LabVIEW SignalExpress™ Express VIs

June 2008, 371267F-01

Use the LabVIEW SignalExpress Express VIs to build common measurement tasks.

The VIs on this palette can return general LabVIEW error codes.

Subpalette	Description
Acquire Signals Express VIs	Use the Acquire Signals Express VIs to acquire and generate signals from hardware device.
Analysis Express VIs	Use the Analog Express VIs to process, test, import and export, and perform frequency-domain and time-domain measurements on analog signals.
Create Signals Express VIs	Use the Create Signals Express VIs to create a signal.
Generate Signals Express VIs	Use the Generate Signals Express VIs to acquire and generate signals from hardware device.
Load/Save Signals Express VIs	Use the Signal I/O VIs to acquire and generate signals from a hardware device, import or export data from ASCII and LVM files; and import data from SPICE files.
Processing Express VIs	Use the Processing Express VIs to filter, scale, resample, and average signals; apply windowing and perform arithmetic operations; and, interactively align two signals.

Acquire Signals Express VIs

Owning Palette: LabVIEW SignalExpress™ Express VIs

Use the Acquire Signals Express VIs to acquire and generate signals from hardware device.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Palette Object	Description
IVI DMM Acquire	Acquires a signal from an instrument in the Digital Multimeter IVI Class.
IVI Scope Acquire	Acquires an analog waveform from an instrument in the Oscilloscope IVI Class.

IVI DMM Acquire Express VI

Owning Palette: Acquire Signals Express VIs

Installed With: LabVIEW SignalExpress

Acquires a signal from an instrument in the Digital Multimeter IVI Class.

Default values are specific to the hardware and driver specified. The default settings may not be applicable to the measurement you are trying to perform. Click the **Initialize** button, located on the step's toolbar, at any time to reset the step to the default settings.

To communicate with an instrument, you need to install the instrument-specific driver and create a session name for the instrument.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Output Display	Displays the measurement, formatted according to the **Measurement function**, **Range**, and **Resolution**.
Configuration	Contains the following configuration options: • **Device**—Contains the following device options: – **IVI session name**—Specifies the session name to use for this step. This step retrieves possible session names from National Instruments Measurement & Automation Explorer (MAX). You also can create a new session or edit/delete an existing session. – **Resource descriptor**—Specifies the interface and the address of the hardware to associate with the step. – **Instrument driver**—Displays the name of the driver in use. • **Basic Parameters**—Contains the following options: – **Measurement function**—Specifies the type of measurement you want the DMM to perform. Options include DC Volts, AC Volts, DC Current, AC Current, 2 Wire Resistance, 4 Wire Resistance, AC + DC Volts, AC + DC Current, Frequency, and Period. – **Range**—Specifies whether **Auto Range** is used. Contains the following options: • **Auto Range**—Specifies that the DMM automatically calculates the range before each measurement. • **Specify Range**—Allows you to specify the range and uses this

value for all subsequent measurements until you change the measurement configuration.

- **Range value (V)**—[Measurement function: DC Volts, AC Volts, AC + DC Volts] The range in volts for the current measurement.
- **Range value (A)**—[Measurement function: DC Current, AC Current, AC + DC Current] The range in amps for the current measurement.
- **Range value (Ohm)**—[Measurement function: 2 Wire Resistance, 4 Wire Resistance] The range in ohms for the current measurement.
- **Range value (Hz)**—[Measurement function: Frequency] The range in hertz for the current measurement.
- **Range value (s)**—[Measurement function: Period] The range in seconds for the current measurement.
- **Resolution**—Specifies the digital resolution of the measurement. Set **Range** to **Specify Range** to enable this option.
- **Sample period (s)**—Specifies how often to execute the step.

- **Measurement Specific Parameters**—Contains the following options:
  - **Auto zero**—Specifies that the DMM internally disconnects the input signal and takes a zero reading. The DMM then subtracts the zero reading from the measurement to prevent offset voltages present from affecting measurement accuracy. This option does not appear if you set **Measurement function** to

Frequency or **Period**. Contains the following options:

- **On**—Configures the DMM to take a zero reading for each measurement. The DMM subtracts the zero reading from the value it measures.
- **Off**—Disables the **Auto zero** option.
- **Once**—Configures the DMM to take a zero reading immediately. The DMM then subtracts this zero reading from all subsequent values it measures.

- **AC min frequency (Hz)**—[Measurement function: AC Volts, AC Current, AC + DC Volts, AC + DC Current] Specifies the minimum expected frequency component of the input signal in hertz.
- **AC max frequency (Hz)**—[Measurement function: AC Volts, AC Current, AC + DC Volts, AC + DC Current] Specifies the maximum expected frequency component of the input signal in hertz.
- **Frequency voltage range**—[Measurement function: Frequency, Period] Specifies whether the frequency voltage **Auto Range** is used. Contains the following options:
  - **Auto Range**—Configures the DMM to automatically calculate the voltage range before each frequency or period measurement.
  - **Specify Range**—Disables auto ranging. The DMM sets the

	voltage range to the range specified in **Frequency range (V)**. – **Frequency range (V)**—[Measurement function: Frequency, Period] Specifies the expected maximum amplitude of the input signal. The minimum peak-to-peak signal amplitude that the DMM can detect is 10% of the specified voltage range. • **Powerline Frequency (Hz)**—Specifies the powerline frequency in hertz.
Trigger	Contains the following trigger options: • **Type**—Specifies the trigger source you want to use. After the DMM receives the trigger, the DMM waits the length of time you specify in the **Delay (s)** parameter. The DMM then takes a measurement. The default is Immediate. Contains the following options: – **Immediate**—Does not wait for a trigger of any kind. – **External**—Waits for a trigger on the external input. – **Software**—Waits until you press the associated trigger button in the toolbar. – **Digital**—Waits for a trigger on a digital input specified by the trigger source. • **Delay (s)**—Specifies the length of time the DMM waits after it receives the trigger and before it takes a measurement. • **Auto**—Configures the DMM to automatically calculate the trigger delay before each measurement. • **Timeout (s)**—Configures the amount of time to wait while retrieving a reading from the DMM. The default is 5 seconds.

- **Slope**—[Type: External] Specifies whether you want a rising edge or a falling edge passing through the trigger level to trigger the DMM. Contains the following options:
  - **Positive**—Triggers on the rising edge of the external trigger.
  - **Negative**—Triggers on the falling edge of the external trigger.
- **Software trigger source**—[Type: Software] Specifies the trigger source to which you want the instrument to respond. To activate the trigger, click the associated trigger button in the toolbar. Execution waits until you click the associated trigger button in the toolbar. Contains the following options:
  - **Trigger A**—(Default) Specifies Trigger A as the trigger source.
  - **Trigger B**—Specifies Trigger B as the trigger source.
  - **Trigger C**—Specifies Trigger C as the trigger source.
- **Source**—[Type: Digital] Specifies the trigger source you want to use. Contains the following options:
  - **PXI TRIG0 or VXI TTL0**—Waits until it receives a trigger on the PXI TRIG0 line (for PXI instruments) or the VXI TTL0 line (for VXI instruments).
  - **PXI TRIG1 or VXI TTL1**—Waits until it receives a trigger on the PXI TRIG1 line (for PXI instruments) or the VXI TTL1 line (for VXI instruments).
  - **PXI TRIG2 or VXI TTL2**—Waits until it receives a trigger on the PXI TRIG2 line (for PXI instruments) or the VXI TTL2 line (for VXI instruments).
  - **PXI TRIG3 or VXI TTL3**—Waits until it

		receives a trigger on the PXI TRIG3 line (for PXI instruments) or the VXI TTL3 line (for VXI instruments). – **PXI TRIG4 or VXI TTL4**—Waits until it receives a trigger on the PXI TRIG4 line (for PXI instruments) or the VXI TTL4 line (for VXI instruments). – **PXI TRIG5 or VXI TTL5**—Waits until it receives a trigger on the PXI TRIG5 line (for PXI instruments) or the VXI TTL5 line (for VXI instruments). – **PXI TRIG6 or VXI TTL6**—Waits until it receives a trigger on the PXI TRIG6 line (for PXI instruments) or the VXI TTL6 line (for VXI instruments). – **PXI TRIG7 or VXI TTL7**—Waits until it receives a trigger on the PXI TRIG7 line (for PXI instruments) or the VXI TTL7 line (for VXI instruments). – **ECL0**—Waits until it receives a trigger on the VXI ECL0 line. – **ECL1**—Waits until it receives a trigger on the VXI ECL1 line. – **PXI Star**—Waits until it receives a trigger on the PXI STAR trigger bus. – **RTSI 0**—Waits until it receives a trigger on RTSI line 0. – **RTSI 1**—Waits until it receives a trigger on RTSI line 1. – **RTSI 2**—Waits until it receives a trigger on RTSI line 2. – **RTSI 3**—Waits until it receives a trigger on RTSI line 3. – **RTSI 4**—Waits until it receives a trigger on RTSI line 4. – **RTSI 5**—Waits until it receives a trigger

	on RTSI line 5. – **RTSI 6**—Waits until it receives a trigger on RTSI line 6.
Execution Control	Contains the following execution control options: • **Start this step after**—Makes the step wait until another step has started before executing. You can make the step wait on any other hardware step in the project by selecting the step to wait on from the pull-down menu. You can use this option to force an acquisition device to start after a generation device starts. You also can use this option to ensure that a device generating a trigger signal starts after the device receiving the signal, which avoids sending the signal before the receiver is ready. • **Step to wait for**—Lists the possible steps for which this step can wait. • **Pre-execution delay (ms)**—Specifies the amount of time to wait before the step executes. If you configure the step to start after another step, the delay represents the amount of time to wait after the specified step starts. • **Post-execution delay (ms)**—Specifies the amount of time to wait after the step executes.

IVI Scope Acquire Express VI

Owning Palette: Acquire Signals Express VIs

Installed With: LabVIEW SignalExpress

Acquires an analog waveform from an instrument in the Oscilloscope IVI Class.

Default values are specific to the hardware and driver specified. The default settings might not be applicable to the measurement you are trying to perform.

The IVI Scope Acquire toolbar includes two buttons you can use to set parameter values. Click the **Initialize** button to set IVI Scope Acquire to the default settings. Click the **Autosetup** button to set parameters to values that IVI Scope Acquire determines best fit the signal you are acquiring.

Note Clicking the **Autosetup** button executes IVI Scope Acquire.

To communicate with an instrument, you need to install the instrument-specific driver and create a session name for the instrument.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Autoscale amplitude	Scales the amplitude axis of the **Acquired signals** graph. The default is to autoscale the amplitude.
Acquired Signals	Displays the waveform from the device. **Range (V)** and **Offset (V)** set the hardware limits.
Configuration	Contains the following configuration options: • **Device**—Contains the following device options: – **IVI session name**—Specifies the session name to use for this step. This step retrieves possible session names from National Instruments Measurement & Automation Explorer (MAX). You also can create a new session or edit/delete an existing session. – **Resource descriptor**—Specifies the interface and the address of the hardware to associate with the step. – **Instrument driver**—Displays the name of the driver in use. • **Vertical**—Contains channel configuration options that affect the data along the Voltage (V) axis. The settings you configure with these options are specific to the channel you select in the **Channels** field. **Vertical** contains the following options: – **Channels**—Specifies the physical channels from which to generate data. – **Enable channel**—Specifies whether to enable data acquisition on the selected channel. – **Range (V)**—Specifies the value of the input range the oscilloscope uses for the channel. For example, to acquire a sine wave that spans -5 to 5 volts, enter 10 as

		the value of this parameter. – **Input impedance (Ohms)**—Specifies the input impedance you want to use for the channel. – **Probe attenuation**—Specifies the scaling factor by which the probe you attach to the channel attenuates the input. Pass -1 to auto detect. – **Offset (V)**—Specifies the location of the center of the range that you specify with **Range (V)**. Enter the value with respect to ground. For example, to acquire a sine wave that spans 0 to 10 volts, enter 5 as the value of this parameter. – **Coupling**—Specifies how you want the oscilloscope to couple the input signal for the channel. Options include AC, DC, and GND. – **Bandwidth (Hz)**—Specifies the maximum frequency for the input signal you want the instrument to accommodate without attenuating the signal by more than 3 dB. • **Horizontal**—Contains the following device-specific options for configuring the Time (s) axis: – **Start time (s)**—Specifies the length of time from the trigger event to the first point in the waveform record. If this value is positive, the first point in the waveform record occurs after the trigger event. If this value is negative, the first point in the waveform record occurs before the trigger event. – **Time per record (s)**—Specifies the time in seconds that corresponds to the record length. – **Min record length (S)**—Specifies the

	minimum number of points you require in the waveform record for each channel.
Trigger	Contains the following trigger options: • **Type**—Specifies the type of trigger you want the oscilloscope to use. Contains the following options: – **Immediate**— Configures the oscilloscope for immediate triggering. The oscilloscope does not wait for a trigger of any kind upon initialization. – **Edge**—Configures the oscilloscope for edge triggering. An edge trigger occurs when the trigger signal crosses the trigger level you specify with the slope you specify. – **TV**—Configures the oscilloscope for TV triggering. – **Runt**—Configures the oscilloscope for runt triggering. A runt trigger occurs when the trigger signal crosses one of the runt thresholds twice without crossing the other runt threshold. – **Glitch**—Configures the oscilloscope for glitch triggering. A glitch trigger occurs when the trigger signal has a pulse with a width that is less than the glitch width. The trigger does not actually occur until the edge of the pulse that corresponds to the glitch width and polarity you specify crosses the trigger level. – **Width**—Configures the oscilloscope for width triggering. A width trigger occurs when the oscilloscope detects a positive or negative pulse with a width between, or optionally outside, the width thresholds. The trigger does not actually occur until the edge of a pulse that

corresponds to the width thresholds and polarity you specify crosses the trigger level.

- **AC Line**—Configures the oscilloscope for AC line triggering.

- **Source**—Specifies the source for the oscilloscope to monitor for a trigger.
- **Holdoff (s)**—Specifies the length of time you want the oscilloscope to wait after it detects a trigger until the oscilloscope enables the trigger subsystem to detect another trigger.
- **Timeout (s)**—Specifies the maximum amount of time to wait for the oscilloscope to acquire data. When a timeout occurs during an acquisition, it is normally due to a failure to trigger. The default is 10.
- **Level (V)**—[Type: Edge] Specifies the voltage you want the oscilloscope to use for edge triggering. The oscilloscope triggers when the trigger signal passes through the threshold you specify with the **Slope** parameter.

  Note This parameter affects instrument behavior only when you select a channel or the external trigger input as the trigger source. You may not configure the trigger level that the oscilloscope uses for other trigger sources, such as VXI TTL trigger lines.

