connect a jumper wire between Bank A, Channel AO0 (A/AO0) and the circuit under test, and between Bank A, AGND and GND of the circuit under test.

Refer to Connecting Signals to Analog Output for more details about connecting to the analog output.

3. Press the application board power button on the workstation to power on the application board.

> **Caution** To ensure safety, you must connect the wires before powering on the application board.

4. Launch the Data Gen Soft Front Panel (SFP).
5. Select **Static** as the channel mode and **Triangle** as the waveform type for A/AO0.
6. Select the desired waveform settings.
7. Click **Run** to start signal generation. While the instrument is running, you can modify the waveform in real time by adjusting the settings.

## Configuration Options (Data Generator)

Use configuration options on the Data Generator (Data Gen) Soft Front Panel (SFP) to control how to generate analog signals. For example, you can generate two or more phase-shifted waveforms.

### Toolbar Controls

Use toolbar controls to perform common tasks for the Data Gen. The toolbar is located at the top of the SFP.

| Control | Description |
|---|---|
| **Drag and drop** | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| **Dropdown** | Click to open the current SFP in a separate browser window. To restore it to the original window, close the SFP by clicking and launch it again. |

| Control | Description |
|---|---|
| **Run/Stop** | Click **Run** to start data acquisition or generation. Click **Stop** to stop data acquisition or generation. |
| **Collaboration** | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the instrument. For more information, refer to Collaborating with Your Team Members. |

## Static Mode

The following settings are available when you select the **Static** mode:

| Setting | Description |
|---|---|
| **Waveform** | Type of waveform you want to generate. |
| **Frequency** | Frequency of the waveform. This option is available only if you select **Sine**, **Square**, or **Triangle** as the waveform type. |
| **Amplitude** | Peak-to-peak voltage of the waveform. This option is available only if you select **Sine**, **Square**, or **Triangle** as the waveform type. |
| **DC offset** | Mean amplitude displacement from zero. This option is available only if you select **Sine**, **Square**, or **Triangle** as the waveform type. |
| **Phase** | Initial phase, in degrees, of the waveform. This option is available only if you select |

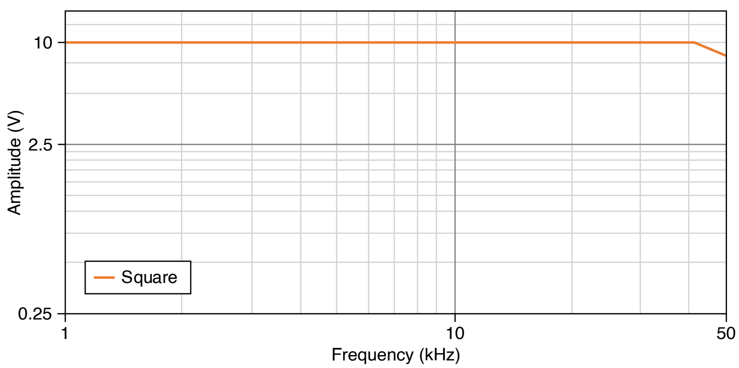
| Setting | Description |
|---|---|
| | **Sine**, **Square**, or **Triangle** as the waveform type.<br><br>**Note** If you want to generate phase-shifted waveforms, you can open multiple channels and set different phases for the waveforms. If you want a stable phase shift, ensure that the waveforms are synchronized. To achieve that, the waveforms must have the same frequency and you are not recommended to change their frequencies when the SFP is running. |
| **Duty cycle** | Percentage of the pulse width in the total period of the waveform. This option is available only if you select **Square** as the waveform type. |
| **Symmetry** | Projection of the line segment between the start and the peak divided by the width of a waveform cycle. The default is 50%. This option is available only if you select **Triangle** as the waveform type. |
| **Voltage** | Amplitude displacement from zero. This option is available |

| Setting | Description |
|---|---|
| | only if you select **DC** as the waveform type. |

### Data Generator Maximum Amplitude vs Frequency

For sine and triangle waveforms, the maximum amplitude is 10 V (20 V peak-to-peak) when the frequency is within the valid range of 100 mHz to 50 kHz. But for a square waveform, the maximum amplitude is less than 10 V when the frequency is close to its upper limit. The following figure illustrates the relationship between maximum amplitude and frequency for a square waveform.

**Figure 6.** Data Generator Maximum Amplitude vs Frequency




The output range for the Data Gen is ±10 V. The output waveform will be distorted when **Amplitude** plus **DC offset** exceeds the total output range. Refer to the following tables to learn more about the output range.

**Table 4.** Output Range

| AC Amplitude Range | DC Offset Range | Frequency | Total Output Range |
|---|---|---|---|
| ±10 V | ±10 V | [100 mHz, 50 kHz] | ±10 V |

# Digital Multimeter

You can use the Digital Multimeter (DMM) to take a variety of measurements.

These measurements include:

- DC or AC voltage
- DC or AC current
- Resistance

- Capacitance
- Inductance
- Voltage drop across a diode
- Continuity

Find an introductory video about using the instrument on ni.com.

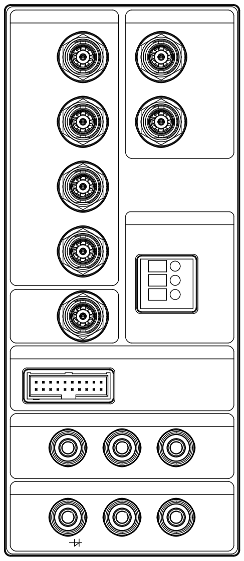
Find out more about the Digital Multimeter in the following topics:

- Connecting Signals to the Digital Multimeter (DMM)
- Taking a Measurement
- Configuration Options (DMM)
- Programming the Digital Multimeter
- Digital Multimeter (DMM) Specifications

## Connecting Signals to the Digital Multimeter (DMM)

The DMM on the NI ELVIS III is isolated and its terminals are the three banana jacks on the instrument panel. For DC Voltage, AC Voltage, Resistance, Diode, and Continuity Test modes, connect signals to the VΩ and COM connectors. For DC Current and AC Current modes, connect signals to the A and COM connectors. For Capacitance and Inductance modes, connect signals to the Current-Voltage (IV) Analyzer connector. The Current-Voltage (IV) Analyzer is non-isolated and its terminals are three screw terminals on the instrument panel.

**Note** The DMM has a fuse for over-current protection. The DC Current and AC Current modes are disabled if the fuse is blown. Replace the fuse (2.5 A fuse, 5MF2.5-R) to re-enable the current modes. Refer to the connectors diagram in Using the Instruments to locate the DMM fuse.



1. Capacitance and inductance measurements are done through the Current-Voltage (IV) Analyzer connector

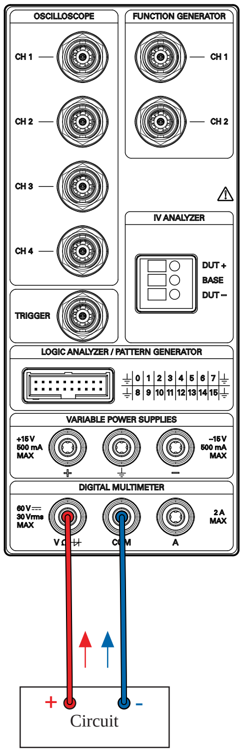
**Note** For capacitance and inductance measurements, insert the component directly into the IV Analyzer screw terminals to achieve the highest accuracy measurements. In cases where the component cannot be directly inserted, use short wires to minimize parasitic effects that impact the overall impedance of the device under test.

## Taking a Measurement

This example illustrates how to use the Digital Multimeter (DMM) to measure voltage.

The procedure for taking other types of measurements is similar. Refer to the Configuration Options (DMM) section to learn about other measurement modes.

1. Connect the circuit under test to the NI ELVIS III, as shown in the following diagram:




Refer to Connecting Signals to the Digital Multimeter (DMM) for more details about connecting to the DMM.

2. Launch the DMM Soft Front Panel (SFP).
3. Depending on the type of the input signal, click the **DC voltage** or **AC voltage** button, and select a proper range.
4. Click **Run** to start measuring the signal.
   Read the real-time voltage from the panel. While the instrument is running, you can change the measurement mode and the range by adjusting the settings on the panel.

**Note** You may hear clicking sounds from the NI ELVIS III while operating the SFP. This is expected behavior and is due to the switching of internal relays.

**Related information:**

- Launching the Soft Front Panels

# Configuration Options (DMM)

Use configuration options on the Digital Multimeter (DMM) Soft Front Panel (SFP) to decide the measurement mode and range. For example, you can measure DC voltage or AC voltage.

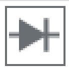
## Toolbar Controls

Use the toolbar controls to perform common tasks for the DMM. The toolbar is located at the top of the SFP.

| Control | Description |
|---|---|
| **Drag and drop** | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| **Dropdown** | Click to open the current SFP in a separate browser window. To restore it to the original window, close the SFP by clicking and launch it again. |
| **Run/Stop** | Click **Run** to start data acquisition or generation. Click **Stop** to stop data acquisition or generation. |
| **Collaboration** | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the instrument. For more information, refer to Collaborating with Your Team Members. |

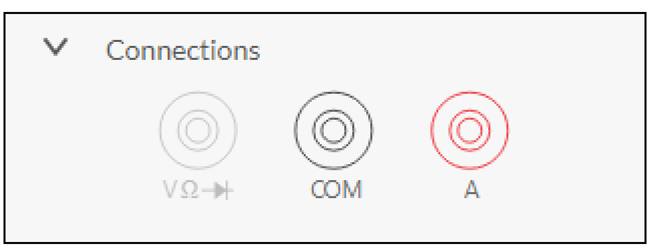
The buttons on the DMM SFP represent different types of measurements. To enable a measurement mode, click the corresponding button.

| Measurement mode | Button | Description |
|---|---|---|
| **DC voltage** | V | Measures the DC component of a voltage source. |

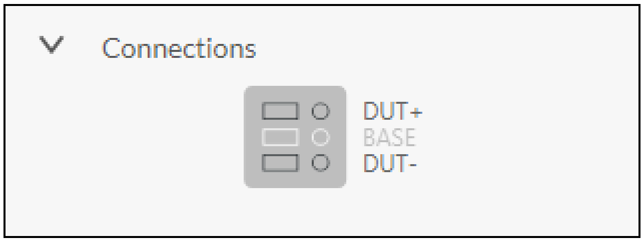
| Measurement mode | Button | Description |
|---|---|---|
| **AC voltage** |  | Measures the AC component of a voltage source. |
| **DC current** |  | Measures the DC component of a current source. |
| **AC current** |  | Measures the AC component of a current source. |
| **Resistance** |  | Measures resistance. |
| **Capacitance** |  | Measures capacitance. |
| **Inductance** |  | Measures inductance. |
| **Diode** |  | Measures the voltage drop across a diode. When you select **Diode**, **Range** specifies the threshold for a valid diode measurement. |
| **Continuity** |  | Tests for continuity. The threshold for continuity measurements is 15 Ω. The DMM beeps when it detects a short circuit. |

**Range** specifies the range of the measurement. When you select **Automatic**, the DMM automatically chooses the range that suits the input signal.

In the **Connections** section, the SFP highlights the channels to connect to depending on the measurement mode. For example, when the measurement mode is **DC current** or **AC current**, the **Connections** section highlights the channels as follows:

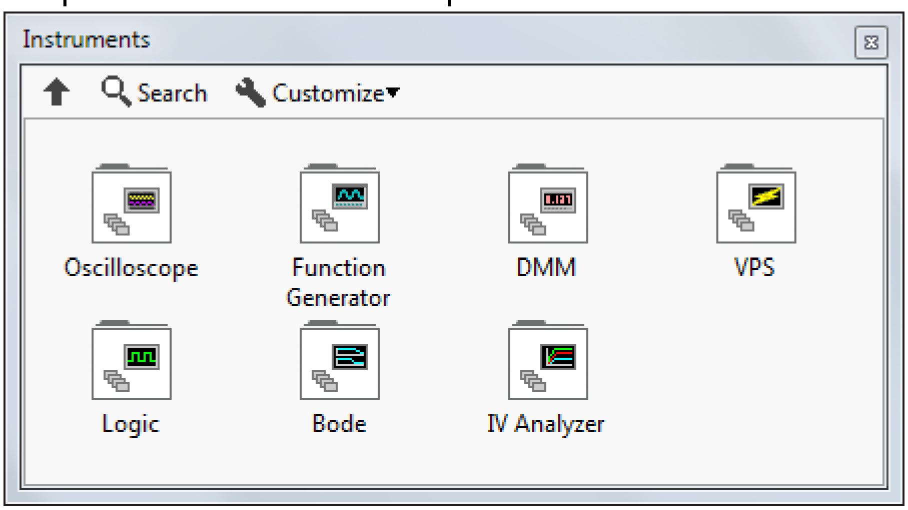


When the measurement mode is **Capacitance** or **Inductance**, the **Connections** section highlights the terminals on the NI ELVIS III to connect to the device under test (DUT):




## Programming the Digital Multimeter

You can use the Digital Multimeter VIs installed with the LabVIEW ELVIS III Toolkit to program the Digital Multimeter (DMM) instrument on the NI ELVIS III. Find the Digital Multimeter VIs on the **Academic I/O** » **Instruments** » **Digital Multimeter** subpalette on the Functions palette.