- **Slope**—[Type: Edge] Specifies whether you want a rising edge or a falling edge passing through the trigger level to trigger the oscilloscope. Options include Positive and Negative.
- **Coupling**—[Type: Edge] Specifies the trigger coupling. Options include AC, DC, HF Reject, LF Reject, and Noise Reject.

- **Polarity**—[Type: TV] Specifies the polarity of the TV signal. Options include Positive and Negative.
- **Signal format**—[Type: TV] Specifies the type of TV signal on which the oscilloscope triggers. Options include NTSC, PAL, and SECAM.
- **Event**—[Type: TV] Specifies the TV event on which you want the oscilloscope to trigger. Options include Field 1, Field 2, Any Field, Any Line, and Line Number.
- **Line number**—[Type: TV] Specifies the line in the field on which you want the oscilloscope to trigger. The specified line number is independent of any field. This means that to trigger on the first line of **Field 2**, you must specify a line number of 263 (if we assume that **Field 1** has 262 lines).
- **Polarity**—[Type: Runt] Specifies the polarity of the runt that you want to trigger the oscilloscope. Contains the following options:
    - **Positive**—Triggers on a positive runt. A positive runt occurs when a rising edge crosses the **Low threshold (V)** and does not cross the **High threshold (V)** before recrossing the **Low threshold (V)**.
    - **Negative**—Triggers on a negative runt. A negative runt occurs when a falling edge crosses the **High threshold (V)** and does not cross the **Low threshold (V)** before recrossing the **High threshold (V)**.
    - **Either**—Triggers on either a positive or negative runt.
- **Low threshold (V)**—[Type: Runt] Specifies the low threshold you want the oscilloscope to use for runt triggering.
- **High threshold (V)**—[Type: Runt] Specifies the high threshold you want the oscilloscope to use

for runt triggering.

- **Level (V)**—[Type: Glitch] Specifies the voltage threshold you want the oscilloscope to use for glitch triggering. The oscilloscope triggers when a glitch crosses the trigger threshold you specify with this parameter.
- **Polarity**—[Type: Glitch] Specifies the polarity of the glitch that you want to trigger the oscilloscope. Options include Positive, Negative, and Either.
- **Condition**—[Type: Glitch] Specifies the glitch condition. The oscilloscope triggers when it detects a pulse with a width less than or greater than the **Width (s)** value. Options include Less Than and Greater Than.
- **Width (s)**—[Type: Glitch] Specifies the length of time you want the oscilloscope to use for the glitch width. The oscilloscope triggers when it detects a pulse with a width less than or greater than this value, depending on the **Condition** parameter.
- **Level (V)**—[Type: Width] Specifies the voltage threshold you want the oscilloscope to use for width triggering. The oscilloscope triggers when the edge of a pulse that corresponds to the **Low threshold (V)**, **High threshold (V)**, **Condition**, and **Polarity** crosses the threshold you specify in this parameter.
- **Polarity**—[Type: Width] Specifies the polarity of the pulse that you want to trigger the oscilloscope. Options include Positive and Negative.
- **High threshold (V)**—[Type: Width] Specifies the high width threshold.
- **Low threshold (V)**—[Type: Width] Specifies the low width threshold.
- **Condition**—[Type: Width] Specifies whether you

	want a pulse that is within or outside the **High threshold (V)** and **Low threshold (V)** to trigger the oscilloscope. Contains the following options: – **Within**—Triggers on pulses that have a width that is less than the **High threshold (V)** and greater than the **Low Threshold (V)**. – **Outside**—Triggers on pulses that have a width that is either greater than the **High threshold (V)** or less than the **Low threshold (V)**. • **Slope**—[Type: AC Line] Specifies whether you want the oscilloscope to trigger on a zero crossing with a positive, negative, or either slope of the network supply voltage. Options include Positive, Negative, and Either.
Advanced	Contains the following option: • **Acquisition Settings**—Contains the following option: – **Acquisition type**—Specifies the manner in which you want the oscilloscope to acquire data and fill the waveform record. Contains the following options: • **Normal**—Sets the oscilloscope to normal acquisition mode. The oscilloscope acquires one sample for each point in the waveform record. The oscilloscope can use real-time or equivalent-time sampling. • **Peak Detect**—Sets the oscilloscope to the peak-detect acquisition mode. The oscilloscope oversamples the input signal and keeps the minimum and maximum values that correspond to each position

	in the waveform record. The oscilloscope uses only real-time sampling. • **High Resolution**—Sets the oscilloscope to the high-resolution acquisition mode. The oscilloscope oversamples the input signal and calculates an average value for each position in the waveform record. The oscilloscope uses only real-time sampling. • **Envelope**—Sets the oscilloscope to the envelope acquisition mode. The oscilloscope acquires multiple waveforms and keeps the minimum and maximum voltages it acquires for each point in the waveform record. The oscilloscope can use real-time or equivalent-time sampling. • **Average**—Sets the oscilloscope to the average acquisition mode. The oscilloscope acquires multiple waveforms and calculates an average value for each point in the waveform record. The oscilloscope can use real-time or equivalent-time sampling.  **Note** When you set this parameter to **Envelope** or **Peak Detect**, the oscilloscope acquires minimum and maximum waveforms.
Execution	Contains the following execution control options:

Control	• **Start this step after**—Makes the step wait until another step has started before executing. You can make the step wait on any other hardware step in the project by selecting the step to wait on from the pull-down menu. You can use this option to force an acquisition device to start after a generation device starts. You also can use this option to ensure that a device generating a trigger signal starts after the device receiving the signal, which avoids sending the signal before the receiver is ready. • **Step to wait for**—Lists the possible steps for which this step can wait. • **Pre-execution delay (ms)**—Specifies the amount of time to wait before the step executes. If you configure the step to start after another step, the delay represents the amount of time to wait after the specified step starts. • **Post-execution delay (ms)**—Specifies the amount of time to wait after the step executes.

Generate Signals Express VIs

Owning Palette: LabVIEW SignalExpress™ Express VIs

Use the Generate Signals Express VIs to acquire and generate signals from hardware device.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Palette Object	Description
IVI FGEN Arbitrary Waveform	Generates an analog arbitrary signal using an instrument in the Arbitrary Waveform/Function Generator IVI class.
IVI FGEN Standard Function	Generates an analog standard function using an instrument in the Arbitrary Waveform/Function Generator IVI class.
IVI Power Supply	Generates a voltage level using an instrument in the DC Power Supply IVI class.

IVI FGEN Arbitrary Waveform Express VI

Owning Palette: Generate Signals Express VIs

Installed With: LabVIEW SignalExpress

Generates an analog arbitrary signal using an instrument in the Arbitrary Waveform/Function Generator IVI class.

Default values are specific to the hardware and driver specified. The default settings may not be applicable to the measurement you are trying to perform. Click the **Initialize** button, located on the step's toolbar, at any time to reset the step to the default settings.

To communicate with an instrument, you need to install the instrument-specific driver and create a session name for the instrument.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Waveform Preview	Displays a preview of the waveform downloaded to the function generator. The vertical and horizontal graph axes are formatted according to the **Sample rate (S/s)**, **Gain**, and **Offset (V)** settings applied to the function generator.
Configuration	Contains the following arbitrary waveform configuration options: • **Device**—Contains the following device options: – **IVI session name**—Specifies the session name to use for this step. This step retrieves possible session names from National Instruments Measurement & Automation Explorer (MAX). You also can create a new session or edit/delete an existing session. – **Resource descriptor**—Specifies the interface and the address of the hardware to associate with the step. – **Instrument driver**—Displays the name of the driver in use. • **Channel Configuration**—Contains the following channel configuration options: – **Channels**—Specifies the physical channels on which data is generated. – **Enable channel**—Specifies whether to enable data acquisition on the selected channel. – **Input signal**—Select the appropriate waveform to generate. – **Generation Mode**—Contains the following generation mode options: • **Generate continuously**—Generates the input signal

continuously. If you run the project continuously, the step generates the input signal repeatedly without discontinuities. If you run the project in Run Once mode, the step generates the input signal once.

- **Generate N waveforms**—Generates the input signal N times in a non-continuous fashion. If you run the project continuously, the step generates the input signal repeatedly but discontinuously. If you run the project in Run Once mode, the step generates the input signal once. You can use this option if you want the device to generate a start trigger every time the device starts generating the signal.
- **Number of waveforms**—Number of times to generate the **Input Signal**.

– **Output Signal**—Contains the following output signal options:

- **Extract from waveform**—Specifies whether settings for **Sample rate (S/s)**, **Gain**, and **Offset (V)** are extracted from the waveform or specified manually. When you remove the checkmark from the **Extract from waveform** checkbox, you first must normalize the data points to a range of -1 to +1.
- **Sample rate (S/s)**—Specifies

		the sample rate at which you want the function generator to output arbitrary waveforms. • **Gain**—Specifies the factor by which the function generator scales the arbitrary waveform data. When you create arbitrary waveforms, you first must normalize the data points to a range of -1 to +1. You use this property to scale the arbitrary waveform to other ranges. For example, to configure the output signal to range from -2 to +2 volts, set **Gain** to 2. • **Offset (V)**—Specifies the value the function generator adds to the arbitrary waveform data. When you create arbitrary waveforms, you first must normalize the data points to a range of -1 to +1. You use this parameter to shift the range of the arbitrary waveform. For example, to configure the output signal to range from 0 to 2 volts instead of -1 to 1 volts, set **Offset (V)** to 1. • **Impedance (Ohms)**—Specifies the impedance value you want the function generator to use. A value of 0 indicates that the function generator is connected to a high impedance load. • **Frequency (Hz)**—Specifies the frequency at which you want the function generator to produce one cycle of an arbitrary

	waveform.
Trigger	Contains the following trigger options: • **Channel Triggering**—Contains the following channel trigger options: – **Channels**—Specifies the physical channels that have a trigger associated with them. Each channel can be triggered independently. – **Type**—Specifies the trigger source to which you want the function generator to respond. The default is Immediate. Contains the following options: • **Immediate**—Does not wait for a trigger of any kind. • **Internal**—Waits for a trigger on the internal trigger input. • **External**—Waits for a trigger on the external trigger input. • **Software Trigger Function**—Waits until the software trigger button specified by the **Software trigger source** is pressed on the toolbar. • **Digital**—Waits for a trigger on a digital input specified by the trigger source. – **Internal trigger rate (trig/s)**—Specifies the rate at which you want the internal trigger rate of the function generator to generate trigger signals. **Internal trigger rate (trig/s)** is available only when you select **Internal** in **Type**. – **Software trigger source**—[Type: Software] Specifies the trigger source to which you want the instrument to respond. To activate the trigger, click the

associated trigger button in the toolbar. Execution waits until you click the associated trigger button in the toolbar. Contains the following options:

- **Trigger A**—(Default) Specifies Trigger A as the trigger source.
- **Trigger B**—Specifies Trigger B as the trigger source.
- **Trigger C**—Specifies Trigger C as the trigger source.

- **Source**—[Type: Digital] Specifies the trigger source you want to use. Contains the following options:
  - **PXI TRIG0 or VXI TTL0**—Waits until it receives a trigger on the PXI TRIG0 line (for PXI instruments) or the VXI TTL0 line (for VXI instruments).
  - **PXI TRIG1 or VXI TTL1**—Waits until it receives a trigger on the PXI TRIG1 line (for PXI instruments) or the VXI TTL1 line (for VXI instruments).
  - **PXI TRIG2 or VXI TTL2**—Waits until it receives a trigger on the PXI TRIG2 line (for PXI instruments) or the VXI TTL2 line (for VXI instruments).
  - **PXI TRIG3 or VXI TTL3**—Waits until it receives a trigger on the PXI TRIG3 line (for PXI instruments) or the VXI TTL3 line (for VXI instruments).
  - **PXI TRIG4 or VXI TTL4**—Waits until it receives a trigger on the PXI TRIG4 line (for PXI instruments) or the VXI TTL4 line

		(for VXI instruments). • **PXI TRIG5 or VXI TTL5**—Waits until it receives a trigger on the PXI TRIG5 line (for PXI instruments) or the VXI TTL5 line (for VXI instruments). • **PXI TRIG6 or VXI TTL6**—Waits until it receives a trigger on the PXI TRIG6 line (for PXI instruments) or the VXI TTL6 line (for VXI instruments). • **PXI TRIG7 or VXI TTL7**—Waits until it receives a trigger on the PXI TRIG7 line (for PXI instruments) or the VXI TTL7 line (for VXI instruments). • **ECL0**—Waits until it receives a trigger on the VXI ECL0 line. • **ECL1**—Waits until it receives a trigger on the VXI ECL1 line. • **PXI Star**—Waits until it receives a trigger on the PXI STAR trigger bus. • **RTSI 0**—Waits until it receives a trigger on RTSI line 0. • **RTSI 1**—Waits until it receives a trigger on RTSI line 1. • **RTSI 2**—Waits until it receives a trigger on RTSI line 2. • **RTSI 3**—Waits until it receives a trigger on RTSI line 3. • **RTSI 4**—Waits until it receives a trigger on RTSI line 4. • **RTSI 5**—Waits until it receives a trigger on RTSI line 5. • **RTSI 6**—Waits until it receives a

	trigger on RTSI line 6. • **Reference clock source**—Specifies the reference clock source you want the function generator to use. The function generator derives the frequencies and sample rates that it uses to generate waveforms from the source you specify. Options include Internal, External, and RTSI Clock. For example, when you set **Reference clock source** to **External**, the function generator uses the signal it receives at its external clock terminal as its reference clock.
Execution Control	Contains the following execution control options: • **Start this step after**—Makes the step wait until another step has started before executing. You can make the step wait on any other hardware step in the project by selecting the step to wait on from the pull-down menu. You can use this option to force an acquisition device to start after a generation device starts. You also can use this option to ensure that a device generating a trigger signal starts after the device receiving the signal, which avoids sending the signal before the receiver is ready. • **Step to wait for**—Lists the possible steps for which this step can wait. • **Pre-execution delay (ms)**—Specifies the amount of time to wait before the step executes. If you configure the step to start after another step, the delay represents the amount of time to wait after the specified step starts. • **Post-execution delay (ms)**—Specifies the amount of time to wait after the step executes.

IVI FGEN Standard Function Express VI

Owning Palette: Generate Signals Express VIs

Installed With: LabVIEW SignalExpress

Generates an analog standard function using an instrument in the Arbitrary Waveform/Function Generator IVI class.

Default values are specific to the hardware and driver specified. The default settings may not be applicable to the measurement you are trying to perform. Click the **Initialize** button, located on the step's toolbar, at any time to reset the step to the default settings.

To communicate with an instrument, you need to install the instrument-specific driver and create a session name for the instrument.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Function Preview	Displays a preview of the function. The vertical and horizontal graph axes are formatted according to the **Channel Configuration** settings that are applied to the signal generator.
Configuration	Contains the following configuration options: • **Device**—Contains the following device options: – **IVI session name**—Specifies the session name to use for this step. This step retrieves possible session names from National Instruments Measurement & Automation Explorer (MAX). You also can create a new session or edit/delete an existing session. – **Resource descriptor**—Specifies the interface and the address of the hardware to associate with the step. – **Instrument driver**—Displays the name of the driver in use. • **Channel Configuration**—Contains the following channel configuration options: – **Channels**—Specifies the physical channels on which data is generated. – **Enable channel**—Specifies whether to enable data acquisition on the selected channel. – **Type**—Specifies the standard waveform that you want the function generator to produce. Options include Sine, Square, Triangle, Ramp Up, Ramp Down, and DC. – **Amplitude (Vpp)**—Specifies the amplitude of the standard waveform that you want the function generator to

produce. This value is the amplitude at the output terminal. For example, to produce a waveform ranging from -5 to +5 volts, set the **Amplitude (Vpp)** to 10 volts.

- **Start phase (deg)**—Specifies the horizontal offset of the standard waveform you want the function generator to produce. You specify this property in degrees of one waveform cycle. A start phase of 180 degrees means output generation begins halfway through the waveform. A start phase of 360 degrees offsets the output by an entire waveform cycle, which is identical to a start phase of 0 degrees.
- **Frequency (Hz)**—Specifies the frequency of the standard waveform that you want the function generator to produce.
- **DC offset (V)**—Specifies the DC offset of the standard waveform that you want the function generator to produce. The value is the offset from ground to the center of the waveform you specify with the **Type** parameter. For example, to configure a waveform with an amplitude of 10 volts to range from 0 to +10 volts, set **DC offset (V)** to 5 volts.
- **Output impedance (Ohms)**—Specifies the impedance value you want the function generator to use. A value of 0 indicates that the function generator is connected to a high impedance load.
- **Duty cycle (%)**—Specifies the percentage of time a square wave remains high versus one entire period. The default is 50%. **Duty cycle (%)** is

	available only when you select **Square** in **Type**. • **Generation Mode**—Contains the following generation mode options: – **Generate continuously**—Generates the input signal continuously. If you run the project continuously, the step generates the input signal repeatedly without discontinuities. If you run the project in Run Once mode, the step generates the input signal once. – **Generate N waveforms**—Generates the input signal N times in a non-continuous fashion. If you run the project continuously, the step generates the input signal repeatedly but discontinuously. If you run the project in Run Once mode, the step generates the input signal once. You can use this option if you want the device to generate a start trigger every time the device starts generating the signal. – **Number of waveforms**—Specifies the number of times to generate the waveform.
Trigger	Contains the following trigger options: • **Channel Triggering**—Contains the following channel trigger options: – **Channels**—Specifies the physical channels that have a trigger associated with them. Each channel can be triggered independently. – **Type**—Specifies the trigger source to which you want the function generator to respond. The default is Immediate. Contains the following options: • **Immediate**—Does not wait for a

trigger of any kind.

- **Internal**—Waits for a trigger on the internal trigger input.
- **External**—Waits for a trigger on the external trigger input.
- **Software**—Waits until the software trigger button specified by the **Software trigger source** is pressed on the toolbar.
- **Digital**—Waits for a trigger on a digital input specified by the trigger source.

- **Internal trigger rate (trig/s)**—Specifies the rate at which you want the internal trigger rate of the function generator to generate trigger signals. **Internal trigger rate (trig/s)** is available only when you select **Internal** in **Type**.
- **Software trigger source**—[Type: Software] Specifies the trigger source to which you want the instrument to respond. To activate the trigger, click the associated trigger button in the toolbar. Execution waits until you click the associated trigger button in the toolbar. Contains the following options:
  - **Trigger A**—(Default) Specifies Trigger A as the trigger source.
  - **Trigger B**—Specifies Trigger B as the trigger source.
  - **Trigger C**—Specifies Trigger C as the trigger source.
- **Source**—[Type: Digital] Specifies the trigger source you want to use. Contains the following options:
  - **PXI TRIG0 or VXI TTL0**—Waits until it receives a trigger on the

		PXI TRIG0 line (for PXI instruments) or the VXI TTL0 line (for VXI instruments). • **PXI TRIG1 or VXI TTL1**—Waits until it receives a trigger on the PXI TRIG1 line (for PXI instruments) or the VXI TTL1 line (for VXI instruments). • **PXI TRIG2 or VXI TTL2**—Waits until it receives a trigger on the PXI TRIG2 line (for PXI instruments) or the VXI TTL2 line (for VXI instruments). • **PXI TRIG3 or VXI TTL3**—Waits until it receives a trigger on the PXI TRIG3 line (for PXI instruments) or the VXI TTL3 line (for VXI instruments). • **PXI TRIG4 or VXI TTL4**—Waits until it receives a trigger on the PXI TRIG4 line (for PXI instruments) or the VXI TTL4 line (for VXI instruments). • **PXI TRIG5 or VXI TTL5**—Waits until it receives a trigger on the PXI TRIG5 line (for PXI instruments) or the VXI TTL5 line (for VXI instruments). • **PXI TRIG6 or VXI TTL6**—Waits until it receives a trigger on the PXI TRIG6 line (for PXI instruments) or the VXI TTL6 line (for VXI instruments). • **PXI TRIG7 or VXI TTL7**—Waits until it receives a trigger on the PXI TRIG7 line (for PXI instruments) or the VXI TTL7 line

	(for VXI instruments). • **ECL0**—Waits until it receives a trigger on the VXI ECL0 line. • **ECL1**—Waits until it receives a trigger on the VXI ECL1 line. • **PXI Star**—Waits until it receives a trigger on the PXI STAR trigger bus. • **RTSI 0**—Waits until it receives a trigger on RTSI line 0. • **RTSI 1**—Waits until it receives a trigger on RTSI line 1. • **RTSI 2**—Waits until it receives a trigger on RTSI line 2. • **RTSI 3**—Waits until it receives a trigger on RTSI line 3. • **RTSI 4**—Waits until it receives a trigger on RTSI line 4. • **RTSI 5**—Waits until it receives a trigger on RTSI line 5. • **RTSI 6**—Waits until it receives a trigger on RTSI line 6. • **Reference clock source**—Specifies the reference clock source you want the function generator to use. The function generator derives the frequencies and sample rates that it uses to generate waveforms from the source you specify. Options include Internal, External, and RTSI Clock. For example, when you set **Reference clock source** to **External**, the function generator uses the signal it receives at its external clock terminal as its reference clock.
Execution Control	Contains the following execution control options: • **Start this step after**—Makes the step wait until another step has started before executing. You can make the step wait on any other hardware

	step in the project by selecting the step to wait on from the pull-down menu. You can use this option to force an acquisition device to start after a generation device starts. You also can use this option to ensure that a device generating a trigger signal starts after the device receiving the signal, which avoids sending the signal before the receiver is ready. • **Step to wait for**—Lists the possible steps for which this step can wait. • **Pre-execution delay (ms)**—Specifies the amount of time to wait before the step executes. If you configure the step to start after another step, the delay represents the amount of time to wait after the specified step starts. • **Post-execution delay (ms)**—Specifies the amount of time to wait after the step executes.

IVI Power Supply Express VI

Owning Palette: Generate Signals Express VIs

Installed With: LabVIEW SignalExpress

Generates a voltage level using an instrument in the DC Power Supply IVI class.

Default values are specific to the hardware and driver specified. The default settings may not be applicable to the measurement you are trying to perform. Click the **Initialize** button, located on the step's toolbar, at any time to reset the step to the default settings.

To communicate with an instrument, you need to install the instrument-specific driver and create a session name for the instrument.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Query device for measurement	Queries the power supply for the actual current and voltage the device is generating.  **Note** When you place a checkmark in the **Query device for measurement** checkbox, the performance of the device decreases.
Voltage Display	Displays the voltage (V) of the power supply.
Current Display	Displays the current (A) of the power supply.
Output	Contains the following output options: • **Export over-voltage tripped**—Exports whether the over-voltage was tripped as a Boolean value in the Project View. • **Export over-current tripped**—Exports whether the over-current was tripped as a Boolean value in the Project View.
Configuration	Contains the following power supply configuration options: • **Device**—Contains the following device options: – **IVI session name**—Specifies the session name to use for this step. This step retrieves possible session names from National Instruments Measurement & Automation Explorer (MAX). You also can create a new session or edit/delete an existing session. – **Resource descriptor**—Specifies the interface and the address of the hardware to associate with the step. – **Instrument driver**—Displays the name of the driver in use. • **Channel Configuration**—Contains the following

	channel configuration options: – **Channels**—Specifies the physical channels from which to generate data. – **Enable channel**—Specifies whether to enable data acquisition on the selected channel. – **Voltage level (V)**—Specifies the DC voltage you want the power supply to attempt to generate. – **OVP enabled**—Specifies whether you want to use an over-voltage protection limit. Place a checkmark in this checkbox to enable the **OVP limit (V)** field. – **OVP limit (V)**—Specifies the over-voltage protection limit you want to use. – **Specify output range**—Select this option to define an output range for the output current or voltage. – **Range type**—Specifies the type of range to configure. – **Range (A)**—Specifies the range in amperes. – **Range (V)**—Specifies the value of the input range the oscilloscope uses for the channel. For example, to acquire a sine wave that spans -5 to 5 volts, enter 10 as the value of this parameter. – **Current limit behavior**—Specifies the behavior you want the power supply to exhibit when the output current is greater than or equal to the value of **Current limit (A)**. Options include Regulate and Trip. – **Current limit (A)**—Specifies the current limit you want to use.
Trigger	Contains the following trigger options:

- **Trigger generation**—Specifies whether any channels wait for triggers. **Trigger generation** is disabled by default. If you do not enable **Trigger generation**, the power supply generates the current and **Voltage level (V)** when you click the **Run** button. Place a checkmark in this checkbox to enable the **Channel Triggering** options and configure triggers.
- **Channel Triggering**—Contains the following channel trigger options:
  - **Channels**—Specifies the physical channels that have a trigger associated with them. Each channel can be triggered independently.
  - **Type**—Specifies the trigger source to which you want the power supply to respond. The default is Immediate. Contains the following options:
    - **Immediate**—Does not wait for a trigger of any kind.
    - **External**—Waits for a trigger on the external trigger input.
    - **Software**—Waits until the software trigger button is pressed from the toolbar, specified by the software trigger source.
    - **Digital**—Waits for a trigger on a digital input specified by the trigger source.
  - **Triggered level (V)**—Specifies the DC voltage level you want the power supply to attempt to generate after it receives a trigger.
  - **Triggered current limit (A)**—Specifies the current limit you want the power supply to use after it receives a trigger.
  - **Software trigger source**—Specifies the

software trigger source to which you want the instrument to respond. Options include Trigger A, Trigger B, and Trigger C. The default is Trigger A. **Software trigger source** is available only when you select **Software** in **Type**.

- **Source**—Specifies the trigger source you want to use. **Source** is available only when you select **Digital** in **Type**. Contains the following options:
    - **PXI TRIG0 or VXI TTL0**—Waits until it receives a trigger on the PXI TRIG0 line (for PXI instruments) or the VXI TTL0 line (for VXI instruments).
    - **PXI TRIG1 or VXI TTL1**—Waits until it receives a trigger on the PXI TRIG1 line (for PXI instruments) or the VXI TTL1 line (for VXI instruments).
    - **PXI TRIG2 or VXI TTL2**—Waits until it receives a trigger on the PXI TRIG2 line (for PXI instruments) or the VXI TTL2 line (for VXI instruments).
    - **PXI TRIG3 or VXI TTL3**—Waits until it receives a trigger on the PXI TRIG3 line (for PXI instruments) or the VXI TTL3 line (for VXI instruments).
    - **PXI TRIG4 or VXI TTL4**—Waits until it receives a trigger on the PXI TRIG4 line (for PXI instruments) or the VXI TTL4 line (for VXI instruments).
    - **PXI TRIG5 or VXI TTL5**—Waits until it receives a trigger on the

	PXI TRIG5 line (for PXI instruments) or the VXI TTL5 line (for VXI instruments). • **PXI TRIG6 or VXI TTL6**—Waits until it receives a trigger on the PXI TRIG6 line (for PXI instruments) or the VXI TTL6 line (for VXI instruments). • **PXI TRIG7 or VXI TTL7**—Waits until it receives a trigger on the PXI TRIG7 line (for PXI instruments) or the VXI TTL7 line (for VXI instruments). • **ECL0**—Waits until it receives a trigger on the VXI ECL0 line. • **ECL1**—Waits until it receives a trigger on the VXI ECL1 line. • **PXI Star**—Waits until it receives a trigger on the PXI STAR trigger bus. • **RTSI 0**—Waits until it receives a trigger on RTSI line 0. • **RTSI 1**—Waits until it receives a trigger on RTSI line 1. • **RTSI 2**—Waits until it receives a trigger on RTSI line 2. • **RTSI 3**—Waits until it receives a trigger on RTSI line 3. • **RTSI 4**—Waits until it receives a trigger on RTSI line 4. • **RTSI 5**—Waits until it receives a trigger on RTSI line 5. • **RTSI 6**—Waits until it receives a trigger on RTSI line 6.
Execution Control	Contains the following execution control options: • **Start this step after**—Makes the step wait until

	another step has started before executing. You can make the step wait on any other hardware step in the project by selecting the step to wait on from the pull-down menu. You can use this option to force an acquisition device to start after a generation device starts. You also can use this option to ensure that a device generating a trigger signal starts after the device receiving the signal, which avoids sending the signal before the receiver is ready. • **Step to wait for**—Lists the possible steps for which this step can wait. • **Pre-execution delay (ms)**—Specifies the amount of time to wait before the step executes. If you configure the step to start after another step, the delay represents the amount of time to wait after the specified step starts. • **Post-execution delay (ms)**—Specifies the amount of time to wait after the step executes.

Create Signals Express VIs

Owning Palette: LabVIEW SignalExpress™ Express VIs

Use the Create Signals Express VIs to create a signal.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Palette Object	Description
Create Analog Signal	Creates an analog signal. You can create various periodic waveform signals as well as noise, multi-tone, or DC signals. You also can use a formula to define a signal. Use Create Analog Signal to create arbitrary signals. For example, you can use Create Analog Signal to create a signal to use as a stimulus for a hardware device. Create Analog Signal can run in continuous signal mode or repeated signal mode, depending on whether you place a checkmark in the **Repeated signal** checkbox. The default is continuous signal mode.
Create Digital Signal	Creates different types of digital signals. Depending on the option you select in the **Signal type** pull-down menu, this step can create a ramp, marching values, single value, random, or toggle pattern.