### Related information:

- Digital Multimeter VIs

## Digital Multimeter (DMM) Specifications

Specifications are **Typical** unless otherwise noted.

| | |
|---|---|
| Isolated functions | DC/AC voltage, DC/AC current, resistance, diode voltage, diode continuity |
| Non-isolated functions | Capacitance, inductance |
| Isolation level | Functional isolation |
| Resolution | 4.5 digits |
| Input impedance | 10 MΩ |
| Input coupling | DC/AC |
| Connectivity | Banana jacks |
| Voltage input protection | ±60 V |
| Current input protection | 2.5 A fuse, 5MF2.5-R |
| **Measurements** | |
| **Voltage measurement** | |
| DC ranges | 50 mV DC, 500 mV DC, 5 V DC, 50 V DC |
| AC ranges | 50 mV RMS, 500 mV RMS, 5 V RMS, 30 V RMS |
| Input frequency range (AC voltage) | 40 Hz to 1 kHz |
| DC voltage measurement accuracy (50 mV DC) | 0.2% of range |
| DC voltage measurement accuracy (500 mV DC, 5 V DC, 50 V DC) | 0.1% of range |
| AC voltage measurement accuracy at 50 Hz and 60 Hz (50 mV RMS) | 0.2% of range |
| AC voltage measurement accuracy at 50 Hz and 60 Hz (500 mV RMS, 5 V RMS, 30 V RMS) | 0.1% of range |
| **Current measurement** | |
| DC ranges | 2 A DC |
| AC ranges | 2 A RMS |
| Shunt resistance | 20 mΩ |
| Input frequency range (AC current) | 40 Hz to 1 kHz |

| | |
|---|---|
| DC current measurement accuracy | 0.1% of range |
| AC current measurement accuracy at 50 Hz and 60 Hz | 0.1% of range |
| **Resistance measurement** | |
| Ranges | 50 Ω, 500 Ω, 5 kΩ, 50 kΩ, 500 kΩ, 5 MΩ, 50 MΩ |
| Resistance measurement accuracy (500 Ω, 5 kΩ, 50 kΩ, 500 kΩ, 5 MΩ) | 0.1% of range |
| Resistance measurement accuracy (50 Ω, 50 MΩ) | 1% of range |

**Table 5.** Capacitance Measurement Range

| Range | Effective Frequency | Effective Test Resistance |
|---|---|---|
| 50 pF to 500 pF | 10 kHz | 100 kΩ |
| 500 pF to 5 nF | 1 kHz | 10 kΩ |
| 5 nF to 50 nF | 1 kHz | 10 kΩ |
| 50 nF to 500 nF | 1 kHz | 1 kΩ |
| 500 nF to 5 µF | 1 kHz | 1 kΩ |
| 5 µF to 50 µF | 1 kHz | 100 Ω |
| 50 µF to 500 µF | 100 Hz | 100 Ω |

| | |
|---|---|
| Capacitance measurement accuracy | 1% of range |

**Table 6.** Inductance Measurement Range

| Range | Effective Frequency | Effective Test Resistance |
|---|---|---|
| 10 µH to 100 µH | 100 kHz | 100 Ω |
| 100 µH to 1 mH | 10 kHz | 100 Ω |
| 1 mH to 10 mH | 10 kHz | 1 kΩ |
| 10 mH to 100 mH | 1 kHz | 1 kΩ |

| | |
|---|---|
| Inductance measurement accuracy | 1% of range |

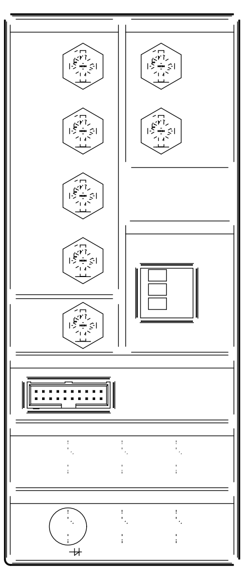
# Variable Power Supply

The Variable Power Supply (VPS) provides two user programmable DC voltage outputs on the NI ELVIS III. The two outputs are independently configurable, one providing positive voltages and the other negative voltages.
Find an introductory video about using the instrument on ni.com.
Find out more about the Variable Power Supply in the following topics:

- Connecting Signals to the Variable Power Supplies
- Supplying a Voltage
- Configuration Options (VPS)
- Programming the Variable Power Supply
- Voltage Sweep
- Variable Power Supplies Specifications

## Connecting Signals to the Variable Power Supplies

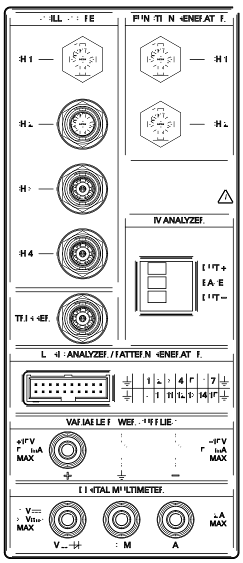
The Variable Power Supplies (VPS) provides access to the adjustable voltage outputs through three banana jacks. Voltages from +1 V to +15 V are supplied by the "+" terminal, voltages from -1 V to -15 V are supplied by the "–" terminal, and both outputs share a common ground provided by the ground terminal. The positive and negative voltage outputs can be used simultaneously and independently.



## Supplying a Voltage

This example illustrates how to supply a voltage with the Variable Power Supplies (VPS) instrument:

1. Connect VPS +/- and ground to the circuit under test, as shown in the following diagram:



   Refer to Connecting Signals to the Variable Power Supplies for more details about connecting to the VPS.
2. Launch the VPS Soft Front Panel (SFP).
3. Adjust the settings for the channel you have connected to.
4. Click **Run** to start supplying a voltage. If the instrument is in **Static** mode, you can modify the voltage in real time by adjusting the setting while the instrument is running.

> **Note** You may hear clicking sounds from the NI ELVIS III while operating the SFP. This is expected behavior and is due to the switching of internal relays.

### Related information:

- Launching the Soft Front Panels

## Configuration Options (VPS)

Use configuration options on the Variable Power Supply (VPS) Soft Front Panel (SFP) to configure DC voltage outputs. For example, you can configure the voltage range.

### Toolbar Controls

Use the toolbar controls to perform common tasks for the VPS. The toolbar is located at the top of the SFP.

| Control | Description |
| --- | --- |
| **Drag and drop** | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| **Dropdown** | Click to open the current SFP in a separate browser window. To restore it to the original window, close the SFP by clicking and launch it again. |
| **Run/Stop** | Click **Run** to start data acquisition or generation. Click **Stop** to stop data acquisition or generation. |
| **Collaboration** | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the instrument. For more information, refer to Collaborating with Your Team Members. |

### Static Mode

When setting the VPS to the **Static** mode, you can control the voltage and current outputs by changing the **Voltage** and **Current limit** settings. The display on the panel shows the actual voltage, current, and power readings of the output signal.

| Setting | Description | Operational Limits |
|---|---|---|
| **Voltage** | Target voltage level.<br>**Note** A slight deviation exists between the actual voltage and the target voltage due to hardware configuration and the input impedance of the circuit under test. | Range: 1 V to 15 V for the + terminal, -15 V to -1 V for the - terminal<br>Resolution: 10 mV |
| **Current limit** | Maximum current sourced from the instrument.<br>**Note** If the actual current surpasses **Current limit**, the current output is coerced to a value smaller than and close to **Current limit** to protect your hardware. The voltage output is also coerced to a value smaller than **Voltage** and an increase in **Voltage** does not result in an | Range: 20 mA to 500 mA for the + terminal, -500 mA to -20 mA for the - terminal |

| Setting | Description | Operational Limits |
|---|---|---|
| | increase in the voltage output. | |

## Sweep Mode

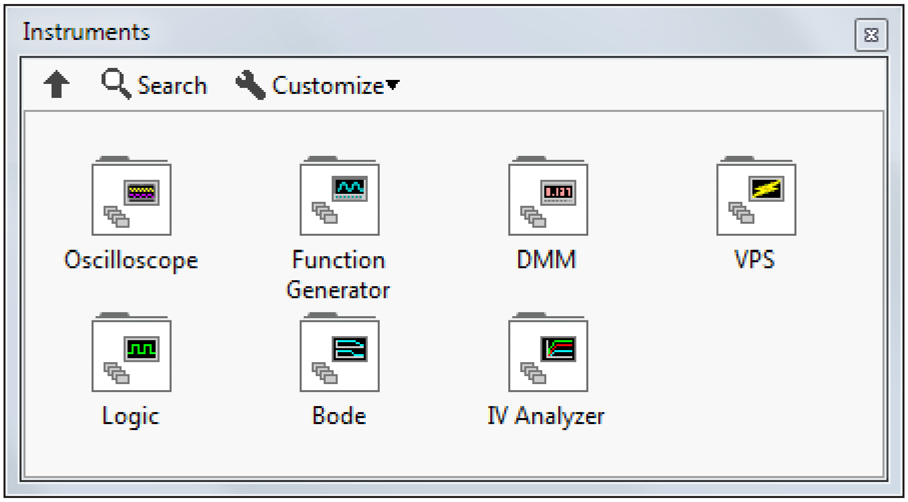
When you set the VPS to the **Sweep** mode, the VPS ramps voltage outputs between the start and stop voltages you specify. The VPS provides the following configuration options. The display on the panel shows the actual voltage, current, and power readings of the output signal.

| Setting | Description | Operational Limits |
|---|---|---|
| **Start voltage** | Target voltage at which the voltage sweep starts. **Start voltage** can be either greater than or less than **Stop voltage**. | Range: 1 V to 15 V for the + terminal, -15 V to -1 V for the - terminal<br><br>Resolution: 10 mV |
| **Stop voltage** | Target voltage at which the voltage sweep stops. **Start voltage** can be either greater than or less than **Stop voltage**. | Range: 1 V to 15 V for the + terminal, -15 V to -1 V for the - terminal<br><br>Resolution: 10 mV |
| **Step** | Voltage interval between each output signal during a voltage sweep. The absolute value of **Step** must be less than the difference between **Start voltage** and **Stop voltage**. | Range: 0 V to 14 V, less than $\lvert V_{Start} - V_{stop} \rvert$<br><br>Resolution: 10 mV |
| **Step interval** | Time interval between each output signal during a voltage sweep. | Range: 0 s to 86400 s |
| **Generation mode** | Mode in which the device generates voltage. Select **Run Once** if you want to generate the voltage sweep only once. Select **Loop** if you want to | - |

| Setting | Description | Operational Limits |
|---|---|---|
| | generate the voltage sweep continuously. | |
| **Current limit** | Maximum current sourced from the instrument.<br>**Note** If the actual current surpasses **Current limit**, the current output is coerced to a value smaller than and close to **Current limit** to protect your hardware. The voltage output is also coerced to a value smaller than **Voltage** and an increase in **Voltage** does not result in an increase in the voltage output. | Range: 20 mA to 500 mA for the + terminal, -500 mA to -20 mA for the - terminal |

## Programming the Variable Power Supply

You can use the Variable Power Supply VIs installed with the LabVIEW ELVIS III Toolkit to program the Variable Power Supply (VPS) instrument on the NI ELVIS III. Find the Variable Power Supply VIs on the **Academic I/O** » **Instruments** » **Variable Power Supply** subpalette on the Functions palette.




### Related information:

- Variable Power Supply VIs

## Voltage Sweep

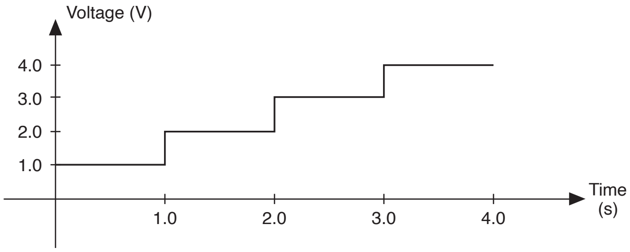
During a voltage sweep, the voltage of the sweep signal changes continuously from the minimum to the maximum or vice versa, at the interval you specify. To generate a sweep signal, select the **Sweep** mode for the Variable Power Supply (VPS).