Create Analog Signal Express VI

Owning Palette: Create Signals Express VIs

Installed With: LabVIEW SignalExpress

Creates an analog signal. You can create various periodic waveform signals as well as noise, multi-tone, or DC signals. You also can use a formula to define a signal. Use Create Analog Signal to create arbitrary signals. For example, you can use Create Analog Signal to create a signal to use as a stimulus for a hardware device. Create Analog Signal can run in continuous signal mode or repeated signal mode, depending on whether you place a checkmark in the **Repeated signal** checkbox. The default is continuous signal mode.

Details

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Output Signal	Displays the signal the step creates.
Configuration	Contains the following options: • **Signal Calculation Setup**—Contains options you can use to configure how the step calculates the signal. The options that appear in this section depend on the **Signal type** you specify. Contains the following options: – **Signal type**—Specifies the type of signal Create Analog Signal creates. You can select from the following options: • **Sine Wave**—(Default) Creates a sine wave with a default amplitude and frequency of 1. • **Triangle Wave**—Creates a triangle wave with a default amplitude and frequency of 1. • **Square Wave**—Creates a square wave with a default amplitude and frequency of 1. • **Sawtooth Wave**—Creates a sawtooth wave with a default amplitude and frequency of 1. • **DC Signal**—Creates a DC signal with a default offset of 0V. • **Noise Signal**—Creates a noise signal with a default level of 1. Use the **Noise type** field to specify Gaussian, rectangular, or triangular amplitude distribution. • **Multi-tone**—Creates a multi-tone signal with a default start frequency and amplitude of 1 and a default stop frequency of

2. Multi-tone signals allow the fast and efficient stimulus of a system across an arbitrary band of frequencies, and you can use them to determine the frequency response of a device.

- **Formula**—Creates a signal according to the formula you enter in the **Formula** field.

- **Frequency (Hz)**—[Signal type: Sine Wave, Triangle Wave, Square Wave, Sawtooth Wave, Formula] Specifies the frequency of a sine, triangle, square, or sawtooth wave in hertz or the value of f if you select the **Formula** signal type. The default is 1.0 kHz.
- **Amplitude (V)**—[Signal type: Sine Wave, Triangle Wave, Square Wave, Sawtooth Wave, Formula] Specifies the amplitude of a sine, triangle, square, or sawtooth wave or the value of a if you select the **Formula** signal type. The default is 1.0 V.
- **Phase (deg.)**—[Signal type: Sine Wave, Triangle Wave, Square Wave, Sawtooth Wave] Specifies the initial phase of a sine, triangle, square, or sawtooth wave in degrees. The default is 0 degrees.
- **Offset (V)**—Specifies the DC offset of the signal. The default is 0 V.
- **Repeated signal**—Specifies if the created signal is repeated or continuous. If you place a checkmark in this checkbox, Create Analog Signal calculates the signal only during the first iteration of the step after you click the **Run** button or the **Reset Signal** button and each time you change a

configuration parameter. The signal then repeats with the same time stamp and start phase.

- **N periods**—[Signal type: Sine Wave, Triangle Wave, Square Wave, Sawtooth Wave] Forces the number of periods in the signal to be an integer. If you select this option and change the values of **Sample rate (S/s)** or **Block size (samples)**, Create Analog Signal coerces the value of **Frequency (Hz)** so that the number of periods remains an integer.
- **Duty cycle (%)**—[Signal type: Square Wave] Specifies the percentage of each period a square wave remains high.
- **Noise type**—[Signal type: Noise Signal] Specifies the type of noise the probability density function represents. Create Analog Signal defines the **Noise type** by the distribution of frequencies that appear on a histogram of the signal.
    - **White (Gaussian)**—(Default) Creates a noise signal with a Gaussian distribution of frequencies.
    - **White (Rectangular)**—Creates a noise signal with a rectangular distribution of frequencies.
    - **White (Triangular)**—Creates a noise signal with a triangular distribution of frequencies.
- **Level (Vrms)**—[Signal type: Noise Signal] Specifies the noise level. The default is 1 V_rms. This option is available only when you select **White (Gaussian)** as the **Noise type**.

- **Start freq. (Hz)**—[Signal type: Multi-tone] Specifies the start frequency of the multi-tone signal. This step coerces the start frequency to be a multiple of the frequency resolution defined by the ratio of the **Sample rate (S/s)** divided by the **Block size (samples)**.
- **Stop freq. (Hz)**—[Signal type: Multi-tone] Specifies the stop frequency of the multi-tone signal. This step coerces the stop frequency to equal **Start freq. (Hz)** + *n* * **Step freq. (Hz)**, where *n* is an integer number.
- **Step freq. (Hz)**—[Signal type: Multi-tone] Specifies the step frequency of the multi-tone signal. This step coerces the step frequency to be a multiple of the frequency resolution defined by the ratio of the **Sample rate (S/s)** divided by the **Block size (samples)**.
- **Formula**—[Signal type: Formula] Specifies the formula string that defines the signal. The default is a*sin(w*t). You can use the following defined variable names:
    - f—Frequency equal to the **Frequency (Hz)** input.
    - a—Amplitude equal to the **Amplitude (V)** input.
    - w—2*pi*f.
    - n—Current number of samples generated.
    - t—Number of elapsed seconds.
    - fs—Sampling frequency equal to the **Sample rate (S/s)**.

- **Sampling Conditions**—Contains the following options:

	– **Sample rate (S/s)**—Specifies the sampling rate of the signal in samples per second. The default is 100 kS/s. – **Block size (samples)**—Specifies the number of samples in the signal. The default is 1000 samples. • **Optional Outputs**—Contains the following option: – **Export coerced values**—Exports coerced frequency values as output scalar values. Create Analog Signal can coerce frequency values when you select a periodic signal type and you place a checkmark in the **N periods** checkbox, or when you select a multi-tone signal type.
Execution Control	Contains the following option: • **Post-execution delay (ms)**—Specifies the amount of time to wait after the step executes.

Create Analog Signal Details

In continuous signal mode, the signal that Create Analog Signal creates at each iteration is contiguous to the previous iteration. The result is a signal with a continuously increasing time stamp and phase continuity. You can use the continuous signal mode to continuously generate a signal with arbitrary frequency or a non-repetitive noise signal with an analog output device if the device supports updating the output buffer while running.

In repeated signal mode, this step calculates the signal only during the first iteration of the step after you click the **Run** button or, in LabVIEW SignalExpress, the **Reset Signal** button and each time you change a configuration parameter. The signal then repeats with the same time stamp and start phase. National Instruments recommends that you use repeated signal mode if you generate the signal with an analog output device that does not support updating the output buffer while running, such as the devices that NI-FGEN Arbitrary Waveform supports.

Signal Frequency Coercion

If you select a standard periodic signal type, you can place a checkmark in the **N periods** checkbox to coerce the signal frequency you specify so that Create Analog Signal creates a signal with an integer number of periods. The periods repeat without phase discontinuities. If you place a checkmark in the **N periods** checkbox, the actual coerced values overwrite the input value you type. When you select a multi-tone signal type, Create Analog Signal coerces the **Start freq. (Hz)**, **Stop freq. (Hz)**, and **Step freq. (Hz)** values to create a repeatable signal. Place a checkmark in the **Export coerced values** checkbox to export coerced frequency values as an output of Create Analog Signal.

Create Digital Signal Express VI

Owning Palette: Create Signals Express VIs

Installed With: LabVIEW SignalExpress

Creates different types of digital signals. Depending on the option you select in the **Signal type** pull-down menu, this step can create a ramp, marching values, single value, random, or toggle pattern.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Output Signal	Displays the signal the step creates.
Configuration	Contains the following options: • **Signal type**—Specifies the type of digital waveform to create. You can select from the following options: – **Ramp**—Creates a digital waveform that contains a binary count-up pattern that starts at zero and counts up by one until it reaches 2^{n-1}, where n = **Number of signals**. – **Marching Values**—Creates a digital waveform in which a binary value placed on the first signal of the first sample is logically shifted to the next signal on each subsequent sample of the waveform. The **Hold value** field specifies the initial value, and the **Marching value** field specifies how the value shifts for subsequent samples. – **Single Value**—Creates a digital waveform in which all bits are set to 0, 1, Z, L, H, X, T, or V, depending on the **Value** you specify. – **Random**—Creates a digital waveform that contains a random digital pattern of 0s and 1s. The random pattern generated assumes no mathematically determinable sequence of values. – **Toggle**—Creates a digital waveform in which the even numbered samples contain binary values you define in the **Toggle value 1** field and the odd numbered samples contain binary values you define in the **Toggle value 2** field.

	• **Hold value**—Specifies the binary value of the generated digital waveform. This option is only available when you select the **Marching Values** option from the **Signal type** pull-down menu. • **Marching value**—Specifies the binary value that marches across the signals of the generated digital waveform. This option is only available when you select the **Marching Values** option from the **Signal type** pull-down menu. • **Value**—Specifies the digital bit state of the generated digital waveform. This option is available only when you select the **Signal Value** option from the **Signal type** pull-down menu. • **Toggle value 1**—Specifies the first digital bit state of the generated digital waveform. This option is available when you select the **Toggle** option from the **Signal type** pull-down menu. • **Toggle value 2**—Specifies the second digital bit state of the generated digital waveform. This option is available when you select the **Toggle** option from the **Signal type** pull-down menu. • **Create one output per signal**—Specifies whether to create an output group that contains a separate signal for each line in the digital waveform. • **Number of signals**—Specifies the number of signals to include in the generated digital waveform. • **Block size (samples)**—Specifies the number of samples in the signal. The default is 1000 samples. • **Sample rate (S/s)**—Specifies the sampling rate of the signal in samples per second. The default is 100 kS/s.
Execution Control	Contains the following option: • **Post-execution delay (ms)**—Specifies the amount of time to wait after the step executes.

Signal Names	Contains the following options: • **Signal names table**—Specifies custom names for the digital signals this step creates. • **Reset to default**—Resets any modified digital signal names to its default name.

Load/Save Signals Express VIs

Owning Palette: LabVIEW SignalExpress™ Express VIs

Use the Signal I/O VIs to acquire and generate signals from a hardware device, import or export data from ASCII and LVM files; and import data from SPICE files.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Palette Object	Description
Load from ASCII (Frequency Domain)	Imports data from an ASCII file.
Load from ASCII (Time Domain)	Imports data from an ASCII file.
Load from LVM (Frequency Domain)	Imports data from a text-based measurement file (.lvm).
Load from LVM (Time Domain)	Imports data from a text-based measurement file (.lvm).
Load from SPICE (Frequency Domain XY)	Imports data from a SPICE, PSpice, or Multisim file.
Load from SPICE	Imports data from a SPICE, PSpice, or Multisim file.

(Frequency Domain)	
Load from SPICE (Time Domain XY)	Imports data from a SPICE, PSpice, or Multisim file.
Load from SPICE (Time Domain)	Imports data from a SPICE, PSpice, or Multisim file.
Save to ASCII/LVM	Saves a signal to an ASCII file or a text-based measurement file (.lvm). Because the .lvm file format is designed to never overwrite the initial header information, Save to ASCII/LVM only writes the header information to the .lvm file the first time you run. If you run again, Save to ASCII/LVM does not update the header information, only the subhead information.

Load from ASCII (Frequency Domain) Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Imports data from an ASCII file.

Dialog Box Options

Dialog Box Options

Parameter	Description
Imported Signal	Displays the signal imported from an ASCII file.
Parse File	Contains the following options: • **Import file path**—Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **File preview**—Displays a preview of the contents of the file to help you determine how to parse the parameters. By default, **File preview** displays the first 50 rows from the file. If you increase the value in **Start row**, **File preview** displays 50 rows beginning with the row you specify in **Start row**. You can resize the column header to show more or less of the column. • **File Parsing Settings**—Contains the following options: – **Delimiter**—Specifies the delimiter to use to separate data. The default is Tab. This option appears only if you set **Export file type** to **Generic ASCII (.txt)**. – **Custom Delimiter**—Specifies a file delimiter other than a tab or a comma. This option appears only if you set **Export file type** to **Generic ASCII (.txt)**. – **Start row**—Specifies the row from which to begin displaying data. The default is 1. – **End row**—Specifies the last row to display data. The default is –1, which specifies to

	display all data. - **Signal names precede data row**—Specifies that the first row of the file contains the signal names. - **Decimal point**—Specifies which character to use as the decimal point. The default is . (dot). - **Domain**—Specifies the data type of the output signal.
Import Signals	Contains the following options: • **Signals**—Lists the signals in the file and specifies which signal the **Imported Signal** graph displays. • **Import**—Specifies whether to import the selected signal from the file you specify in **Import file path**. If you place a checkmark in one of the **Import** checkboxes, LabVIEW SignalExpress exports that signal to the Project View. You can then send that signal to another step or plot it on the Data View. • **Resampling Setup**—Contains the following options:  **Note** When you select an input channel as your X value and specify your own df, LabVIEW SignalExpress resamples the waveform according to the new df and based on the selected X values. This may change the number of data points in the output waveform. - **Input X values**—Specifies the X data to use for resampling. Options include None (default) or Point Index. - **Interpolation mode**—Specifies the interpolation method. Options include Coerce, Linear, or Spline. The default is Coerce. - **User specified df**—Specifies the interval size that represents the sampling step size to use to obtain data.

	- **Use same df**—Specifies whether to use the same df for all **Signals**. When you place a checkmark in this checkbox, Load from ASCII applies the df of the currently selected signal to all **Signals**. Select this option to group the imported **Signals** into one output. - **Signal Type**—Defines the type of frequency signal. Options include: Magnitude - linear, Magnitude - dB, Phase - degrees, or Phase - radians.

Load from ASCII (Time Domain) Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Imports data from an ASCII file.

Dialog Box Options

Dialog Box Options

Parameter	Description
Imported Signal	Displays the signal imported from an ASCII file.
Parse File	Contains the following options: • **Import file path**—Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **File preview**—Displays a preview of the contents of the file to help you determine how to parse the parameters. By default, **File preview** displays the first 50 rows from the file. If you increase the value in **Start row**, **File preview** displays 50 rows beginning with the row you specify in **Start row**. You can resize the column header to show more or less of the column. • **File Parsing Settings**—Contains the following options: – **Delimiter**—Specifies the delimiter to use to separate data. The default is Tab. This option appears only if you set **Export file type** to **Generic ASCII (.txt)**. – **Custom Delimiter**—Specifies a file delimiter other than a tab or a comma. This option appears only if you set **Export file type** to **Generic ASCII (.txt)**. – **Start row**—Specifies the row from which to begin displaying data. The default is 1. – **End row**—Specifies the last row to display data. The default is –1, which specifies to

	display all data. - **Signal names precede data row**—Specifies that the first row of the file contains the signal names. - **Decimal point**—Specifies which character to use as the decimal point. The default is . (dot). - **Domain**—Specifies the data type of the output signal.
Import Signals	Contains the following options: • **Signals**—Lists the signals in the file and specifies which signal the **Imported Signal** graph displays. • **Import**—Specifies whether to import the selected signal from the file you specify in **Import file path**. If you place a checkmark in one of the **Import** checkboxes, LabVIEW SignalExpress exports that signal to the Project View. You can then send that signal to another step or plot it on the Data View. • **Resampling Setup**—Contains the following options:  **Note** When you select an input channel as your X value and specify your own dt, LabVIEW SignalExpress resamples the waveform according to the new dt and based on the selected X values. This may change the number of data points in the output waveform. - **Input X values**—Specifies the X data to use for resampling. Options include None (default) or Point Index. - **Interpolation mode**—Specifies the interpolation method. Options include Coerce, Linear, or Spline. The default is Coerce. - **User specified dt**—Specifies the interval size that represents the sampling step size to use to obtain data.

	- **Use same dt**—Indicates whether to use the same dt for all **Signals**. When you place a checkmark in this checkbox, Load from ASCII applies the dt of the currently selected signal to all **Signals**. Select this option to group the imported **Signals** into one output. • **Y axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated unit within the data. • **X axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated unit within the data.

Load from LVM (Frequency Domain) Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Imports data from a text-based measurement file (.lvm).

Dialog Box Options

Dialog Box Options

Parameter	Description
Imported Signal	Displays the signal imported from the .lvm file.
File and Signal Selection	Contains the following options: • **Import file path**—Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **Signals**—Lists the signals in the file and specifies which signal the **Imported Signal** graph displays. • **Import**—Specifies whether to import the selected signal from the file you specify in **Import file path**. If you place a checkmark in one of the **Import** checkboxes, LabVIEW SignalExpress exports that signal to the Project View. You can then send that signal to another step or plot it on the Data View. • **File Information**—Contains the following options: – **ID**—Contains the following options: • **Project**—Displays the name of the project associated with this data set. • **User**—Displays the identity of the user who saved this data. • **Save date**—Indicates the date when the data was saved. • **Save time**—Indicates the time when the data was saved. – **Notes**—Specifies miscellaneous information related to the saved data. – **UUT**—Contains the following options:

- **UUT name or description**—Specifies the name and/or description of the Unit Under Test (UUT).
- **Serial number**—Specifies the serial number of the UUT.
- **Model number**—Specifies the model number of the UUT.

- **Test**—Specifies testing information to add to the header. Contains the following options:
  - **Test name or description**—Specifies the name and/or description of the test.
  - **Series**—Specifies the test series of this data.
  - **Numbers**—Specifies the numbers in the test series to which this data corresponds.
- **Waveform**—Displays information about the waveform.
  - **Waveform name**—Specifies the name of the channel you select.
  - **Waveform notes**—Specifies miscellaneous information associated with the channel you select.

- **Domain**—Specifies the data type of the output signal. The default is Frequency Waveform.
- **Signal Type**—Defines the type of frequency signal. Options include: Magnitude - linear, Magnitude - dB, Phase - degrees, or Phase - radians.

Load from LVM (Time Domain) Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Imports data from a text-based measurement file (.lvm).

Dialog Box Options

Dialog Box Options

Parameter	Description
Imported Signal	Displays the signal imported from the .lvm file.
File and Signal Selection	Contains the following options: • **Import file path**—Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **Signals**—Lists the signals in the file and specifies which signal the **Imported Signal** graph displays. • **Import**—Specifies whether to import the selected signal from the file you specify in **Import file path**. If you place a checkmark in one of the **Import** checkboxes, LabVIEW SignalExpress exports that signal to the Project View. You can then send that signal to another step or plot it on the Data View. • **File Information**—Contains the following options: – **ID**—Contains the following options: • **Project**—Displays the name of the project associated with this data set. • **User**—Displays the identity of the user who saved this data. • **Save date**—Indicates the date when the data was saved. • **Save time**—Indicates the time when the data was saved. – **Notes**—Specifies miscellaneous information related to the saved data. – **UUT**—Contains the following options:

        - **UUT name or description**—Specifies the name and/or description of the Unit Under Test (UUT).
        - **Serial number**—Specifies the serial number of the UUT.
        - **Model number**—Specifies the model number of the UUT.
    - **Test**—Specifies testing information to add to the header. Contains the following options:
        - **Test name or description**—Specifies the name and/or description of the test.
        - **Series**—Specifies the test series of this data.
        - **Numbers**—Specifies the numbers in the test series to which this data corresponds.
    - **Waveform**—Displays information about the waveform.
        - **Waveform name**—Specifies the name of the channel you select.
        - **Waveform notes**—Specifies miscellaneous information associated with the channel you select.
- **Domain**—Specifies the data type of the output signal.
- **Y axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated unit within the data.
- **X axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated

	unit within the data.

Load from SPICE (Frequency Domain XY) Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Imports data from a SPICE, PSpice, or Multisim file.

SPICE is a general-purpose circuit simulation program for nonlinear DC, nonlinear transient, and linear AC analyses. PSpice is SPICE for Windows and is part of the OrCAD® product line by Cadence Design Systems Inc. Multisim is an integrated desktop design entry and simulation system for design engineers created by Electronics Workbench.

Dialog Box Options

Dialog Box Options

Parameter	Description
Imported Signal	Displays the signal you imported from a SPICE, PSpice, or Multisim file.
File and Signal Selection	Contains the following options: • **Simulation file type**—Specifies the type of file to import. Options include SPICE, PSpice, and Multisim. The default is SPICE. • **Import file path**—Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **Signals in file**—Lists the signals in the file and specifies which signal the **Imported Signal** graph displays. • **Import**—Specifies whether to import the selected signal from the file you specify in **Import file path**. If you place a checkmark in one of the **Import** checkboxes, LabVIEW SignalExpress exports that signal to the Project View. You can then send that signal to another step or plot it on the Data View. • **Signal Type**—Defines the type of frequency signal. Options include: Magnitude - linear, Magnitude - dB, Phase - degrees, or Phase - radians. • **Domain**—Specifies the data type of the output signal. The default is XY Pairs-Frequency. • **Magnitude**—Specifies the data type of the output signal as **Linear** or **dB**. This option is available only when the file you import contains complex signals that include real and imaginary components. • **Phase**—Specifies the data type of the output signal

	as **degrees** or **radians**. This option is available only when the file you import contains complex signals that include real and imaginary components.

Load from SPICE (Frequency Domain) Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Imports data from a SPICE, PSpice, or Multisim file.

SPICE is a general-purpose circuit simulation program for nonlinear DC, nonlinear transient, and linear AC analyses. PSpice is SPICE for Windows and is part of the OrCAD® product line by Cadence Design Systems Inc. Multisim is an integrated desktop design entry and simulation system for design engineers created by Electronics Workbench.

Dialog Box Options

Dialog Box Options

Parameter	Description
Imported Signal	Displays the signal you imported from a SPICE, PSpice, or Multisim file.
File and Signal Selection	Contains the following options: • **Simulation file type**—Specifies the type of file to import. Options include SPICE, PSpice, and Multisim. The default is SPICE. • **Import file path**—Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **Signals in file**—Lists the signals in the file and specifies which signal the **Imported Signal** graph displays. • **Import**—Specifies whether to import the selected signal from the file you specify in **Import file path**. If you place a checkmark in one of the **Import** checkboxes, LabVIEW SignalExpress exports that signal to the Project View. You can then send that signal to another step or plot it on the Data View. • **Resampling Setup**—Contains the following options: – **Interpolation mode**—Specifies the interpolation method. Options include Coerce, Linear, or Spline. The default is Coerce. – **df**—Specifies the df. The default is 1. – **Signal Type**—Defines the type of frequency signal. Options include: Magnitude - linear, Magnitude - dB, Phase - degrees, or Phase - radians.

	- **Magnitude**—Specifies the data type of the output signal as **Linear** or **dB**. This option is available only when the file you import contains complex signals that include real and imaginary components. - **Phase**—Specifies the data type of the output signal as **degrees** or **radians**. This option is available only when the file you import contains complex signals that include real and imaginary components. • **Domain**—Specifies the data type of the output signal. The default is Frequency Waveform.

Load from SPICE (Time Domain XY) Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Imports data from a SPICE, PSpice, or Multisim file.

SPICE is a general-purpose circuit simulation program for nonlinear DC, nonlinear transient, and linear AC analyses. PSpice is SPICE for Windows and is part of the OrCAD® product line by Cadence Design Systems Inc. Multisim is an integrated desktop design entry and simulation system for design engineers created by Electronics Workbench.

Dialog Box Options

Dialog Box Options

Parameter	Description
Imported Signal	Displays the signal you imported from a SPICE, PSpice, or Multisim file.
File and Signal Selection	Contains the following options: • **Simulation file type**—Specifies the type of file to import. Options include SPICE, PSpice, and Multisim. The default is SPICE. • **Import file path**—Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **Signals in file**—Lists the signals in the file and specifies which signal the **Imported Signal** graph displays. • **Import**—Specifies whether to import the selected signal from the file you specify in **Import file path**. If you place a checkmark in one of the **Import** checkboxes, LabVIEW SignalExpress exports that signal to the Project View. You can then send that signal to another step or plot it on the Data View. • **Domain**—Specifies the data type of the output signal. The default is XY Pairs-Time. • **Y axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated unit within the data. • **X axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom**

	specifies that the input signal contains an associated unit within the data.

Load from SPICE (Time Domain) Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Imports data from a SPICE, PSpice, or Multisim file.

SPICE is a general-purpose circuit simulation program for nonlinear DC, nonlinear transient, and linear AC analyses. PSpice is SPICE for Windows and is part of the OrCAD® product line by Cadence Design Systems Inc. Multisim is an integrated desktop design entry and simulation system for design engineers created by Electronics Workbench.

Dialog Box Options

Dialog Box Options

Parameter	Description
Imported Signal	Displays the signal you imported from a SPICE, PSpice, or Multisim file.
File and Signal Selection	Contains the following options: • **Simulation file type**—Specifies the type of file to import. Options include SPICE, PSpice, and Multisim. The default is SPICE. • **Import file path**—Specifies the name and location of the file you want to import. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **Signals in file**—Lists the signals in the file and specifies which signal the **Imported Signal** graph displays. • **Import**—Specifies whether to import the selected signal from the file you specify in **Import file path**. If you place a checkmark in one of the **Import** checkboxes, LabVIEW SignalExpress exports that signal to the Project View. You can then send that signal to another step or plot it on the Data View. • **Resampling Setup**—Contains the following options: – **Interpolation mode**—Specifies the interpolation method. Options include Coerce, Linear, or Spline. The default is Coerce. – **dt**—Specifies the dt. The default is 1. – **Y-axis**—Specifies to display the y-axis as **linear** or **dB**. • **Domain**—Specifies the data type of the output

	signal. • **Y axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated unit within the data. • **X axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated unit within the data.

Save to ASCII/LVM Express VI

Owning Palette: Load/Save Signals Express VIs

Installed With: LabVIEW SignalExpress

Saves a signal to an ASCII file or a text-based measurement file (.lvm). Because the .lvm file format is designed to never overwrite the initial header information, Save to ASCII/LVM only writes the header information to the .lvm file the first time you run. If you run again, Save to ASCII/LVM does not update the header information, only the subhead information.

Dialog Box Options

Dialog Box Options

Parameter	Description
Input Signals	Displays the input signals.
Signals	Contains the following options: • **Add Input**—Adds the signal you want to write to file. • **Remove**—Removes the highlighted input. • **Inputs**—Lists the signals to write to the ASCII file or the text-based measurement file (.lvm). • **Input Data**—Specifies the signal you want to save to the file.
File Settings	Contains the following options: • **Export file path**—Specifies the location where you want to save the file. You can specify an absolute or relative path to the file. If you specify an absolute path, this step saves the path with the project. If you specify a relative path and you do not save the project, this step assumes the path is relative to the **My Documents** folder. If you specify a relative path and you save the project, the path is relative to the location where you save the project. • **If file already exists**—Specifies how LabVIEW SignalExpress saves data to an existing file. Contains the following options: – **Overwrite**—Replaces data in the existing file. – **Overwrite once, then append to file**—Overwrites the file once and then appends to information to the end of the existing file. – **Overwrite and backup previous**—Performs a backup of the file and replaces data in the existing file. – **Append to file**—Appends the data to the existing file. – **Next available file name**—Appends the

next sequential number to the filename. For example, if test.lvm exists, LabVIEW SignalExpress saves the file as test1.lvm.

- **Export file type**—Specifies in which file format to save the file. Options include text-based measurement file (.lvm) or Generic ASCII. The default is text-based measurement file (.lvm). If you select Generic ASCII, you also can save as .csv or .txt. A .lvm file format contains header information and signal data. An ASCII file format contains only signal data. Both file formats are ASCII. You can use the Load from LVM or Load from ASCII steps to load these files into LabVIEW SignalExpress.
- **LVM File Annotations**—Contains the following options:
  - **ID**—Contains the following options:
    - **Project**—Displays the name of the project associated with this data set.
    - **User**—Displays the identity of the user who saved this data.
  - **Notes**—Specifies miscellaneous information associated with the channel you select.
  - **UUT**—Contains the following options:
    - **UUT name or description**—Specifies the name and/or description of the Unit Under Test (UUT).
    - **Serial number**—Specifies the serial number of the UUT.
    - **Model number**—Specifies the model number of the UUT.
  - **Test**—Specifies testing information to add to the header. Contains the following options:
    - **Test name or description**—Specifies the name and/or description of the test.

- **Series**—Specifies the test series of this data.
- **Numbers**—Specifies the numbers in the test series to which this data corresponds.

- **Waveform**—Displays information about the waveform.
  - **Waveform name**—Specifies the name of the channel you select.
  - **Waveform notes**—Specifies miscellaneous information associated with the channel you select.

- **Delimiter**—Specifies the delimiter to use to separate data. The default is Tab. This option appears only if you set **Export file type** to **Generic ASCII (.txt)**.
- **Custom Delimiter**—Specifies a file delimiter other than a tab or a comma. This option appears only if you set **Export file type** to **Generic ASCII (.txt)**.
- **Include Signal Names**—Includes the names of the signals in the ASCII file. This option appears only if you set **Export file type** to **Generic ASCII (.txt)**.
- **X Value Columns**—Contains the following options:
  - **One column per channel**—Creates a separate column for time data each channel generates. This option includes a column of values from the x-axis for every column of values from the y-axis.
  - **One column only**—Creates only one column for the time data the channels generate. This option includes only one column of values from the x-axis.
  - **Empty time column**—Creates an empty column for the time data each channel generates. This option does not include the data from the x-axis.