The following table shows an example of configuring the VPS to generate a voltage sweep.

| Control | Value |
|---|---|
| **Mode** | **Sweep** |
| **Start voltage** | 1.0 V |
| **Stop voltage** | 4.0 V |
| **Step** | 1.0 V |
| **Step interval** | 1.0 s |
| **Generation mode** | **Run once** |

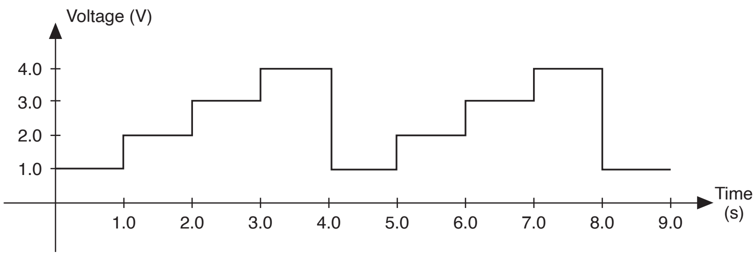
When you configure the VPS as specified in the previous table, the instrument generates a voltage sweep as follows:

**Figure 7.** Output voltage when **Generation mode** is **Run once**




If **Generation mode** is set to **Loop**, the instrument generates the voltage sweep continuously. The voltage first increases from 1 V to 4 V, and then goes back to 1 V and increases to 4 V again. The sweep continues until the instrument stops running.

**Figure 8.** Output voltage when **Generation mode** is **Loop**




## Variable Power Supplies Specifications

Specifications are **Typical** unless otherwise noted.

> **Notice** Exceeding the power limits may cause unpredictable device behavior.

| **Positive variable power output** | |
|---|---|
| Output voltage | +1 V to +15 V |
| Output current | +500 mA maximum |
| DC Voltage accuracy | ±50 mV - $\|I_{out}\|$ × 0.25 mV/mA |
| Ripple and noise | 20 $mV_{pk-pk}$ |
| Voltage readback accuracy | ±15 mV |
| Current readback accuracy | ±5 mA |
| **Negative variable power output** | |

| | |
|---|---|
| Output voltage | -1 V to -15 V |
| Output current | -500 mA maximum |
| DC Voltage accuracy | ±50 mV + $\lvert I_{out} \rvert$ × 0.25 mV/mA |
| Ripple and noise | 55 $mV_{pk-pk}$ + $\lvert V_{out} \rvert$ × 10 $mV_{pk-pk}$/V |
| Voltage readback accuracy | ±15 mV |
| Current readback accuracy | ±5 mA |

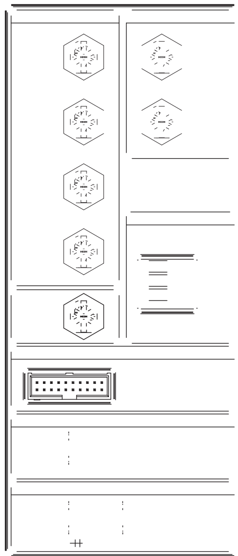
# Bode Analyzer

The Bode Analyzer measures the gain and phase versus frequency for linear circuits.
Find an introductory video about using the instrument on ni.com.
Find out more about the Bode Analyzer in the following topics:

- Connecting Signals to the Bode Analyzer
- Analyzing Frequency Response
- Configuration Options (Bode Analyzer)
- Programming the Bode Analyzer
- Bode Analyzer Specifications

## Connecting Signals to the Bode Analyzer

The Bode Analyzer Soft Front Panel (SFP) uses the Function Generator on the NI ELVIS III to output a stimulus and then uses the Oscilloscope on the NI ELVIS III to measure the stimulus and multiple responses.

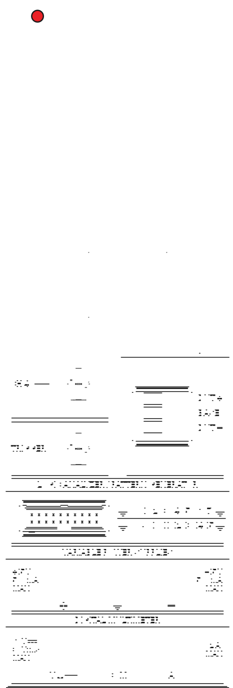


## Analyzing Frequency Response

This topic tells you how to set up the hardware and start analyzing frequency response.

Complete the following steps:

1. Build the circuit for Bode analysis on the NI ELVIS III Application Board.
2. Connect Function Generator CH 1 and Oscilloscope CH1 to the input of the circuit. Connect Oscilloscope CH 2 to the output of the circuit, as shown in the following diagram:



Refer to Connecting Signals to the Bode Analyzer for more details about connecting to the Bode.

3. Launch the Bode Analyzer.
4. In the **Stimulus channel** section, adjust the parameters as desired.
5. Click the **Run** button.
   Results display in the Gain and Phase sections of the display.

> **Note** You may hear clicking sounds from the NI ELVIS III while operating the SFP. This is expected behavior and is due to the switching of internal relays.

**Related information:**

- Launching the Soft Front Panels

## Configuration Options (Bode Analyzer)

Use configuration options on the Bode Analyzer (Bode) Soft Front Panel (SFP) to adjust the stimulus signals and manipulate the display of response signals. For example, you can configure the frequency range of a stimulus signal.

- Toolbar Controls
- Stimulus Channel Settings
- Response Channel Settings
- Reference Channel Settings
- Display Window

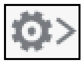

**Note** Click the expand arrow near the upper right corner of the SFP to expand the configuration pane if it is collapsed by default.

## Toolbar Controls (Bode Analyzer)

Use the toolbar controls to perform common tasks for Bode Analyzer. The toolbar is located at the top of the SFP.

The layout of the controls may vary depending on the device from which you launch the SFP. Some controls are hidden by default when you resize the SFP. Click the ellipsis on the toolbar to access hidden controls.

| Control | Description |
|---|---|
| **Drag and drop** | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| **Dropdown** | Click to open the current SFP in a separate browser window. To restore it to the original window, close the SFP by clicking and launch it again. |
| **Run/Stop** | Click **Run** to start data acquisition or generation. Click **Stop** to stop data acquisition or generation. |
| **Collaboration** | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the |

| Control | Description |
|---|---|
| | instrument. For more information, refer to Collaborating with Your Team Members. |
| **Collapse/Expand**  | Collapses or expands the configuration pane on the right side of the panel. |

## Stimulus Channel Settings

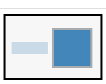
Use this section of the Bode Analyzer to enter basic measurement settings.

The following **Stimulus channel** settings are available.

| Setting | Description | Operational Limits |
|---|---|---|
| **Start frequency** | Specifies the frequency at which to start the Bode plot sweep. | Range: 1 Hz to 2.5 MHz<br>Resolution: 1 Hz |
| **Stop frequency** | Specifies the frequency at which to stop the Bode plot sweep. | Range: 1 Hz to 15 MHz<br>Resolution: 1 Hz |
| **Steps per decade** | Specifies the number of evenly spaced frequency points to sweep per decade. | Range: 1 to 1000<br>Resolution: 1 |
| **Peak amplitude** | Peak amplitude of the sine wave during the sweep. | Range: 50 mV to 10 V<br>Resolution: 10 mV |

## Response Channel Settings

The following **Response channel** settings are available.

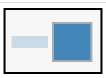
| Setting | Description |
|---|---|
| Colored bar | Matches the color of the corresponding channel's trace on the display. It displays when the channel is set to show on the graph (toggle is on.) |
| Channel name | For information only, for example, Response 1. This is not editable. |
| Toggle  | Shows/hides graph trace. |

| Setting | Description |
|---|---|
| Assigned channel indicator | Shows below any response that is toggled on, for example, Oscilloscope CH2. This refers to the name of the connector on the device. |

## Reference Channel Settings

Configure the reference channel settings to compare the measured signal with the signal on a reference channel.

The following **Reference channel** settings are available.

| Setting | Description |
|---|---|
| Colored bar | Matches the color of this channel's trace on the graph. It is only shown when the channel is set to show on the graph (toggle is on.) |
| Channel name | For information only, for example, Reference 1. This is not editable. |
| Toggle  | Shows/hides graph trace. |
| **Mode** | Origin of data being referenced. This is a Multisim style CSV file. |
| **Source file** | The Multisim style CSV file you wish to reference. This will plot on the display. |
| **Channel** | Auto-populates with valid choices from the **Source file**. Determines what channel in the file to display data from. Select the desired channel. |

## Display Window (Bode Analyzer)

Use **Cursors** to measure a given point on the signal trace, or compare the differences between two points.

| Cursor Mode | Description |
|---|---|
| **Off** | Turns off the cursors. |
| **Track** | Cursor positions are restricted to the acquired data points on a trace. |
| **Manual** | Cursors positions are not restricted to the acquired data points on a trace. |

When **Cursors** is set to **Manual** or **Track**, a table appears above the display with the following data:

| Column | Description |
|---|---|
| **C1** | Displays the Frequency, Gain and Phase readings for Cursor 1. Use the drop-down beside **C1** to select the response to measure. |
| **C2** | Displays the Frequency, Gain and Phase readings for Cursor 2. Use the drop-down beside **C2** to select the response to measure. |
| **\|Δ Value\|** | Displays the Frequency, Gain and Phase readings for C2 - C1. |

## Related reference:

- Response Channel Settings

Display Controls (Bode)

Use the display controls to manipulate the display of the data on the graph.

Click  to access these controls:

| Control | Description |
|---|---|
| **Zoom all** | Click to view all data on the graph. Useful when you have zoomed or panned the display. |
| **Autoscale** | Select **On** to change the scale to display all the data. Select **Off** to keep the scale consistent. Data may flow off the display with this setting, or not fill the display. |
| **Gain mapping** | Choose **Logarithmic** or **Linear** for the **Gain** axis. |
| **Phase mapping** | Choose **Degrees** or **Radians** for the **Phase** axis. |
| **Display with phase shift** | Select to show phase shift display options for Response 1 through Response 3, and Reference 1 and Reference 2.<br><br>If Degrees is selected in **Phase mapping**, Response 1 through Response 3, and Reference |

| Control | Description |
| --- | --- |
| | 1 and Reference 2 will each have a drop-down with phase shift display options in degrees.<br>If Radians is selected in **Phase mapping**, Response 1 through Response 3, and Reference 1 and Reference 2 will each have a drop-down with phase shift display options in rads. |

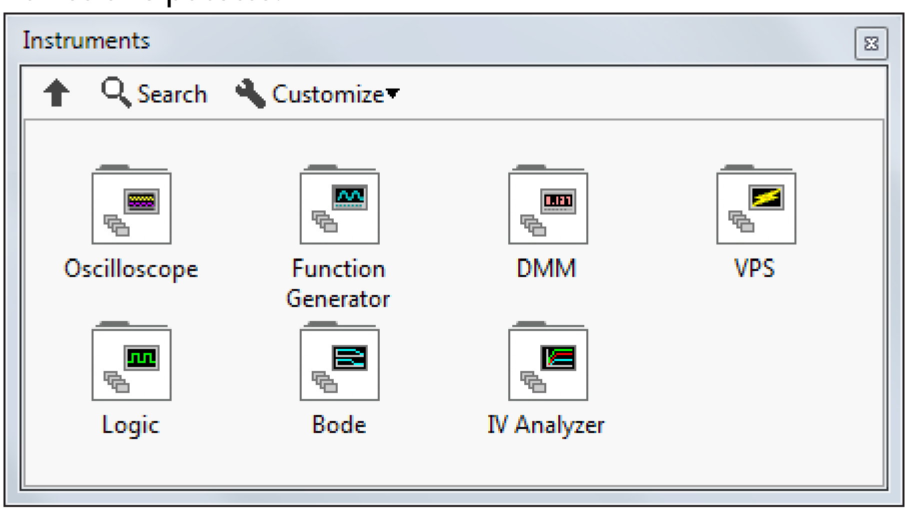
## Zooming and Panning

Use the mouse to do the following:

| Mouse Action | Result |
| --- | --- |
| Scroll mouse wheel | Zooms in/out. |
| Hold down left button and drag | Pans left/right, up/down. |

## Programming the Bode Analyzer

You can use the Bode Analyzer VIs installed with the LabVIEW ELVIS III Toolkit to program the Bode Analyzer instrument on the NI ELVIS III. Find the Bode Analyzer VIs on the **Academic I/O** » **Instruments** » **Bode Analyzer** subpalette on the Functions palette.




### Related information:

- Bode Analyzer VIs

# Bode Analyzer Specifications

Specifications are **Typical** unless otherwise noted.