  This option appears only if you set **Export file type**

	to **Generic ASCII (.txt)**. • **Time Axis Preference**—Contains the following options: – **Absolute Time**—Displays the time elapsed since 12:00 a.m., Friday, January 1, 1904, Universal Time. – **Relative Time**—Displays the time in milliseconds starting from 0. This option appears only if you set **Export file type** to **Generic ASCII (.txt)**.

Processing Express VIs

Owning Palette: LabVIEW SignalExpress™ Express VIs

Use the Processing Express VIs to filter, scale, resample, and average signals; apply windowing and perform arithmetic operations; and, interactively align two signals.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Palette Object	Description
Arithmetic (Frequency Domain)	Performs arithmetic operations on two signals. The available operations change depending on the type of the input signals you select.
Arithmetic (Time Domain)	Performs arithmetic operations on two signals. The available operations change depending on the type of the input signals you select.
Convert Analog to Digital	Converts an analog waveform to a digital waveform.
Convert Digital to Analog	Converts an input digital waveform into an analog waveform.
Filter	Filters a time signal using an infinite impulse response (IIR) or finite impulse response (FIR) filter. Use this step to remove or attenuate unwanted frequencies from a signal using various standard filter types and topologies.
Formula	Performs math operations on up to four input variables. By default, Formula processes only one variable, but you can place checkmarks in the **Enable** checkboxes to enable more variables. Enter a formula in the **Formula** field using the variable names you specify in the **Alias** fields to represent the **Input variable** values.

Interactive Alignment	Aligns two plots so you can compare them. You can align the **Test signal in** signal with the **Ref. signal in** signal manually by dragging and/or expanding the **Test** plot on the graph or by using algorithms to automatically align steps, pulses, or periodic parameters.
Scaling and Conversion (Frequency Domain)	Performs gain and offset scaling on a time signal, gain scaling on a frequency-domain magnitude signal, or time-delay correction on a frequency-domain phase signal.
Scaling and Conversion (Time Domain)	Performs gain and offset scaling on a time signal, gain scaling on a frequency-domain magnitude signal, or time-delay correction on a frequency-domain phase signal.
Subset and Resample (Frequency Domain)	Extracts a subset of an input signal using the **Start frequency** and **Subset length** you specify or resamples an input signal using the frequency interval (**df**) you specify. If you specify a larger **df**, Subset and Resample downsamples the signal. If you specify a smaller **df**, Subset and Resample upsamples the signal.
Subset and Resample (Time Domain)	Extracts a subset of an input signal using the **Start position** and **Subset length** you specify or resamples an input signal using the time interval (**dt**) you specify. If you specify a larger **dt**, Subset and Resample downsamples the signal. If you specify a smaller **dt**, Subset and Resample upsamples the signal.
Time Averaging	Performs time averaging on a time signal or scalar input.
Window	Applies the window you select to the time-domain signal.

Arithmetic (Frequency Domain) Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Performs arithmetic operations on two signals. The available operations change depending on the type of the input signals you select.

To select the correct type of operation, select the first signal to process from the **Input signal 1** pull-down menu. The **Input signal 2** pull-down menu displays only the list of compatible signals, and the step displays the available operations.

Dialog Box Options

Dialog Box Options

Parameter	Description
Input Signals	Displays the two input signals. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Resulting Signal	Displays the signal that results from the arithmetic operation.
Input	Contains the following options: • **Input signal 1**—Specifies the first input signal. • **Input signal 2**—Specifies the second input signal.
Configuration	Contains the following option: • **Operation Setup**—Contains the following options: – **Operation**—Specifies to add, subtract, multiply, or divide the signals after alignment. The **Resulting Signal** graph displays the result of the operation. The default is Subtract. – **Output unit**—Specifies to represent the result of a magnitude operation in decibels or to represent the result of a phase operation in degrees or radians. – **Interpolate if needed**—Resamples the signals to align the frequency bins. – **Interpolation type**—Contains the following options: • **Coerce**—Sets each output

	sample value to equal the input sample value that is closest to it in frequency. • **Linear**—Sets each output sample value to be a linear interpolation between the two input samples that are closest to it in frequency. • **Spline**—Uses the spline interpolation algorithm to compute the resampled values. • **Filter based**—Uses an interpolation method based on the convolution of the signal with a finite impulse response (FIR) filter. – **Resulting interval**—Specifies if the resulting signal covers the Common or Global frequency span of the signals. The default is Global.
Filter Setup	Available when you select **Filter based** from **Interpolation type**. Contains the following option: • **FIR Filter Specification**—Contains the following options: – **Normalized bandwidth**—The normalized cut-off frequency of the FIR filter to use. The default is 0.4000. – **Alias rejection (dB)**—The minimum stopband attenuation of the FIR filter to use. The default is 80 dB.

Arithmetic (Time Domain) Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Performs arithmetic operations on two signals. The available operations change depending on the type of the input signals you select.

To select the correct type of operation, select the first signal to process from the **Input signal 1** pull-down menu. The **Input signal 2** pull-down menu displays only the list of compatible signals, and the step displays the available operations.

Dialog Box Options

Dialog Box Options

Parameter	Description
Input Signals	Displays the two input signals. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Resulting Signal	Displays the signal that results from the arithmetic operation.
Input	Contains the following options: • **Input signal 1**—Specifies the first input signal. • **Input signal 2**—Specifies the second input signal.
Configuration	Contains the following option: • **Operation Setup**—Contains the following options: – **Operation**—Specifies to add, subtract, multiply, or divide the signals or compute the RMS sum of the signals. The default is to add the signals. – **Ignore timestamp**—Ignores eventual differences in timestamps by forcing the second signal timestamp to be equal to the first signal timestamp. – **Interpolate if needed**—Resamples the signals to align the samples. – **Optimized for single record**—Optimizes the resampling operation for executing Arithmetic once. Place a checkmark in this checkbox to speed

	execution when Arithmetic runs once. – **Interpolation type**—Contains the following options: • **Coerce**—Sets each output sample value to equal the input sample value that is closest to it in time. • **Linear**—Sets each output sample value to be a linear interpolation between the two input samples that are closest to it in time. • **Spline**—Uses the spline interpolation algorithm to compute the resampled values. • **Filter based**—Uses an interpolation method based on the convolution of the signal with a finite impulse response (FIR) filter. – **Resulting interval**—Specifies if the resulting signal covers the **Common** or **Global** time interval of the signals.
Filter Setup	Available when you select **Filter based** from **Interpolation type**. Contains the following option: • **FIR Filter Specification**—Contains the following options: – **Normalized bandwidth**—The normalized cut-off frequency of the FIR filter to use. The default is 0.4000. – **Alias rejection (dB)**—The minimum stopband attenuation of the FIR filter to use. The default is 80 dB.

Convert Analog to Digital Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Converts an analog waveform to a digital waveform.

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Analog signal preview	Displays the analog waveform you want to convert into a digital waveform.
X scale slider	Selects the data displayed in the **Digital waveform preview**.
Signal name preview	Displays the signal names associated with the signals in the **Digital waveform preview**.
Digital waveform preview	Displays the digital waveform converted from the input analog waveform.
Digital preview Y scrollbar	Scrollbar that allows you to scroll through the **Digital waveform preview**.
Input	Contains the following options: • **Input signal**—Specifies the analog waveform for the step.
Configuration	Contains the following options: • **Analog full scale**—The total peak-to-peak range, or the difference between the minimum and maximum, for the analog waveform. For example, if the maximum range of a waveform is 1 and the minimum is -1, the full-scale range for the waveform is 2. • **Resolution (bits)**—Specifies the number of bits represented in the digital waveform. LabVIEW SignalExpress supports a maximum resolution of 32 bits. • **Digital data format**—Specifies which binary representation you want to use for the digital data. – **Unsigned binary**—The data is converted to unsigned binary. – **Offset binary**—The largest negative

	value (negative full-scale) is represented by all zeros, and the largest positive value (positive full-scale) is represented by all ones. Zero-scale is represented by a one (MSB) followed by all zeros, for example, binary 1000. – **2's complement**—Uses two's complement format, which is a common format for representing signed binary values. This format is similar to Offset Binary, but the MSB is inverted. • **Dithering enabled**—Specifies whether the analog waveform can be dithered. Dithering a waveform adds Gaussian noise to an analog input signal to increase resolution.
Signal Names	Contains the following options: • **Signal names table**—Allows you to specify custom names for your signals. • **Reset to default**—Specifies whether the Signal names are reset to their default states.

Block Diagram Inputs

Parameter	Description
error in (no error)	Describes error conditions that occur before this VI or function runs.

Block Diagram Outputs

Parameter	Description
error out	Contains error information. If **error in** indicates that an error occurred before this VI or function ran, **error out** contains the same error information. Otherwise, it describes the error status that this VI or function produces. Right-click the **error out** front panel indicator and select **Explain Error** from the shortcut menu for more information about the error.

Filter Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Filters a time signal using an infinite impulse response (IIR) or finite impulse response (FIR) filter. Use this step to remove or attenuate unwanted frequencies from a signal using various standard filter types and topologies.

In LabVIEW SignalExpress, the Filter step filters the input signal continuously. The step resets the signal to its original value the first time the step runs, if LabVIEW SignalExpress detects a discontinuity in the input signal, or if you press the **Reset Filter** button.

In LabVIEW, the Filter Express VI filters the input signal continuously. The Express VI resets the signal to its original value the first time the Express VI runs, if LabVIEW detects a discontinuity in the input signal, or if the **reset** input receives a TRUE value.

Details

Dialog Box Options

Block Diagram Inputs

Block Diagram Outputs

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input Signals	Displays the input signal to filter.
Autoscale amplitude	Autoscales the preview graph along the y-axis. The default is to autoscale the amplitude.
Displayed signal	Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Output Signals	Displays the filtered signal.
Autoscale amplitude	Autoscales the preview graph along the y-axis. The default is to autoscale the amplitude.
Input	The following option applies to the LabVIEW SignalExpress step: • **Input signal**—Specifies the input signal to filter.
Configuration	Contains the following option: • **Filter Specifications**—Contains the following options: – **Mode**—Specifies the mode of filter to use. You can select from the following options: • **IIR filter**—Specifies an IIR filter, which is a recursive digital filter with infinite impulse response. IIR filters operate on current and past input values and current and past output values. IIR filters can achieve the same level of attenuation as FIR filters but with fewer coefficients. For this

reason, IIR filters can be faster and more efficient than FIR filters.

- **FIR filter**—Specifies an FIR filter, which is a digital filter with finite impulse response. FIR filters operate only on current and past input values. Because an FIR filter does not depend on past outputs, the impulse response decays to zero in a finite amount of time. Use FIR filters for applications that require linear phase responses.

- **Type**—Specifies the type of filter to use. You can select from the following options:
  - **Lowpass**—(Default) Passes low frequencies and attenuates high frequencies.
  - **Highpass**—Passes high frequencies and attenuates low frequencies.
  - **Bandpass**—Passes a certain band of frequencies. Use the **Low cutoff (Hz)** and the **High cutoff (Hz)** fields to specify the band.
  - **Bandstop**—Attenuates a certain band of frequencies. Use the **Low cutoff (Hz)** and the **High cutoff (Hz)** fields to specify the band.
- **Topology**—[Mode: IIR Filter] Specifies the design type of an IIR filter. You can select from the following options:
  - **Off**—Does not filter the signal.

- **Butterworth**—(Default) Applies a Butterworth filter to the signal. Butterworth filters have a smooth, monotonically decreasing frequency response.
- **Chebyshev**—Applies a Chebyshev filter to the signal. Chebyshev filters can achieve a sharper transition between the passband and the stopband with a lower order filter than Butterworth filters.
- **Inverse Chebyshev**—Applies an Inverse Chebyshev filter to the signal. Inverse Chebyshev filters are similar to Chebyshev filters, but they distribute the error over the stopband instead of the passband and are maximally flat in the passband instead of the stopband.
- **Elliptic**—Applies an Elliptic filter to the signal. Elliptic filters minimize the peak error by distributing it over the passband and the stopband. Elliptic filters provide the sharpest transition between the passband and the stopband.
- **Bessel**—Applies a Bessel filter to the signal. Bessel filters have maximally flat response in both magnitude and phase. You can use Bessel filters to reduce nonlinear phase distortion inherent in all IIR filters.

– **Order**—[Mode: IIR filter] Determines the order of an IIR filter, which must be

	greater than zero. The default is 2. Increasing the value of **Order** causes the transition between the passband and the stopband to become steeper. However, as the value of **Order** increases, the processing speed becomes slower, and the number of distorted points at the start of the signal increases. – **Number of taps**—[Mode: FIR filter] Specifies the total number of FIR coefficients, which must be greater than zero. The default is 49. Increasing the value of taps causes the transition between the passband and the stopband to become steeper. However, as the value of **Number of taps** increases, the processing speed becomes slower. – **Cutoff (Hz)**—[Type: Lowpass, Highpass] Specifies the cutoff frequency of the filter when you select a **Lowpass** or **Highpass** filter type. The default is 100 Hz. – **Low cutoff (Hz)**—[Type: Bandpass, Bandstop] Specifies the lower cutoff frequency when you select a **Bandpass** or **Bandstop** filter type. The default is 100 Hz. – **High cutoff (Hz)**—[Type: Bandpass, Bandstop] Specifies the higher cutoff frequency when you select a **Bandpass** or **Bandstop** filter type. **High cutoff (Hz)** must be greater than **Low cutoff (Hz)** and observe the Nyquist criterion. The default is 200 Hz. – **Filter Magnitude Response (dB)**—Displays the magnitude response of the filter you specify.