## Oscilloscope

| | |
|---|---|
| Number of channels | 4 |
| Resolution | 14 bits |
| Bandwidth | 50 MHz at -3 dB |
| Input impedance | 1 MΩ ‖ 15 pF |
| Overvoltage protection | ±50 V |
| Accuracy | 2% of input + 1% of full scale |

**Table 7.** Input Range

| Range | Full Scale | Offset | Offset Accuracy |
|---|---|---|---|
| High gain (≤200 mV/div) | 2 V peak-to-peak | ±1 V | ±25 mV |
| Low gain (>200 mV/div) | 50 V peak-to-peak | ±25 V | ±625 mV |

**Note** Input voltages should not exceed 50 V DC or 30 V RMS.

## Function and Arbitrary Waveform Generator

| | |
|---|---|
| Number of channels | 2 |
| Maximum update rate (per channel) | 100 MS/s |
| Resolution | 14 bits |
| Slew rate | 188 V/µs |
| Small signal bandwidth (-3 dB) | 15 MHz with no load |

**Table 8.** Output Range

| Gain Setting | AC Amplitude Range | DC Offset Range | Resolution | Amplitude Error | DC Offset Error | Total Output Range |
|---|---|---|---|---|---|---|
| High gain | ±10 V | ±10 V | 1.25 mV/LSB | ±0.5% | ±50 mV | ±10 V |
| Low gain | ±2.5 V | ±10 V | 0.3 mV/LSB | ±0.5% | ±20 mV | ±10 V |

| | |
|---|---|
| Output impedance | 50 Ω |
| DC current drive | 30 mA maximum |
| Overvoltage protection (per channel) | ±10 V, short-circuit to ground |
| Power-on state | High Impedance |

## Current-Voltage Analyzer

Use the Current-Voltage (IV) Analyzer to measure the current-voltage (IV) response of the following devices:

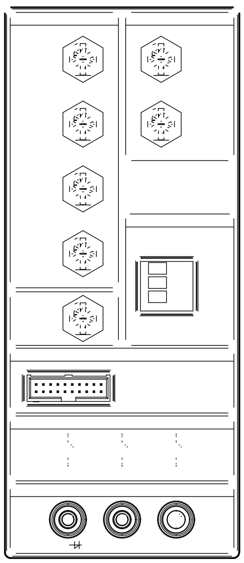
- Diode
- PNP BJT
- NPN BJT

Find an introductory video about using the instrument on ni.com.
Find out more about the Current-Voltage (IV) Analyzer in the following topics:

- Connecting Signals to the IV Analyzer
- Analyzing a Diode
- Analyzing a BJT
- Configuration Options (IV Analyzer)
- Programming the IV Analyzer
- Current-Voltage (IV) Analyzer Specifications

## Connecting Signals to the IV Analyzer

The Current-Voltage (IV) Analyzer on the NI ELVIS III instrument panel includes DUT (device under test) connectors. Use the DUT+, DUT–, and BASE connectors to plot the current-voltage response of a NPN or PNP bipolar transistor. Use DUT+ and DUT- to plot the current-voltage response of a diode.



**Note** Insert the component directly into the IV Analyzer screw terminals to achieve the highest accuracy measurements. In cases where the component cannot be directly inserted, use short wires to minimize parasitic effects that impact the overall impedance of the device under test.

## Analyzing a Diode

Complete the following steps to run current-voltage (IV) analyses for a diode:

1. Launch the IV Analyzer Soft Front Panel (SFP).
2. Confirm that **Diode** is selected under **Analyzer mode**.
3. Connect a diode to the IV ANALYZER terminals on the NI ELVIS III. Attach its anode to DUT+ and its cathode to DUT-. This is shown in the **Connections** diagram under Analyzer & hardware on the SFP.
4. Adjust the parameters as desired.
5. Click the **Run** button.
6. Use the cursors to take measurements.

> **Note** You may hear clicking sounds from the NI ELVIS III while operating the SFP. This is expected behavior and is due to the switching of internal relays.

**Related information:**

- Launching the Soft Front Panels

## Analyzing a BJT

Complete the following steps to run current-voltage (IV) analyses for a BJT:

1. Launch the IV Analyzer Soft Front Panel (SFP).
2. Select **Transistor** under **Analyzer mode**.
3. Connect a BJT to the IV ANALYZER terminals on the NI ELVIS III. Attach its collector to DUT+, its base to BASE, and its emitter to DUT-. This is shown in the **Connections** diagram under Analyzer & hardware on the SFP.

   > **Note** Insert the component directly into the IV Analyzer screw terminals to achieve the highest accuracy measurements. In cases where the component cannot be directly inserted, use short wires to minimize parasitic effects that impact the overall impedance of the device under test.

4. Select **NPN** or **PNP** under **Transistor type**.
5. Adjust the parameters as desired.
6. Click the **Run** button.
7. Use the cursors to take measurements.

   > **Note** You may hear clicking sounds from the NI ELVIS III while operating the SFP. This is expected behavior and is due to the switching of internal relays.

**Related information:**

- Launching the Soft Front Panels

## Configuration Options (IV Analyzer)

Use configuration options on the Current-Voltage (IV) Analyzer Soft Front Panel (SFP) to control how to acquire and analyze the current-voltage response and decide the unit under test. For example, you can choose to measure a diode or a transistor.

- Toolbar Controls
- Analyzer & Hardware Settings
- Voltage Sweep Settings
- Current Range Settings
- Collector Voltage Sweep Settings
- Base Current Sweep Settings
- Display Window

**Note** Click the expand arrow near the upper right corner of the SFP to expand the configuration pane if it is collapsed by default.

### Toolbar Controls (IV Analyzer)

Use the toolbar controls to perform common tasks for Current-Voltage Analyzer. The toolbar is located at the top of the SFP.

The layout of the controls may vary depending on the device from which you launch the SFP. Some controls are hidden by default when you resize the SFP. Click the ellipsis on the toolbar to access hidden controls.

| Control | Description |
|---|---|
| **Drag and drop** | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| **Dropdown** | Click to open the current SFP in a separate browser window. To restore it to the original |

| Control | Description |
|---|---|
| | window, close the SFP by clicking [X] and launch it again. |
| **Run/Stop** | Click **Run** to start data acquisition or generation. Click **Stop** to stop data acquisition or generation. |
| **Collaboration** | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the instrument. For more information, refer to Collaborating with Your Team Members. |
| **Collapse/Expand** | Collapses or expands the configuration pane on the right side of the panel. |

## Analyzer & Hardware Settings (IV Analyzer)

Use this section of the IV Analyzer to enter basic measurement settings.

### Diode

The following settings are available when you select **Diode** for **Analyzer mode**.

| Setting | Description |
|---|---|
| **Series resistance** | Appears when Diode is selected in **Analyzer mode**. This controls an internal variable resistor in the NI ELVIS III that is connected to the DUT+ connector. This resistor limits the current through the diode under test. The steps in this control are logarithmic: 10 Ω, 100 Ω, 1 kΩ, 10 kΩ, 100 kΩ, 1 MΩ. |
| **Connections** | Connection diagram that shows how to connect the diode being measured to the NI ELVIS III. |

### Transistor

The following settings are available when you select **Transistor** for **Analyzer mode**.

| Setting | Description |
|---|---|
| **Transistor type** | This appears when Transistor is selected in **Analyzer mode**. Select NPN or PNP. |
| **Collector resistance** | Appears when Transistor is selected in **Analyzer mode**. This controls an internal variable resistor in the NI ELVIS III that is connected to the DUT+ connector. This resistor limits the current through the transistor under test. The steps in this control are logarithmic: 10 Ω, 100 Ω, 1 kΩ, 10 kΩ, 100 kΩ, 1 MΩ. |
| **Connections** | Connection diagram that shows how to connect the transistor being measured to the NI ELVIS III. This changes to reflect the selected **Transistor type**. |

## Voltage Sweep Settings (IV Analyzer)

To better display the signals on the graph, adjust the voltage sweep settings according to the parameters of the diode under test.

The following **Voltage sweep** settings are available.

| Setting | Description | Operational Limits |
|---|---|---|
| **Start** | Initial value of voltage. | Range: -10 V to 10 V<br>Resolution: 10 mV |
| **Stop** | Final value of voltage. | Range: -10 V to 10 V<br>Resolution: 10 mV |
| **Step** | Voltage interval during a voltage sweep. | Range: 10 mV to ($V_{stop}$ - $V_{start}$)<br>Resolution: 10 mV |

**Note** The settings appear for the Transistor **Analyzer mode** only.

## Current Range Settings (IV Analyzer)

To better display measured signals on the graph, adjust the current range settings according to the parameters of the diode under test.

The following **Current range** settings are available.

| Setting | Description | Operational Limits |
|---|---|---|
| **Negative** | Maximum negative value of current during the sweep. | Range: -30 mA to 0 µA<br>Resolution: 1 µA |
| **Positive** | Maximum positive value of current during the sweep. | Range: 0 µA to 30 mA<br>Resolution: 1 µA |

**Note** The settings appear for the Transistor **Analyzer mode** only.

## Collector Voltage Sweep Settings (IV Analyzer)

To better display measured signals on the graph, adjust the collector voltage sweep settings according to the parameters of the transistor under test.

The following **Collector voltage sweep** settings are available.

| Setting | Description | Operational Limits |
|---|---|---|
| **Vc start** | Initial collector voltage. | Range: -10 V to 10 V<br>Resolution: 10 mV |
| **Vc stop** | Final collector voltage. | Range: -10 V to 10 V<br>Resolution: 10 mV |
| **Vc step** | Size of the collector voltage steps during the sweep. | Range: 10 mV to ($Vc_{stop}$ - $Vc_{start}$)<br>Resolution: 10 mV |
| **Ic limit** | Maximum collector current during the sweep. | Range: 0 µA to 30 mA<br>Resolution: 1 µA |

**Note** The settings appear for the Transistor **Analyzer mode** only.

## Base Current Sweep Settings (IV Analyzer)

To better display measured signals on the graph, adjust the base current sweep settings according to the parameters of the transistor under test.

The following **Base current sweep** settings are available.

| Setting | Description | Operational Limits |
|---|---|---|
| **Ib start** | Initial base current. | Range: 0 µA to 300 µA<br>Resolution: 1 µA |
| **Ib step** | Current interval during a base current sweep. | Range: 1 µA to 200 µA<br>Resolution: 1 µA |
| **Curve count** | Number of curves in the sweep. | Range: 1 to 16<br>Resolution: 1 |

**Note** The settings appear for the Transistor **Analyzer mode** only.

## Display Window (IV Analyzer)

Use **Cursors** to measure a given point on the signal trace, or compare the differences between two points.

| Cursor Mode | Description |
|---|---|
| **Off** | Turns off the cursors. |
| **Track** | Cursor positions are restricted to the acquired data points on a trace. |
| **Manual** | Cursors positions are not restricted to the acquired data points on a trace. |

When **Cursors** is set to **Manual** or **Track**, a table appears above the display with the following data:

| Column | Description |
|---|---|
| **C1** | Displays the Voltage and Current readings for Cursor 1. |
| **C2** | Displays the Voltage and Current readings for Cursor 2. |
| **\|Δ Value\|** | Displays the absolute value of the Voltage and Current readings for C2 - C1. |

Display Controls (IV)

Use the display controls to manipulate the display of the data on the graph.

Click  to access these controls:

| Control | Description |
|---|---|
| **Zoom all** | Click to view all data on the graph. Useful when you have zoomed or panned the display. |
| **Autoscale** | Select **On** to change the scale to display all the data. Select **Off** to keep the scale consistent. Data may flow off the display with this setting, or not fill the display. |

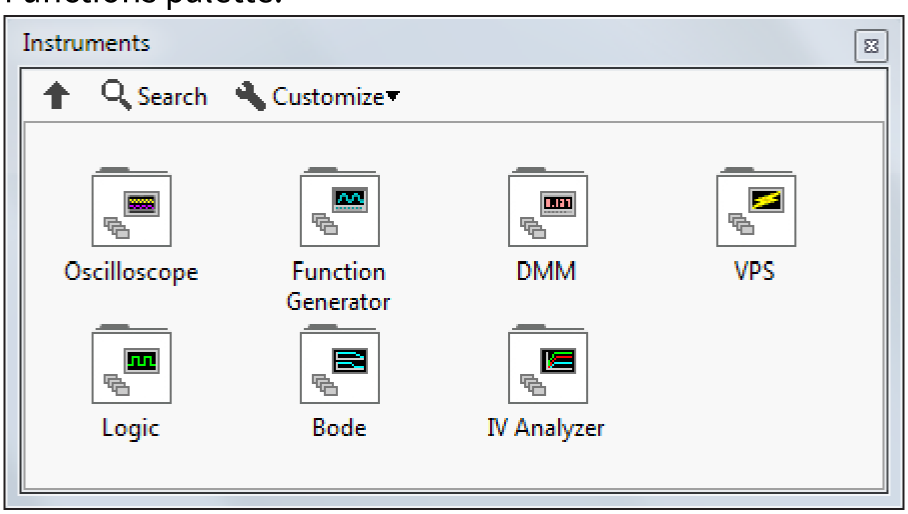
## Zooming and Panning

Use the mouse to do the following:

| Mouse Action | Result |
|---|---|
| Scroll mouse wheel | Zooms in/out. |
| Hold down left button and drag | Pans left/right, up/down. |

## Programming the IV Analyzer

You can use the IV Analyzer VIs installed with the LabVIEW ELVIS III Toolkit to program the Current-Voltage (IV) Analyzer instrument on the NI ELVIS III. Find the IV Analyzer VIs on the **Academic I/O** » **Instruments** » **IV Analyzer** subpalette on the Functions palette.



## Related information:

- IV Analyzer VIs

# Current-Voltage (IV) Analyzer Specifications

Specifications are **Typical** unless otherwise noted.

| **2 wire impedance analyzer** | |
|---|---|
| Current range | ±30 mA |
| Voltage sweep range | ±10 V |
| Excitation frequency | 1 Hz to 15 MHz |
| **2/3 wire current-voltage analyzer** | |
| Supported devices | Diodes, NPN and PNP bipolar transistors |
| Base current range | ±1 mA |
| Maximum collector current | ±30 mA |
| Maximum collector voltage | ±10 V |

**Table 9.** Capacitance Measurement Range

| Range | Effective Frequency | Effective Test Resistance |
|---|---|---|
| 50 pF to 500 pF | 10 kHz | 100 kΩ |
| 500 pF to 5 nF | 1 kHz | 10 kΩ |
| 5 nF to 50 nF | 1 kHz | 10 kΩ |
| 50 nF to 500 nF | 1 kHz | 1 kΩ |
| 500 nF to 5 µF | 1 kHz | 1 kΩ |
| 5 µF to 50 µF | 1 kHz | 100 Ω |
| 50 µF to 500 µF | 100 Hz | 100 Ω |

| | |
|---|---|
| Capacitance measurement accuracy | 1% of range |

**Table 10.** Inductance Measurement Range

| Range | Effective Frequency | Effective Test Resistance |
|---|---|---|
| 10 µH to 100 µH | 100 kHz | 100 Ω |
| 100 µH to 1 mH | 10 kHz | 100 Ω |
| 1 mH to 10 mH | 10 kHz | 1 kΩ |
| 10 mH to 100 mH | 1 kHz | 1 kΩ |

| | |
|---|---|
| Inductance measurement accuracy | 1% of range |

# Logic Analyzer and Pattern Generator

The Logic Analyzer and Pattern Generator (Logic) allows you to record, analyze, and generate digital signals.

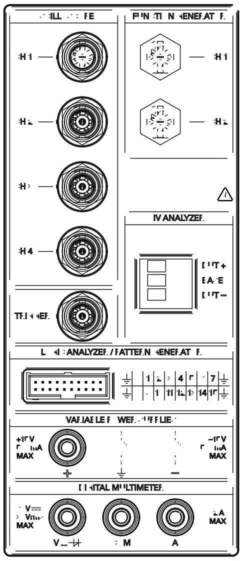
Find an introductory video about using the instrument on ni.com.

Find out more about the Logic Analyzer and Pattern Generator in the following topics:

- Connecting Signals to the Logic Analyzer and Pattern Generator
- Generating and Measuring a Digital Signal
- Configuration Options (Logic)
- TDMS File Format (Logic)
- Programming the Logic Analyzer and Pattern Generator
- Logic Analyzer and Pattern Generator Specifications

## Connecting Signals to the Logic Analyzer and Pattern Generator

The Logic Analyzer and Pattern Generator on the NI ELVIS III provides signals from a 20-pin connector on the instrument panel.



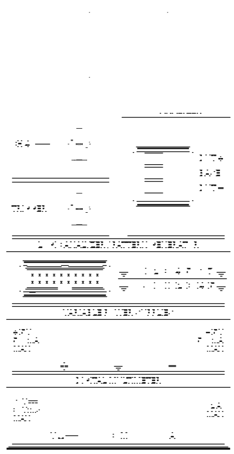

> **Note** NI recommends using the cables provided in the Cables and Accessory Kit (786520-01) when connecting to the Logic Analyzer and Pattern Generator.

## Generating and Measuring a Digital Signal

This example illustrates how to perform a loopback test with the Logic Analyzer and Pattern Generator (Logic). In the example, you use the Pattern Generator to generate a digital signal, and use the Logic Analyzer to read the signal.

The procedure for generating and measuring other types of digital signals is similar. Refer to the Configuration Options (Logic) section for more information.

1. Connect a jumper wire between the Logic 0 line and the Logic 1 line on the NI ELVIS III, as shown in the following diagram:



Refer to Connecting Signals to the Logic Analyzer and Pattern Generator for more details about connecting to the Logic instrument.

2. Launch the Logic Soft Front Panel (SFP).
3. Configure the Pattern Generator to generate a clock signal on the Logic 0 line.
4. Configure the Logic Analyzer to measure the input signal on the Logic 1 line.
5. Click **Run**.
   You can now see two square waves in the display window, one of which is the output signal on the Logic 0 line, and the other is the input signal on the Logic 1 line.
6. If necessary, adjust the trigger settings to stabilize the waveform in the display window and adjust **Time per division** to change the number of periods displayed.

> **Note** You may hear clicking sounds from the NI ELVIS III while operating the SFP. This is expected behavior and is due to the switching of internal relays.

### Related information:

- Launching the Soft Front Panels

## Configuration Options (Logic)

Use configuration options on the Logic Analyzer and Pattern Generator (Logic) Soft Front Panel (SFP) to control how to acquire, display, and analyze digital signals and generate digital patterns. For example, you can configure the lines from which you want to read the digital signals.

- Toolbar Controls
- Trigger Settings
- Horizontal & Acquisition Settings
- Logic Analyzer Settings
- Pattern Generator Settings

### Toolbar Controls (Logic)

Use the toolbar controls to perform basic tasks for the Logic Analyzer and Pattern Generator (Logic).

| Control | Description |
|---|---|
| **Automatic/Default** | When you select **Automatic**, the instrument automatically chooses some settings, including channel and trigger settings, that are best suited to the input signal. When you select **Default**, the settings fall back to default values. |
| **Run/Stop** | Click **Run** to start continuous data acquisition. Click **Stop** to stop data acquisition. To take a single-sweep measurement, click **Single** instead. |
| **Single** | Takes a single-sweep measurement. |
| **Status** | Indicates the status of the Logic instrument.<br>• **Waiting**—The instrument is waiting for trigger.<br>• **Triggered**—The instrument is triggered.<br>• **Completed**—The instrument completes one acquisition.<br>• **Stopped**—The instrument is stopped. |

| Control | Description |
|---|---|
| **Screenshot** | Takes a snapshot of the graph. |
| **Export** | Exports acquired data to file formats compatible with LabVIEW. The instrument generates a TDMS file. Refer to TDMS File Format (Logic) for more information about the exported TDMS file format. |
| **Help** | Contains links to related help documentation. |
| **Collapse/Expand** | Collapses or expands the configuration pane on the right side of the panel. |

## Trigger Settings (Logic)

You can configure a trigger by using the trigger settings.

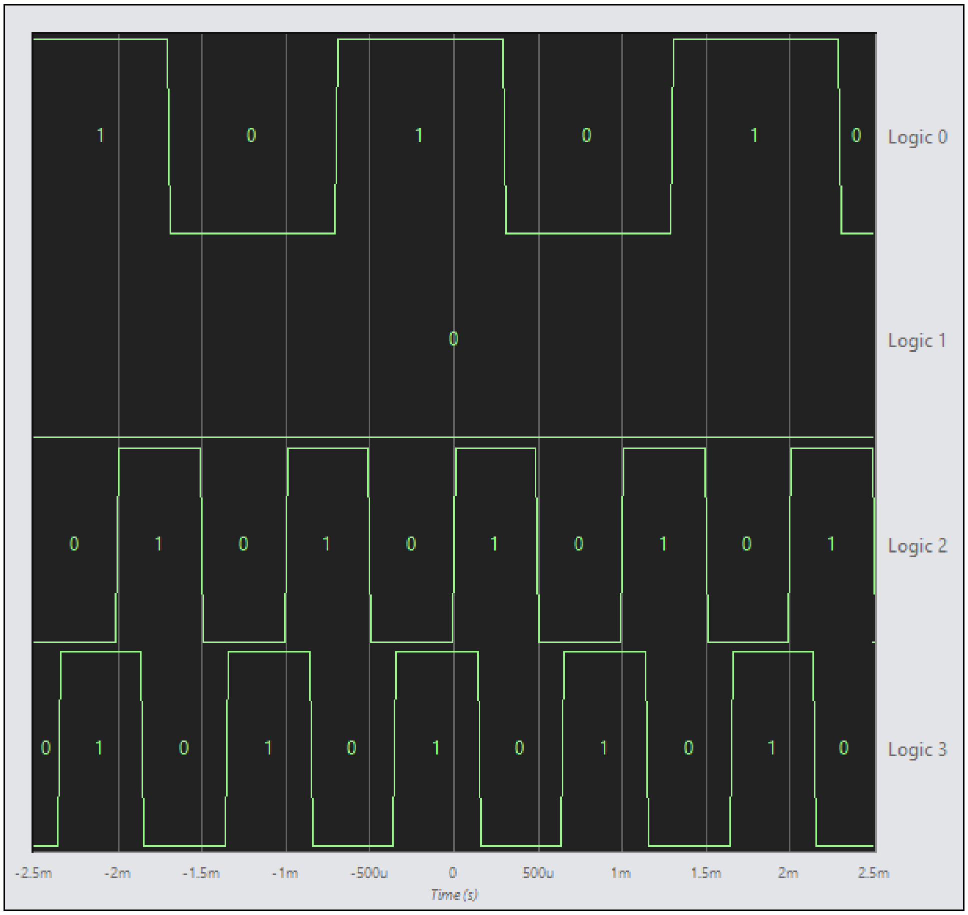
You can configure some settings directly on the panel. To access additional settings, click the configuration button on the top right side of the section.

| Setting | Description |
|---|---|
| **Force** | Manually triggers the Logic instrument. This setting is available on the panel only. |
| **Mode** | Mode in which the Logic instrument responds to a trigger.<br>▪ **Automatic**—The instrument waits momentarily for any occurrence of trigger condition before automatically triggering.<br>▪ **Normal**—The instrument waits indefinitely for any occurrence of trigger condition. |
| **Acquisition delay** | Decides whether the instrument waits for a certain period of time after a trigger is detected before it starts sampling. Enable **Acquisition delay** when there is a delay between when the trigger occurs and when the desired data is generated, or when you need a high sampling rate for the desired acquisition window. |

| Setting | Description |
|---|---|
| **Position** | The time between the trigger position and the middle of the display pane. The value of **Position** changes when you move the trigger horizontally. |
| **Type** | Type of the trigger. When you select **Digital**, the instrument is triggered when the trigger signals meet all level triggering conditions you define, and meet at least one of the edge triggering conditions. If you do not set any edge triggering condition, the trigger signals only need to meet all level triggering conditions to activate a trigger. |
| **Line** | Trigger signal. |
| **Name** | Name of the trigger signal. |
| **State** | Triggering condition. |

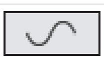
For example, if you configure a trigger as the following table shows, the Logic Analyzer will be triggered off when DIO0 is high, DIO1 is low, and DIO2 is rising or DIO3 is falling, as the screenshot illustrates.

| Setting | Value |
|---|---|
| **Type** | **Digital** |
| **DIO0** | 1 |
| **DIO1** | 0 |
| **DIO2** | Rising |
| **DIO3** | Falling |




## Horizontal & Acquisition Settings (Logic)

You can configure the data display using the **Horizontal & Acquisition** settings.

| Setting | Description |
|---|---|
| **Time per division** | The horizontal scale of the display. |
| **Expand**  | Expands the trace by decreasing **Time/div**. |
| **Shrink**  | Contracts the trace by increasing **Time/div**. |

## Logic Analyzer Settings (Logic)

You can use the **Logic Analyzer Settings** window to configure the Logic Analyzer on the NI ELVIS III.