Block Diagram Inputs

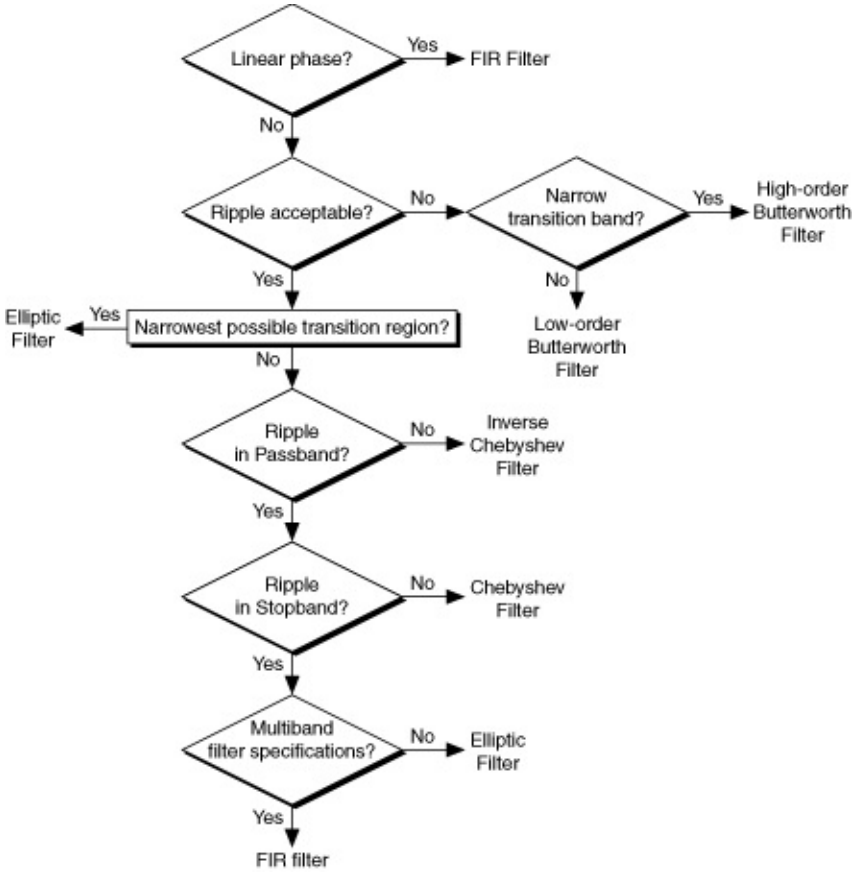
Parameter	Description
error in (no error)	Describes error conditions that occur before this VI or function runs.
reset	Specifies if the filter has to be reset. The default is FALSE. The filter is automatically reset the first time this VI is called.
signal in	Specifies the input signal.

Block Diagram Outputs

Parameter	Description
error out	Contains error information. If **error in** indicates that an error occurred before this VI or function ran, **error out** contains the same error information. Otherwise, it describes the error status that this VI or function produces. Right-click the **error out** front panel indicator and select **Explain Error** from the shortcut menu for more information about the error.
filtered data	Returns the filtered signal.

Filter Details

The best filter **Mode**, **Type**, and **Topology** to use depends on the analysis you want to perform. Use the following illustration as a guideline for selecting the appropriate filter for an analysis project:



Formula Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Performs math operations on up to four input variables. By default, Formula processes only one variable, but you can place checkmarks in the **Enable** checkboxes to enable more variables. Enter a formula in the **Formula** field using the variable names you specify in the **Alias** fields to represent the **Input variable** values.

You can use time waveform or scalar values for the **Input variable** values. If you apply the **Formula** to multiple time waveforms, the waveforms must be the same size or Formula returns an error.

If all the **Input variable** values are time waveforms, Formula returns a time waveform. If all the **Input variable** values are scalar values, Formula returns a scalar value. If the **Input variable** values are a mix of time waveforms and scalar values, Formula returns a time waveform.

Details

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input and Output Waveforms	Displays the waveforms you specify in the **Input variable** fields and the processed data that Formula returns after you apply the **Formula** to the waveforms. This graph appears only when you select a waveform as an **Input variable**.
Input and Output Scalars	Displays the scalar values you specify in the **Input variable** fields and the processed data that Formula returns when you apply the **Formula** to the values. This chart appears only when you select a scalar value as an **Input variable**.
Input variable 0	Specifies the first variable to use.
Alias 0	Specifies an alias name for the first variable.
Enable 1	Enables a second variable.
Input variable 1	Selects the second variable to use.
Alias 1	Specifies an alias name for the second variable.
Enable 2	Enables a third variable.
Input variable 2	Selects the third variable to use.
Alias 2	Specifies an alias name for the third variable.
Enable 3	Enables a fourth variable.
Input variable 3	Selects the fourth variable to use.
Alias 3	Specifies an alias name for the fourth variable.
Operation Setup	Contains the following options: • **Formula**—Specifies the formula for the computation. • **Valid**—Reports if the formula is valid or invalid. • **Ignore timestamps**—Specifies to ignore timestamps on input variables that are time waveforms. Place a checkmark in this checkbox to

	use time waveforms in the **Formula** that have different timestamps.

Formula Details

If you specify multiple waveforms for the **Input variable** values, LabVIEW SignalExpress applies the **Formula** to each sample of the waveforms, which is why multiple waveforms must be the same size. If you specify a mix of waveforms and scalar values for the **Input variable** values, LabVIEW SignalExpress applies the **Formula** to the scalar value and each sample of the waveform. For example, if you specify a **Formula** that multiplies a waveform and a scalar value, LabVIEW SignalExpress multiplies each sample of the waveform by the scalar value.

The following table displays the math functions Formula supports.

Function	Description
abs(*x*)	Returns the absolute value of *x*.
acos(*x*)	Computes the inverse cosine of *x* in radians.
acosh(*x*)	Computes the inverse hyperbolic cosine of *x*.
asin(*x*)	Computes the inverse sine of *x* in radians.
asinh(*x*)	Computes the inverse hyperbolic sine of *x*.
atan(*x*)	Computes the inverse tangent of *x* in radians.
atanh(*x*)	Computes the inverse hyperbolic tangent of *x*.
ceil(*x*)	Rounds *x* to the next higher integer (smallest integer ≤ *x*).
ci(*x*)	Evaluates the cosine integral for any real nonnegative number *x*.
cos(*x*)	Computes the cosine of *x*, where *x* is in radians.
cosh(*x*)	Computes the hyperbolic cosine of *x*.
cot(*x*)	Computes the cotangent of *x* (1/tan(*x*)), where *x* is in radians.
csc(*x*)	Computes the cosecant of *x* (1/sin(*x*)), where *x* is in radians.
exp(*x*)	Computes the value of *e* raised to the *x* power.
expm1(*x*)	Computes one less than the value of *e* raised to the *x* power ((*e*^*x*)–1).
floor(*x*)	Truncates *x* to the next lower integer (largest integer ≤ *x*).
getexp(*x*)	Returns the exponent of *x*.
gamma(*x*)	Evaluates the gamma function or incomplete gamma function

	for *x*.
getman(*x*)	Returns the mantissa of *x*.
int(*x*)	Rounds *x* to the nearest integer.
intrz(*x*)	Rounds *x* to the nearest integer between *x* and zero.
ln(*x*)	Computes the natural logarithm of *x* (to the base of *e*).
lnp1(*x*)	Computes the natural logarithm of (*x* + 1).
log(*x*)	Computes the logarithm of *x* (to the base of 10).
log2(*x*)	Computes the logarithm of *x* (to the base of 2).
rand( )	Produces a floating-point number between 0 and 1 exclusively.
si(*x*)	Evaluates the sine integral for an real number *x*.
sec(*x*)	Computes the secant of *x*, where *x* is in radians (1/cos(*x*)).
sign(*x*)	Returns 1 if *x* is greater than 0, returns 0 if *x* is equal to 0, and returns –1 if *x* is less than 0.
sin(*x*)	Computes the sine of *x*, where *x* is in radians.
sinc(*x*)	Computes the sine of *x* divided by *x* (sin(*x*)/x), where *x* is in radians.
sinh(*x*)	Computes the hyperbolic since of *x*.
spike(*x*)	Generates the spike function for any real number *x*.
sqrt(*x*)	Computes the square root of *x*.
step(*x*)	Generates the step function for any real number *x*.
tan(*x*)	Computes the tangent of *x*, where *x* is in radians.
tanh(*x*)	Computes the hyperbolic tangent of *x*.

Interactive Alignment Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Aligns two plots so you can compare them. You can align the **Test signal in** signal with the **Ref. signal in** signal manually by dragging and/or expanding the **Test** plot on the graph or by using algorithms to automatically align steps, pulses, or periodic parameters.

Details

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input Signals	Displays the two signals to align.
Autoscale Y(x)	Adjusts the vertical scale to reflect the data from the input signals.
Comparison Result Signal	Displays the comparison signal that results from the operation you specified with **Operation** in the **Resampling and Comparison Setup** section of the **Resampling** page.
Autoscale Comparison Signal	Adjusts the vertical scale to reflect the result of the operation on the two aligned signals.
Autoscale x	Adjusts the time scale to reflect the data to display.
Input/Output	Contains the following options: • **Ref. signal in**—Specifies the reference input signal. • **Test signal in**—Specifies the test input signal to align with the reference signal. • **Export aligned signals**—Exports the **Ref. signal in** and **Test signal in** signals to the Project View. The Interactive Alignment step resamples the **Test signal in** to match the **Ref. signal in** timing parameters. • **Export x-offset result**—Exports the x-offset value the **Geometry Parameters** section of the **Alignment** page displays to the Project View. • **Export y-offset result**—Exports the y-offset value the **Geometry Parameters** section of the **Alignment** page displays to the Project View. • **Export x-gain result**—Exports the x-gain value the **Geometry Parameters** section of the **Alignment** page displays to the Project View. • **Export y-gain result**—Exports the y-gain value the **Geometry Parameters** section of the

	Alignment page displays to the Project View.
Alignment	Contains the following options: • **Geometry Parameters**—Contains the following options: – **x-offset**—Sets or returns the time shift (offset) of the alignment operation. – **y-offset**—Sets or returns the amplitude offset of the alignment operation. – **x-gain**—Sets or returns the time stretch (gain) of the alignment operation. – **y-gain**—Sets or returns the amplitude gain of the alignment operation. – **Ignore x0**—Forces the timestamp value of the test signal to equal the timestamp value of the reference signal. – **Allow x-offset**—Allows manual time shift (offset) of the test signal. – **Allow y-offset**—Allows manual amplitude offset of the test signal. – **Allow x-gain**—Allows manual time stretch (gain) of the test signal. – **Allow y-gain**—Allows manual amplitude gain of the test signal. • **Alignment Conditions**—Contains the following options: – **Mode**—Specifies the mode Interactive Alignment uses to align the signals. You can select from the following options: • **Manual**—Allows manual alignment of the test signal. • **Auto-Impulse**—Selects an automatic alignment algorithm based on the assumption that the signals include a positive or negative impulse pattern. • **Auto-Step**—Selects an automatic

alignment algorithm based on the assumption that the signals include a rising or falling step pattern.

- **Auto-Periodic**—Selects an automatic alignment algorithm based on the assumption that the signals are periodic.

- **Invert signal**—Inverts the input test signal.
- **Criterion**—Specifies the following alignment criterion:

  - **Align Base and Peak**—Aligns the two impulses to align the base and peak levels and to align the peak positions in time.
  - **Align 50-50%**—Aligns the two impulses to superpose their respective 50% rising and falling edge points.
  - **Align Edge to User Levels**—Aligns the rising or the falling edge of the impulses to superpose the **Low level (%)** and **High level (%)** points.
  - **Align Impulse to User Levels**—Aligns the two impulses to superpose the points the **Rising level (%)** and **Falling level (%)** specify on both plots respectively.

  Step criterion:

  - **Align Low, High and User**—Aligns the two steps so the low levels (0%) and high levels (100%) are aligned and the points on the edges **Mid level (%)**

specifies are superposed.

- **Align 10% and 90%**—Aligns the two steps so the 10% and 90% points on the rising or the falling edges are superposed.
- **Align to User Levels**—Aligns the two steps so the points **Low level (%)** and **High level (%)** specify on the rising or the falling edges are superposed.

Periodic criterion:

- **Align Freq, Phase and p-p**—Aligns the two periodic signals so the fundamental tones are superposed.

- **Falling edge**—Specifies to perform the edge alignment operation on the rising or falling edge of the impulses or steps.
- **Level A**—Contains the following options:
  - **Low level (%)**—Specifies the level of a signal point to use as the low reference in an edge alignment operation. The unit is a percentage of the amplitude of the impulse or the step to align. The default is 10.
  - **Rising level (%)**—Specifies the level of the rising edge points to superpose in an impulse alignment operation. The default is 50.
  - **Mid level (%)**—Specifies the level of a signal point to use as the medium reference in a step alignment operation. The unit is a percentage of the impulse or the

	step to align. The default is 50%. - **Level B**—Contains the following options: • **High level (%)**—Specifies the level of a signal point to use as the high reference in an edge alignment operation. The unit is a percentage of the amplitude of the impulse or the step to align. The default is 90. • **Falling level (%)**—Specifies the level of the falling edge points to superpose in an impulse alignment operation. The default is 50.
Resampling	Contains the following options: • **Resampling and Comparison Setup**—Contains the following options: - **Interpolation type**—Contains the following options: • **Coerce**—Sets each output sample value to equal the input sample value that is closest to it in time. • **Linear**—Sets each output sample value to be a linear interpolation between the two input samples that are closest to it in time. • **Spline**—Uses the spline interpolation algorithm to compute the resampled values. • **Filter based**—Uses an interpolation method based on the convolution of the signal with a finite impulse response (FIR) filter. - **Operation**—Specifies to add, subtract,

	multiply, or divide the signals after alignment. The **Resulting Signal** graph displays the result of the operation. The default is Subtract. • **FIR Filter Specification**—Contains the following options: – **Normalized bandwidth**—The normalized cut-off frequency of the FIR filter to use. The default is 0.4000. – **Alias rejection (dB)**—The minimum stopband attenuation of the FIR filter to use. The default is 80 dB.

Interactive Alignment Details

Moving the Test plot

You can drag the **Test** plot to move it. When you release the mouse button, the graph performs an autoscale operation to optimize the viewing of the plots unless you remove the checkmarks from the **Autoscale** checkboxes. LabVIEW SignalExpress does not update the lower graph that displays the **Comparison Result Signal** when you drag the **Test** plot, but it performs a new comparison operation as soon as you release the mouse button.

A small cross on the upper graph called the Anchor point marks the location where you last released the mouse. To move the Anchor point position, click on the new location.

Expanding the Test plot

You also can expand the **Test** plot in both directions, corresponding to a gain/attenuation of the signal amplitude in the vertical direction and a time expansion/compression in the horizontal direction. To expand the plot, press the <Alt> key, click the graph, and drag it. The expansion keeps the position of the Anchor point unchanged; and the mouse position point in the plane at the start of the expansion follows the mouse move.

Locking Move or Expansion

You can prevent unwanted moves and/or expansions in specific directions by removing the checkmark from the corresponding **Allow x-gain**, **Allow y-gain**, **Allow x-offset**, or **Allow y-offset** checkbox. Notice that preventing certain moves or expansion conflicts with the actual position of the Anchor point and results in slightly different expansion behaviors.

Alignment Evaluation

You can evaluate the alignment on the lower graph that displays the **Comparison Result Signal**. This signal displays the result of an arithmetic operation you can specify LabVIEW SignalExpress to perform on the two aligned signals. The default is Subtract.