- Line Settings
- Bus Settings

Line Settings (Logic Analyzer)

You can use the **Lines** tab to add, remove, and configure digital lines for the Logic Analyzer. Access the **Lines** tab by clicking the icon in the **Logic Analyzer** section and selecting **Lines**.

| Setting | Description |
|---|---|
| **Status** | Enables or disables signal acquisition on the digital line. |
| **Name** | Name of the digital line. |
| **Pin number** | Displays the pin number of the digital line you have added. |

Bus Settings (Logic Analyzer)

You can use the **Buses** tab to add, remove, and configure buses for the Logic Analyzer. Access the **Buses** tab by clicking the icon in the **Logic Analyzer** section and selecting **Buses**.

The following configuration options are available when **Mode** is **Custom**. The **Custom** mode allows you to analyze a customized set of data lines.

| Setting | Description |
|---|---|
| **Status** | Enables or disables signal acquisition on the bus lines. |
| **Name** | Bus name. |
| **Mode** | Bus mode. |
| **Bus lines** | Data lines. |
| **Clock** | Clock line. When you select **None**, the Logic Analyzer samples the data line only when the data value changes. Selecting a data line for **Clock** or **Enable** adds that line to the display, but the setting does not affect the signal acquisition in any way. |
| **Enable** | Optional enable line. Selecting a data line for **Clock** or **Enable** adds that line to the display, but the setting does not affect the signal acquisition in any way. |

| Setting | Description |
| --- | --- |
| **Clock active** | Sampling edge of the clock line. The Logic Analyzer samples the data line when the clock line is on the edge you specify in **Clock active**. |
| **Enable active** | Polarity of the enable line. The Logic Analyzer samples the data line only when the enable line meets the condition you specify in **Enable active**. |
| **Endianness** | Determines the bit order during data interpretation. |

The following configuration options are available when **Mode** is **I2C**:

| Setting | Description |
| --- | --- |
| **Status** | Enables or disables signal acquisition on the bus lines. |
| **Name** | Bus name. |
| **Mode** | Bus mode. |
| **Format** | Display format of the interpreted data. |
| **Clock (SCL)** | Clock signal. |
| **Data (SDA)** | Data signal. |

The following configuration options are available when **Mode** is **SPI**:

| Setting | Description |
| --- | --- |
| **Status** | Enables or disables signal acquisition on the bus lines. |
| **Name** | Bus name. |
| **Mode** | Bus mode. |
| **Format** | Display format of the interpreted data. |
| **MOSI** | Master output, slave input line. |
| **MISO** | Master input, slave output line. |
| **SCLK** | Serial clock line. |
| **SS** | Slave select line. |
| **Bits per transfer** | Number of bits in one transmission word. |

| Setting | Description |
| --- | --- |
| **Bit order** | Determines the bit order during data interpretation. |
| **Clock polarity (CPOL)** | Determines the idle state of the clock line. |
| **Clock phase(CPHA)** | Determines whether the data line is sampled on the leading or tailing edge of the clock line. |
| **SS active** | Determines whether the data is sampled when the SS line is high or low. |

The following configuration options are available when **Mode** is **UART**:

| Setting | Description |
| --- | --- |
| **Status** | Enables or disables signal acquisition on the bus lines. |
| **Name** | Bus name. |
| **Mode** | Bus mode. |
| **Format** | Display format of the interpreted data. |
| **Data** | Data line. |
| **Bits per transfer** | Number of bits in one transmission word. |
| **Polarity** | Determines whether to invert the bits that the Logic Analyzer reads from the data line. |
| **Parity** | Type of the parity bit. Select **None** if no parity bit is used. |
| **Baud rate** | Sampling rate of the Logic Analyzer. |
| **Stop bits** | Number of the stop bits in the data line. |

## Pattern Generator Settings (Logic)

You can use the **Pattern Generator Settings** window to configure the Pattern Generator on the NI ELVIS III.

- Line Settings
- Bus Settings

### Line Settings (Pattern Generator)

You can use the **Lines** tab to add, remove, and configure digital lines for the pattern generator. Access the **Lines** tab by clicking the ⚙ icon in the **Pattern Generator** section and selecting **Lines**.

The following configuration options are available when **Mode** is **Clock**:

| Setting | Description |
|---|---|
| **Status** | Enables or disables signal generation on the digital line. |
| **Name** | Name of the digital line. |
| **Mode** | Type of the output signal.<br>▪ **Clock**—The Pattern Generator generates a square wave with the frequency and duty cycle you define. You can use the square wave as clock signals.<br>▪ **Custom**—The Pattern Generator generates signals defined by the data file that you import. |
| **Frequency** | Frequency of the output signal. |
| **Duty cycle** | Percentage of the pulse width in the total period of the signal. |

The following configuration options are available when **Mode** is **Custom**:

| Setting | Description |
|---|---|
| **Status** | Enables or disables signal generation on the digital line. |
| **Name** | Name of the digital line. |
| **Mode** | Type of the output signal.<br>▪ **Clock**—The Pattern Generator generates a square wave with the frequency and duty cycle you define. You can use the square wave as clock signals. |

| Setting | Description |
| --- | --- |
| | ■ **Custom**—The Pattern Generator generates signals defined by the data file that you import. |
| **Source file** | Path to the data file that you import. You can import TDMS files only and the files must contain digital waveform data only. You can load TDMS files generated by LabVIEW or the Logic Analyzer and Pattern Generator in the NI ELVIS III. |
| **Source file channel** | Channel you want to import. If **Source file** contains data of multiple channels, you can use **Source file channel** to select which channel of data you want to import to the Pattern Generator. |
| **Update rate** | Number of samples that the instrument generates per second. Selecting a file in **Source file** auto-populates **Update rate** with the update rate defined by the source file. You can change the update rate by entering a new value. |

## Bus Settings (Pattern Generator)

You can use the **Buses** tab to add, remove, and configure buses for the pattern generator. Access the **Buses** tab by clicking the icon in the **Pattern Generator** section and selecting **Buses**.

| Setting | Description |
| --- | --- |
| **Status** | Enables or disables signal generation on the bus lines. |
| **Name** | Name of the bus line. |
| **Mode** | Type of the output signal. When **Mode** is **Custom**, the Pattern Generator generates the signal defined in the data file(s) you import. |
| **Bus lines** | Table containing bus lines that you can add, remove, or define the name for. |

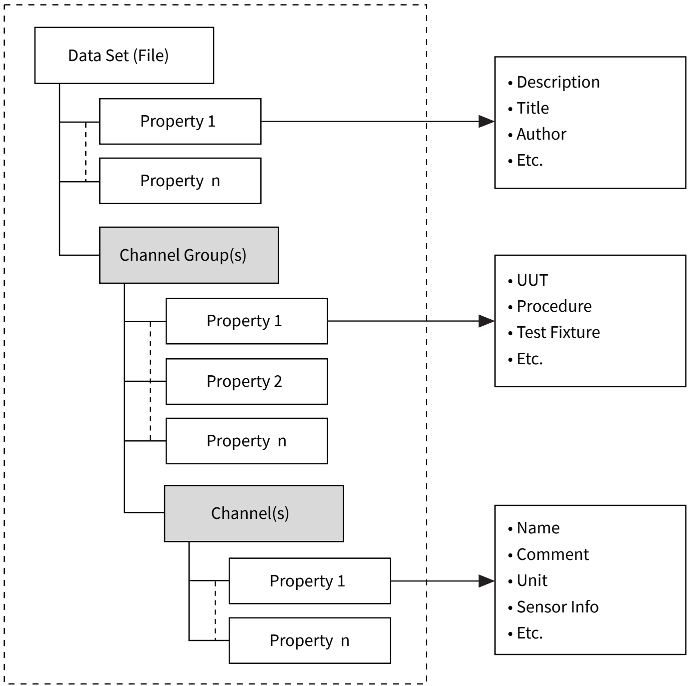
| Setting | Description |
|---|---|
| | **Note** After you add bus lines to the table, you must click one bus line to enable the corresponding **Source file** and **Source file channel** settings. |
| **Source file** | Path to the data file that you import. You can import TDMS files only and the files must contain digital waveform data only. You can load TDMS files generated by LabVIEW or the Logic Analyzer and Pattern Generator in the NI ELVIS III. |
| **Source file channel** | Channel you want to import. If **Source file** contains data of multiple channels, you can use **Source file channel** to select which channel of data you want to import to the Pattern Generator. |
| **Update rate** | Number of samples that the instrument generates per second. Selecting the first line in **Bus lines** and selecting a file in **Source file** auto-populates **Update rate** with the update rate defined by the source file. When you select lines except for the first line in **Bus lines** and select a file in **Source file**, **Update rate** does not change. You can change the update rate by entering a new value. |

## TDMS File Format (Logic)

You can export the data acquired by the Logic Analyzer to TDMS files.

The TDMS file format is supported by the TDM data model. The TDM data model arranges the data in three hierarchical levels: file, channel groups, and channels.

The following figure illustrates the TDM data model:




The TDMS files that you export from the SFPs contain some custom properties, for example, **product-name**. For all TDMS files that you generate using the NI ELVIS III instruments, the value of **product-name** is always NI ELVIS III. These TDMS files also have designated names for the channel group and channels.

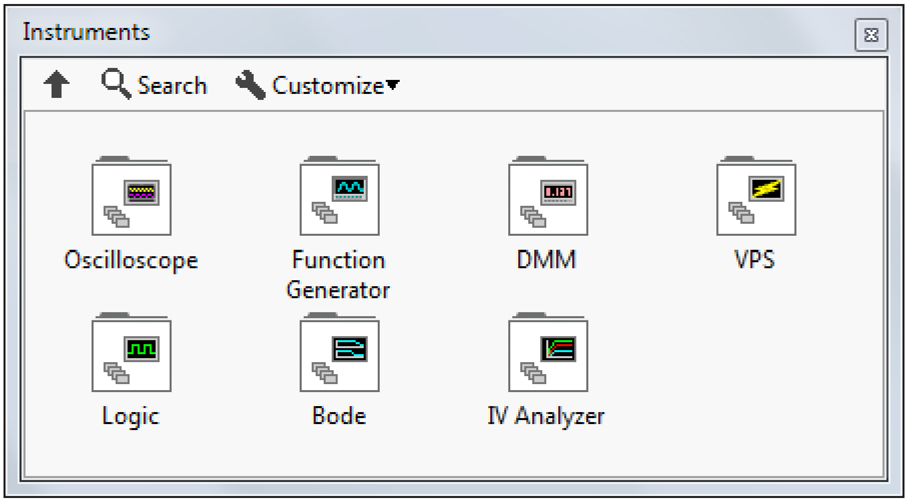
The TDMS files you export from the Logic Analyzer have the following data structure:

- **channel_group**: Logic Analyzer
  - **channel**: Bus 0:CLK
  - **channel**: Bus 0:MISO
  - **channel**: Bus 0:MOSI
  - **channel**: Logic 0
  - **channel**: Logic 1
  - ...

The TDMS files list enabled channels only. The order of the channels in the TDMS files matches the top-to-bottom order of the channels on the configuration panel on the right side of the SFP.

## Programming the Logic Analyzer and Pattern Generator

You can use the Logic Analyzer and Pattern Generator VIs installed with the LabVIEW ELVIS III Toolkit to program the Logic Analyzer and Pattern Generator (Logic) instrument on the NI ELVIS III. Find the Logic Analyzer and Pattern Generator VIs on the **Academic I/O** » **Instruments** » **Logic Analyzer and Pattern Generator** subpalette on the Functions palette.




### Related information:

- Logic Analyzer and Pattern Generator VIs

## Logic Analyzer and Pattern Generator Specifications

Specifications are **Typical** unless otherwise noted.

| | |
|---|---|
| Number of channels | 16 |
| Maximum sampling rate (per channel) | 100 MS/s |
| Logic level | 5 V compatible LVTTL input; 3.3 V LVTTL output |
| Pull down | 1 MΩ |
| Direction control | Individually programmable as Logic Analyzer or Pattern Generator |
| **Input logic levels** | |
| **Input low voltage, $V_{IL}$** | |
| Minimum | 0 V |

| | |
|---|---|
| Maximum | 0.8 V |
| **Input high voltage, $V_{IH}$** | |
| Minimum | 2.0 V |
| Maximum | 5.25 V |
| **Output logic levels** | |
| **Output low voltage, $V_{OL}$ sinking 4 mA** | |
| Minimum | 0 V |
| Maximum | 0.4 V |
| **Output high voltage, $V_{OH}$ sourcing 4 mA** | |
| Minimum | 2.4 V |
| Maximum | 3.465 V |
| Protection | Short-circuit to ground |

# Digital I/O

The Digital I/O instrument acquires and generates digital signals by using the digital input/output on the NI ELVIS III.

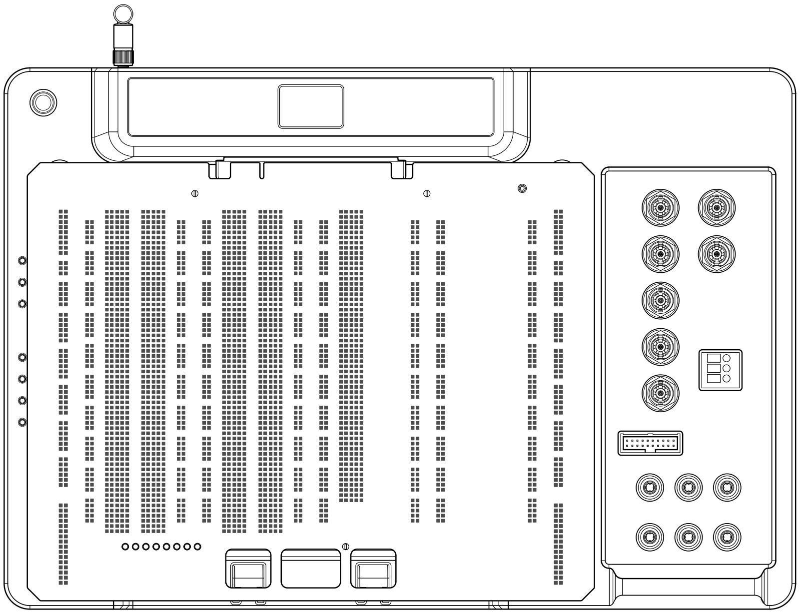
Find out more about the Digital I/O instrument in the following topics:

- Connecting Signals to Digital I/O
- Monitoring the Prototyping Board Switch with Digital I/O
- Controlling the LED with Digital I/O
- Configuration Options
- Programming Digital I/O with LabVIEW
- Digital I/O Specifications

## Monitoring the Prototyping Board Switch with Digital I/O

This example illustrates how to use the Digital I/O to monitor the switch.

1. Install the NI ELVIS III prototyping board onto the NI ELVIS III workstation.
2. Connect a jumper wire between Bank A, Channel DIO 0 (A/DIO 0) and SW0 A, and between DGND and SW0 B, as shown in the following diagram:



Refer to Connecting Signals to Digital I/O for more details about connecting to the digital input/output.

3. Press the application board power button on the workstation to power on the application board.

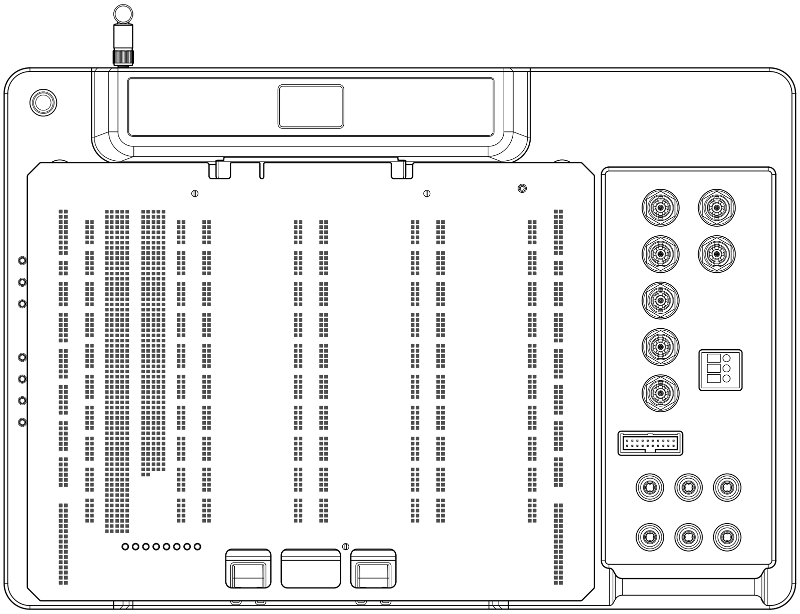
> **Caution** To ensure safety, you must connect the wires before powering on the application board.

4. Launch the Digital I/O Soft Front Panel (SFP).
5. Configure **A/DIO7:0** to the **Read** mode.
6. Click **Run** to start reading data from the SW0 A switch.
7. Turn on/off the SW0 A switch on the workstation.
   The readback value of A/DIO 0 is 1 when you turn on the switch and 0 when you turn off the switch.

## Controlling the LED with Digital I/O

This example illustrates how to use the Digital I/O to control the LED.

1. Install the NI ELVIS III prototyping board onto the NI ELVIS III workstation.
2. Connect a jumper wire between Bank A, Channel DIO 0 (A/DIO 0) and LED 0, as shown in the following diagram:



Refer to Connecting Signals to Digital I/O for more details about connecting to the digital input/output.

3. Press the application board power button on the workstation to power on the application board.

> **Caution** To ensure safety, you must connect the wires before powering on the application board.

4. Launch the Digital I/O Soft Front Panel (SFP).
5. Configure the **A/DIO7:0** to the **Write** mode.
6. Click **Run** to start writing data from A/DIO 0 to LED 0.
7. Change the output status of A/DIO 0 by changing **Value** or **Line status**. The LED is on when you configure A/DIO 0 to 1 and off when you configure A/DIO 0 to 0.

## Configuration Options (Digital I/O)

Use configuration options on the Digital I/O Soft Front Panel (SFP) to control how to acquire and generate digital signals. For example, you can configure it to work as a 8-bit digital counter.

## Toolbar Controls

Use toolbar controls to perform common tasks for the Digital I/O. The toolbar is located at the top of the SFP.

| Control | Description |
| --- | --- |
| **Drag and drop** | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| **Dropdown** | Click to open the current SFP in a separate browser window. To restore it to the original window, close the SFP by clicking and launch it again. |
| **Run/Stop** | Click **Run** to start data acquisition or generation. Click **Stop** to stop data acquisition or generation. |
| **Collaboration** | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the instrument. For more information, refer to Collaborating with Your Team Members. |

## Read Mode

Configure a group of lines to the **Read** mode to read digital data from the lines. You can view the readback value in real time and change the format of the value.

| Setting | Description |
| --- | --- |
| **Readback value** | Displays the real-time readback value for the group of lines. |
| **Format** | Configures the format of the readback value. You can choose from the following four formats: |

| Setting | Description |
|---|---|
| | ▪ Binary<br>▪ Octal<br>▪ Decimal<br>▪ Hexadecimal |

**Note** A line is an individual signal. It refers to a physical terminal. The data that the line carries are called bits, or binary values, which are either 0 (low) or 1 (high).

## Write Mode

Configure a group of lines to the **Write** mode to write digital data to the lines. You can configure the value of the lines using **Value** or **Line status**.

| Setting | Description |
|---|---|
| **Value** | Configures the value for the group of lines. You can choose the format from the drop-down list by clicking the radix box on the left side. |
| **Line status** | Toggles the bit of each line. Click the toggle to change a bit from 0 to 1 or from 1 to 0. |

## Pattern (Auto) Mode

Configure a group of lines to the **Pattern (Auto)** mode to write digital data to the lines at a fixed rate and in a desired pattern mode. You cannot change the configuration options when the SFP is running.

| Setting | Description |
|---|---|
| **Readback value** | Displays the real-time readback value for the group of lines. |
| **Value** | Value written to the lines when the SFP runs. You can enter a value in binary, decimal, or hex format. Choose the format from the drop-down list by clicking the radix box on the left side. |

| Setting | Description |
| --- | --- |
| **Mode** | ▪ Shift—Moves all the bits to the left or right by one and places 0 in the vacated bit on either end at the configured update rate.<br>▪ Rotate—Moves all the bits to the left by one and moves the first bit to the final position, or performs the opposite operation, at the configured update rate.<br>▪ Count—Increases/decreases all the bits by one at the configured update rate.<br>▪ Invert—Flips the value of all the bits to the opposite at the configured update rate.<br>To learn more about the different modes, refer to the pattern mode examples at the end of the this topic. |
| **Update rate** | Frequency of updating the value to the lines. The valid value must be an integer ranging from 1 Hz to 10 Hz. |

## Pattern (Manual) Mode

Configure a group of lines to the **Pattern (Manual)** mode to write digital data to the lines in a desired pattern mode. You can change the value and mode manually when the SFP is running.

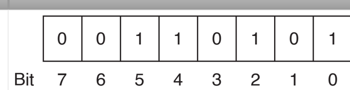
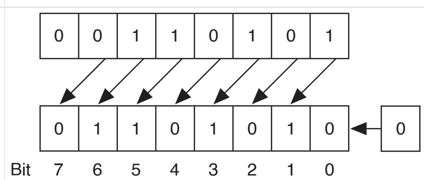
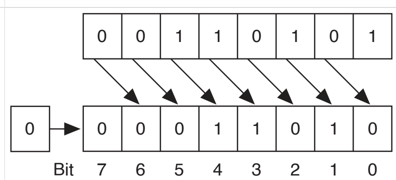
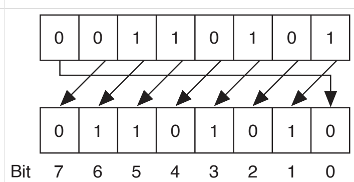
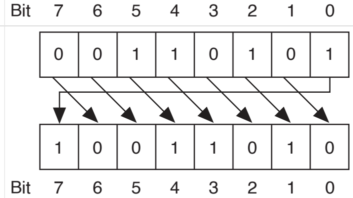
| Setting | Description |
| --- | --- |
| **Readback value** | Displays the readback value for the group of lines. |
| **Value** | Value written to the lines when the SFP runs. You can enter a value in binary, decimal, or hex format. Choose the format from the drop-down list by clicking the radix box on the left side. |
| **Shift** | Moves all the bits to the left or right by one and places 0 in the vacated bit on either end. |

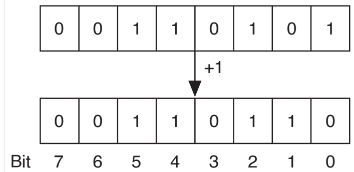
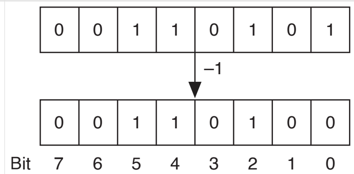
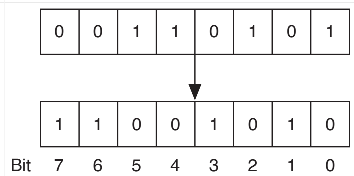
| Setting | Description |
| --- | --- |
| **Rotate** | Moves all the bits to the left by one and moves the first bit to the final position or performs the opposite operation. |
| **Count** | Increases/decreases all the bits by one. |
| **Invert** | Flips the value of all the bits to the opposite. |

**Note** To learn more about the different modes, refer to the following pattern mode examples.

## Pattern Mode Examples

The following examples demonstrate how each pattern mode works.

| Mode | Example |
| --- | --- |
| Initial value (in binary) |  |
| Shift left |  |
| Shift right |  |
| Rotate left |  |
| Rotate right |  |

| Mode | Example |
|---|---|
| Count up |   |
| Count down |   |
| Invert |   |

# Data Logger

The Data Logger acquires, records, and stores analog signals by using the analog input on the NI ELVIS III.

Find out more about the Data Logger in the following topics:

- Connecting Signals to Analog Input
- Logging a Data Set
- Configuration Options
- TDMS File Format
- Programming Analog Input with LabVIEW
- Analog Input Specifications

**Related reference:**

- Choosing the Instruments

# Logging a Data Set

This example illustrates how to use the Data Generator (Data Gen) to generate a data set and use the Data Logger to log the data.

1. Install the NI ELVIS III prototyping board onto the NI ELVIS III workstation.

2. Connect a jumper wire between Bank A, Channel AI0 (A/AI0) and A/AO0, and between A/AI1 and A/AO1. Refer to Connecting Signals to Analog Input for more details about connecting to the analog input.
3. Press the application board power button on the workstation to power on the application board.

> **Caution** To ensure safety, you must connect the wires before powering on the application board.

4. Launch the Data Gen Soft Front Panel (SFP).
5. Configure the Data Gen to generate data on A/AO0 and A/AO1.
6. Click **Run** to start generating data.
7. Launch the Data Logger SFP.
8. Configure the Data Logger to log the input signal on A/AI0 and A/AI1.
9. Click **Run** to start logging the input signal.
   A waveform appears in the display window.
10. Adjust the display and trigger settings as needed.

> **Note** You must stop the instrument before adjusting the trigger settings.

The Data Logger automatically saves a data file to your local computer when the logging stops.

## Configuration Options (Data Logger)

Use configuration options on the Data Logger Soft Front Panel (SFP) to control how to log analog signals to a file. For example, you can choose to log large amount of data to a USB drive.

- Toolbar Controls
- Channels Settings
- Acquisition Setting
- Trigger Settings
- Log Settings

- Display Window

**Note** Click the expand arrow near the upper right corner of the SFP to expand the configuration pane if it is collapsed by default.

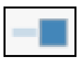
## Toolbar Controls (Data Logger)

Use the toolbar controls to perform common tasks for the Data Logger. The toolbar is located at the top of the SFP.

| Control | Description |
|---|---|
| **Drag and drop** | Drags and drops the SFP to the desired position within the browser window. Large SFPs such as the Oscilloscope are always located to the left of small SFPs such as the Variable Power Supplies (VPS). Your window size decides whether the SFPs are arranged vertically or horizontally. |
| **Dropdown** | Click to open the current SFP in a separate browser window. To restore it to the original window, close the SFP by clicking and launch it again. |
| **Run/Stop** | Click **Run** to start data acquisition or generation. Click **Stop** to stop data acquisition or generation. |
| **Collaboration** | Indicates whether you are controlling or viewing the instrument, and whether others are using the instrument simultaneously. Click the button to request or release your control over the instrument. For more information, refer to Collaborating with Your Team Members. |
| **Collapse/Expand** | Collapses or expands the configuration pane on the right side of the panel. |

## Channels Settings (Data Logger)

You can use the channels settings to add, remove, and configure analog input channels for the Data Logger.

| Setting | Description |
|---|---|
| **Add/Remove** | Add a channel by selecting the checkbox beside the channel. Remove a channel by clearing the checkbox. Once you add a channel, the Data Logger automatically logs the data to the channel and saves the data to your local computer. |
| **Status**  | Shows/Hides the channel's trace on the graph. The control also decides whether Measurements Live exports the data on this channel when you click the Export Data button on the top right side of Measurements Live. |
| **Range** | Voltage input range for the channel. Refer to the Gain Selector for more information about input range. |
| **Vertical position** | Vertical positioning of the waveform on the graph. Use **Vertical position** to move the waveform up and down the graph. The Data Logger does not apply **Vertical position** to the input signal and this setting does not affect the logged data. The range of **Vertical position** is -10 V to 10 V. |
| **Channel name** | Name of the analog input channel. The channel name in the logged data file matches the name you specify for **Channel name**. |

## Acquisition Setting (Data Logger)

You can configure the data acquisition by using the **Acquisition** setting.

**Sample rate** specifies the number of samples acquired from the input signal per second. The unit is samples per second (S/s). The maximum sample rate per channel is 100 kS/s divided by the number of open channels. For example, when only one channel is open, the valid sample rate for this channel is between 200 mS/s and 100 kS/s. When four channels are open, the valid sample rate per channel is between 200 mS/s and 25 kS/s.

## Trigger Settings (Data Logger)

You can configure the start and stop triggers by using the trigger settings. Once a trigger occurs, the Data Logger starts or stops logging data for all open channels.

## Start on

The Data Logger offers the following types of start trigger.

| Trigger Type | Description |
|---|---|
| Immediate | Starts logging immediately after you click **Run**. |
| Delay | Starts logging after the period of time, in seconds, that you specify. |
| Rises above value | Starts logging when the input signal rises above the specified value. The range for **Value** equals the input range you specified for the corresponding channel. |
| Falls below value | Starts logging when the input signal falls below the specified value. The range for **Value** equals the input range you specified for the corresponding channel. |
| Enters range | Starts logging when the input signal falls within the specified range. The valid range equals the input range you specified for the corresponding channel. |
| Leaves range | Starts logging when the input signal falls outside the specified range. The valid range equals the input range you specified for the corresponding channel. |

## Stop on

The Data Logger stops logging when one of the following three conditions are met:

- You click **Stop**.
- A buffer overflow occurs, that is, the aggregate number of logged samples exceeds the buffer size. The buffer size is 3,000,000 on the computer and 2,000,000 on mobile devices. If you choose to log data to a USB drive, the buffer size depends on the size of your USB storage.

- A stop trigger occurs.

The Data Logger offers the following types of stop trigger.

| Trigger Type | Description |
|---|---|
| **No trigger** | Stops logging only when you click **Stop** or a buffer overflow occurs. |
| **Duration** | Stops logging after the period of time, in seconds, that you specify. $\text{Maximum Duration} = \frac{\text{Buffer Size}}{\text{Number of Open Channels} \times \text{Sample Rate}}$ $\text{Maximum Duration} = \frac{\text{Buffer Size}}{\text{Number of Open Channels} \times \text{Sample Rate}}$ |
| **Rises above value** | Stops logging when the input signal rises above the specified value. The range for **Value** equals the input range you specified for the corresponding channel. |
| **Falls below value** | Stops logging when the input signal falls below the specified value. The range for **Value** equals the input range you specified for the corresponding channel. |
| **Enters range** | Stops logging when the input signal falls within the specified range. The valid range equals the input range you specified for the corresponding channel. |
| **Leaves range** | Stops logging when the input signal falls outside the specified range. The valid range equals the input range you specified for the corresponding channel. |
| **Number of samples** | Stops logging when the number of logged samples per channel exceeds the value you specify. The maximum value for **Number of samples** equals the buffer size divided by the number of open channels. |

## Log Settings (Data Logger)

You can save the logged file to your local computer or a USB drive plugged in the NI ELVIS III by using the log settings.

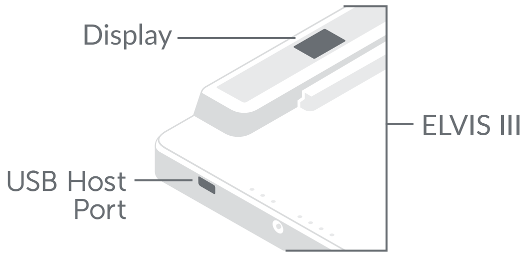
| Setting | Description |
|---|---|
| **File format** | File format of the logged file. You can save the logged file in the CSV or TDMS format. |
| **File location** | Location where you want to save your logged file. You can save the logged file to your local computer or the USB storage you plugged in the NI ELVIS III. |
| **Filename** | Name of the logged file. A timestamp (YYYYMMDD_HHMMSS) is appended to the filename you entered. A valid filename can be up to 200 characters long and can contain only letters, numbers, and the following special characters: underscore (_), hyphen (-), and period (.). This option is available only if you select **USB storage plugged in ELVIS III** as the file location. |

**Note**

- When you select the local computer as the file location, the Data Logger saves the data file to your local computer when the logging stops.
- When you select USB storage as the file location, the Data Logger creates a file in the USB when the logging starts and continuously saves data to that file until the logging stops.

## Plugging a USB in the

If you want to save your logged file to a USB drive, plug the USB flash drive in the NI ELVIS III USB host port. The following figure shows the NI ELVIS III with the USB host port highlighted.




**Note** The NI ELVIS III USB host port supports USB flash drives formatted with the FAT32 file system.

## Display Window (Data Logger)

The Data Logger displays the acquired signals on the left side of the panel.

Use **Cursors** to measure a given point on the signal trace, or compare the differences between two points. You can select one of the three modes for a cursor:

| Cursor Mode | Description |
|---|---|
| **Off** | Turns off the cursors. |
| **Track** | Cursor positions are restricted to the acquired data points on a trace. |
| **Manual** | Cursors positions are not restricted to the acquired data points on a trace. |

When **Cursors** is set to **Manual** or **Track**, a table appears above the display with the following data:

| Column | Description |
|---|---|
| **C1** | Displays the time and voltage readings for Cursor 1. |
| **C2** | Displays the time and voltage readings for Cursor 2. |
| **\|Δ Value\|** | Displays the absolute time and voltage differences between Cursor 1 and Cursor 2. |

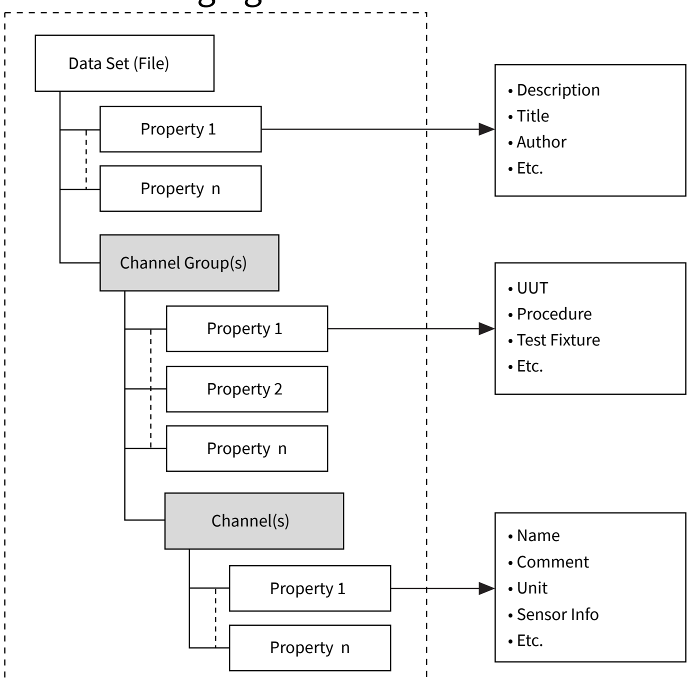
The Data Logger's logging status shows in the upper right corner of the graph. The Data Logger has the following logging statuses:

- **Waiting**—The instrument is waiting for trigger.

- **Logging**—The instrument is logging data.
- **Stopped**—The instrument is stopped. This occurs when you click **Stop**
- **Completed**—The instrument completes logging data. This occurs when a stop trigger or a buffer overflow occurs.

## TDMS File Format (Data Logger)

You can export the data acquired by the Data Logger to TDMS files.
The TDMS file format is supported by the TDM data model. The TDM data model arranges the data in three hierarchical levels: file, channel groups, and channels.
The following figure illustrates the TDM data model:




The TDMS files that you export from the SFPs contain some custom properties, for example, **product-name**. For all TDMS files that you generate using the NI ELVIS III instruments, the value of **product-name** is always NI ELVIS III. These TDMS files also have designated names for the channel group and channels.
The TDMS files you export from the Data Logger have the following data structure:
**channel_group**: Data Logger

- **channel_group**: Data Logger
  - **channel**: Bank A: AI0
  - **channel**: Bank A: AI0 - AI4

- **channel**: Bank B: AI0
- **channel**: Bank B: AI0 - AI4
- ...

The TDMS files list enabled channels only. The order of the channels in the TDMS files matches the top-to-bottom order of the channels on the configuration panel on the right side of the SFP.
The channel name in the TDMS files matches the name you specified for each channel.

# Collaborating with Your Team Members

The NI ELVIS III enables you to work in groups on a shared device.

When multiple users are using different instruments on a shared device, they have control over their respective instruments. When multiple users are using the same instrument on a shared device, only one user can control the instrument. Others can only view the controller's operation and the acquired or generated signals on the Soft Front Panel (SFP). You can easily request or release control over the instrument.

Before collaborating with your team members, read through the following sections to find answers to commonly raised questions.

## Why Can't I Run the Instrument?

One possible reason is that you are in the view mode. In that case, request control to run the instrument. You may also fail to run the instrument because of incorrect settings.

## Why Can't I Change My SFP Settings?

In the view mode, the hardware settings of the device are dimmed. Only the controller can change the hardware settings, for example, running the Oscilloscope or configuring its trigger. Viewers can only change certain display settings, for example, configuring the cursors on the Oscilloscope display. Request control to change the hardware settings.

## How Do I Know If I Am a Controller or a Viewer?

You can find out whether you are a controller or a viewer through the icon on the top right side of the SFP.

- —Only you are using the instrument and you are a controller.
- —Only you are using the instrument and you are a viewer.
- —Multiple users are using the instrument and you are a controller.
- —Multiple users are using the instrument and you are a viewer.

## How Do I Request Control?

Perform the following steps to request control:

1. Click the icon on the top right side of the SFP you are viewing.
2. Click **Request Control**.
3. Wait until the controller approves the request. The controller can decide whether to approve or deny the request. When multiple viewers are requesting control, the controller can choose whom to assign the control to. By default, the first requestor becomes the controller if the current controller does not respond within 30 seconds.

## Why Did I Lose Control over the Instrument?

Whenever you close the SFP, you lose control over that instrument. Another possible reason is poor network connection.

## Where Can I Find All the Users Connecting to the Shared Device?

When you connect to a shared device successfully, you can see the names of all the users connecting to this device by clicking the arrow next to the icon in the **Manage device connection** tab. You can also find all the users who are using the same SFP by clicking that icon on the top right side of the SFP.

## How Do I Give Control to Others?

You can give control to your team member in the following two ways:

- Release control. Your team member requests control and becomes a controller immediately.
- Your team member requests control and you approve the request.