Exporting Alignment Results

You can export the alignment information using the following checkboxes on the **Input/Output** page:

- Export aligned signals
- Export x-offset result
- Export y-offset result
- Export x-gain result
- Export y-gain result

Resampling the Test signal

To align the **Test signal in** with the **Ref. signal in**, you must resample the signal so you can perform a sample by sample arithmetic operation like subtraction. The resampling process ensures that LabVIEW SignalExpress samples the aligned waveforms at the same rate and in phase. You can select different resampling options on the **Resampling** page.

Scaling and Conversion (Frequency Domain) Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Performs gain and offset scaling on a time signal, gain scaling on a frequency-domain magnitude signal, or time-delay correction on a frequency-domain phase signal.

Dialog Box Options

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Scaled Signal	Displays the scaled signal.
Input	Contains the following option: • **Input signal**—Specifies the analog waveform for the step.
Configuration	Contains the following option: • **Operation Setup**—Contains the following options: – **Gain**—Specifies the gain to apply to the signal. – **Gain representation**—Specifies if the gain value is represented in decibels. – **Equivalent diagram**—Displays the LabVIEW equivalent block diagram of the scaling operation you select. – **Correction delay**—Specifies the delay value to use to correct the phase signal. – **Output in degrees**—Specifies if the scaled phase signal is represented in radian or in degrees. – **Unwrap phase**—Specifies if the phase of the output signal is unwrapped.

Block Diagram Inputs

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).

Scaling and Conversion (Time Domain) Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Performs gain and offset scaling on a time signal, gain scaling on a frequency-domain magnitude signal, or time-delay correction on a frequency-domain phase signal.

Dialog Box Options

Dialog Box Options

Parameter	Description
Input Signal	Displays the input time signal to be scaled. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Scaled Signal	Displays the scaled time signal.
Input	Contains the following option: • **Input signal**—Specifies the analog waveform for the step.
Configuration	Contains the following option: • **Operation Setup**—Contains the following options: – **Pre-gain offset**—Specifies the amount of offset to add to the signal before gain. The default is 0. – **Gain**—Specifies the multiplication factor to apply to the time signal. The default is 1. – **Post-gain offset**—Specifies the amount of offset to add to the signal after gain. The default is 0. – **Equivalent diagram**—Displays the LabVIEW equivalent block diagram of the scaling operation you select.

Subset and Resample (Frequency Domain) Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Extracts a subset of an input signal using the **Start frequency** and **Subset length** you specify or resamples an input signal using the frequency interval (**df**) you specify. If you specify a larger **df**, Subset and Resample downsamples the signal. If you specify a smaller **df**, Subset and Resample upsamples the signal.

Dialog Box Options

Dialog Box Options

Parameter	Description
Input Signal	Displays the input frequency-domain signal to be scaled. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Processed Signal	Displays the processed signal.
Input	Contains the following option: • **Input signal**—Specifies the analog waveform for the step.
Configuration	Contains the following options: • **Subset Setup**—Contains the following options: – **Extract subset**—Specifies whether to extract a frequency subset of the signal using the **Start frequency** and **Subset length** you specify. – **Start frequency**—Specifies the start frequency of the subset signal. – **Subset length**—Specifies the frequency span of the subset signal. • **Resampling Setup**—Contains the following options: – **Resample**—Resamples the entire signal or subset you select. – **Unwrap phase**—Specifies if the phase of the output signal is unwrapped. – **Open interval**—Specifies if the subset is an open or closed interval. For example,

	if an input waveform contains 3 data elements at t={0, dt, 2dt}, an open interval defines the waveform as extending over the time interval 0≤t<2dt, and a closed interval defines the waveform as extending over the time interval 0≤t<3dt. Place a checkmark in this checkbox to specify an open interval. – **Interpolation type**—Contains the following options: • **Coerce**—Sets each output sample value to equal the input sample value that is closest to it in frequency. • **Linear**—Sets each output sample value to be a linear interpolation between the two input samples that are closest to it in frequency. • **Spline**—Uses the spline interpolation algorithm to compute the resampled values. • **Filter based**—Uses an interpolation method based on the convolution of the signal with a finite impulse response (FIR) filter. – **df**—Specifies the frequency resolution of the resampled frequency-domain signal. The default is 0.
Filter Setup	Available when you select **Filter based** from **Interpolation type**. Contains the following option: • **FIR Filter Specification**—Contains the following options: – **Normalized bandwidth**—The normalized cut-off frequency of the FIR filter to use. The default is 0.4000.

	– **Alias rejection (dB)**—The minimum stopband attenuation of the FIR filter to use. The default is 80 dB.

Subset and Resample (Time Domain) Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Extracts a subset of an input signal using the **Start position** and **Subset length** you specify or resamples an input signal using the time interval (**dt**) you specify. If you specify a larger **dt**, Subset and Resample downsamples the signal. If you specify a smaller **dt**, Subset and Resample upsamples the signal.

Dialog Box Options

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Processed Signal	Displays the processed signal.
Input/Output	Contains the following options: • **Input**—Contains the following option: – **Input signal**—Specifies the analog waveform for the step. • **Output**—Contains the following options: – **Export pre-subset signal**—Adds the signal that precedes the subset as an output of Subset and Resample. – **Export post-subset signal**—Adds the signal that follows the subset as an output of Subset and Resample.
Configuration	Contains the following options: • **Subset Setup**—Contains the following options: – **Extract subset**—Specifies whether to extract a time subset of the input signal using the **Start position** and **Subset length** you specify. – **Relative time**—Specifies if the **Start position** value is an absolute timestamp value or a time offset relative to the first sample of the input signal.

- **Start position**—Specifies the start position of the subset signal.
- **Subset length**—Specifies the time span of the subset signal.

- **Resampling Setup**—Contains the following options:
  - **Resample**—Resamples the entire signal or subset you select.
  - **Optim. for single record**—Optimizes the resampling operation for a single record. If you remove the checkmark from this checkbox, the resampling process assumes that the signals are continuous until you reset the signals.
  - **Open interval**—Specifies if the subset is an open or closed interval. For example, if an input waveform contains 3 data elements at t={0, dt, 2dt}, an open interval defines the waveform as extending over the time interval $0 \le t < 2dt$, and a closed interval defines the waveform as extending over the time interval $0 \le t < 3dt$. Place a checkmark in this checkbox to specify an open interval.
  - **Interpolation type**—Contains the following options:
    - **Coerce**—Sets each output sample value to equal the input sample value that is closest to it in time.
    - **Linear**—Sets each output sample value to be a linear interpolation between the two input samples that are closest to it in time.
    - **Spline**—Uses the spline interpolation algorithm to

	compute the resampled values. • **Filter based**—Uses an interpolation method based on the convolution of the signal with a finite impulse response (FIR) filter. – **dt**—Specifies the time resolution of the resampled time-domain signal. The default is 0.
Filter Setup	Available when you select **Filter based** from **Interpolation type**. Contains the following option: • **FIR Filter Specification**—Contains the following options: – **Normalized bandwidth**—The normalized cut-off frequency of the FIR filter to use. The default is 0.4000. – **Alias rejection (dB)**—The minimum stopband attenuation of the FIR filter to use. The default is 80 dB.

Block Diagram Inputs

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).

Time Averaging Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Performs time averaging on a time signal or scalar input.

Dialog Box Options

Block Diagram Inputs

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Autoscale amplitude	Autoscales the preview graph along the y-axis. The default is to autoscale the amplitude.
Averaged Signal	Displays the averaged time signal.
Autoscale amplitude	**Autoscale amplitude**—Autoscales the preview graph along the y-axis. The default is to autoscale the amplitude.
Configuration	Contains the following options: • **Input signal**—Specifies the input signal for the step. • **Averaging Configuration**—Contains the following options: – **Weighting mode**—Specifies a linear or exponential weighting mode for the time averaging operation. The default is Exponential. – **Number of avg.**—Specifies the number of averages to use for time averaging. – **Averaging mode**—Sets the averaging mode. Contains the following options: • **Running avg.**—Specifies to calculate a running average of all input signals for the duration of

	the operation. • **Block avg. (auto-restart)**—Restarts the averaging process as soon as Avg. count. reaches the value you specify in Number of avg. • **Status**—Contains the following options: – **Data ready**—Indicates when the averaging process is done and the averaged data are ready. – **Avg. counter**—Displays the averaging progress.
Advanced Measurement Setup	Contains the following option: • **Only return data when ready**—Specifies to only return an averaged signal when LabVIEW SignalExpress has processed the number of signals specified in the **Number of avg.** field.

Block Diagram Inputs

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).

Window Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Applies the window you select to the time-domain signal.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input and Output Signals	Displays the input signal and the windowed signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Input signal	Specifies the analog waveform for the step.
Window Setup	Contains the following option: • **Window**—Specifies the window to apply to the input signal. Options include None, Hanning, Hamming, Blackman-Harris, Exact Blackman, Blackman, Flat Top, 4 Term B-Harris, 7 Term B-Harris, Low Sidelobe, and Gaussian. The default is None.
Window Information	Contains the following results: • **Coherent gain**—Indicates the coherent gain of the window you select. • **Eq. noise bandwidth**—Indicates the equivalent noise bandwidth of the window you select.

Convert Digital to Analog Express VI

Owning Palette: Processing Express VIs

Installed With: LabVIEW SignalExpress

Converts an input digital waveform into an analog waveform.

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Digital waveform preview	Displays the input digital waveform.
Digital preview Y scrollbar	Scrollbar that allows you to scroll through the **Digital waveform preview**.
Signal name preview	Displays the signal names associated with the signals in the **Digital waveform preview**.
Analog signal preview	Displays a preview of the analog waveform.
X scale slider	Selects the data displayed in the **Digital waveform preview**.
Input	Contains the following options: • **Input signal**—Specifies the digital waveform to convert to an analog waveform.
Configuration and Results	Contains the following options: • **Analog full scale**—The total peak-to-peak range, or the difference between the minimum and maximum, for the analog waveform. For example, if the maximum range of a waveform is 1 and the minimum is -1, the full-scale range for the waveform is 2. • **Digital data format**—Specifies which binary representation you want to use for the digital data. – **Unsigned binary**—The data is converted to unsigned binary. – **Offset binary**—The largest negative value (negative full-scale) is represented by all zeros, and the largest positive value (positive full-scale) is represented by all ones. Zero-scale is represented by

	a one (MSB) followed by all zeros, for example, binary 1000. – **2's complement**—Uses two's complement format, which is a common format for representing signed binary values. This format is similar to Offset Binary, but the MSB is inverted. • **Output signal unit**—Specifies the unit type for the output signal. Selecting **Unitless** specifies to not associate any unit with the output signal. Selecting **Custom** specifies that the output signal contains an associated unit within the data. • **Resolution (bits)**—Returns the number of bits in the converted digital waveform. • **Export resolution**—Exports the resolution to the Project View. • **Y axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated unit within the data. • **X axis unit**—Specifies the unit type for the input signal. Selecting **Unitless** specifies to not associate any unit with the input signal. Selecting **Custom** specifies that the input signal contains an associated unit within the data.

Block Diagram Inputs

Parameter	Description
error in (no error)	Describes error conditions that occur before this VI or function runs.

Block Diagram Outputs

Parameter	Description
error out	Contains error information. If **error in** indicates that an error occurred before this VI or function ran, **error out** contains the same error information. Otherwise, it describes the error status that this VI or function produces. Right-click the **error out** front panel indicator and select **Explain Error** from the shortcut menu for more information about the error.

Analysis Express VIs

Owning Palette: LabVIEW SignalExpress™ Express VIs

Use the Analog Express VIs to process, test, import and export, and perform frequency-domain and time-domain measurements on analog signals.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Subpalette	Description
Frequency-Domain Measurements Express VIs	Use the Frequency-Domain Measurements Express VIs to perform signal analysis that require the data to be converted into frequency domain. The Frequency-Domain Measurements VIs perform frequency domain transformations and frequency domain analysis.
Test and Compare Express VIs	Use the Test and Compare Express VIs to compare an input signal to user-specified limits.
Time-Domain Measurements Express VIs	Use the Time-Domain Measurements Express VIs to perform time domain analysis. The Time-Domain Measurements VIs implement some operations commonly used in signal processing.

Frequency-Domain Measurements Express VIs

Owning Palette: Analysis Express VIs

Use the Frequency-Domain Measurements Express VIs to perform signal analysis that require the data to be converted into frequency domain. The Frequency-Domain Measurements VIs perform frequency domain transformations and frequency domain analysis.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Palette Object	Description
Distortion	Performs harmonic distortion analysis and/or SINAD measurement on the input signal.
Frequency Response	Computes the frequency response and the coherence based on the stimulus and response input signals and returns the results as magnitude, phase, and coherence.
Power Spectrum	Computes the averaged magnitude spectrum, power spectrum, or power spectral density for a single or multiple channels. This step can return the spectra in root-mean-square, peak, and peak-to-peak units.
Tone Extraction	Finds the single tone with the highest amplitude or searches a frequency range you specify to find the single tone with the highest amplitude and returns the frequency, amplitude, and absolute phase for the detected tone. The step also can export **Export signals** to the Project View.

Distortion Express VI

Owning Palette: Frequency-Domain Measurements Express VIs

Installed With: LabVIEW SignalExpress

Performs harmonic distortion analysis and/or SINAD measurement on the input signal.

This step returns the fundamental frequency, the percentage of total harmonic distortion, the total harmonic distortion plus noise value, and the SINAD value in decibels.

This step also returns a time-domain waveform and frequency-domain power spectrum for the different components of the signal, such as fundamental signal, residual signal, or harmonics.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Exported Signal	Displays the time signal you selected with **Export signals** in the **Measurement Setup** section on the **Configuration** page. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Exported Power Spectrum	Displays the power spectrum of the signal the **Exported Signal** graph displays.
Autoscale magnitude	Scales the magnitude of the exported power spectrum. The default is to autoscale the magnitude.
Input/Output	Contains the following options: • **Input signal**—Specifies the input signal to be measured. • **Export time signal**—Exports the signal to the Project View as **Export Signals (THD)** or **Export Signals (SINAD)** specifies. • **Export power spectrum**—Exports the power spectrum to the Project View as **Export Signals (THD)** or **Export Signals (SINAD)** specifies.
Configuration	Contains the following options: • **Measurement Setup**—Contains the following options: – **Distortion types**—Specifies **Harmonic Only**, **SINAD Only**, or **Harmonic and SINAD** distortion. – **Highest harm.**—Specifies the highest

harmonic to include in the calculation of the harmonic distortion. The default is 19, so the THD result is based on harmonics 2 to 19 (both included) assuming that the **Exclude aliased harmonics** checkbox does not include a checkmark. If you place a checkmark in the **Exclude aliased harmonics** checkbox, **Highest harm.** includes only the harmonics below the Nyquist frequency. The Nyquist frequency is half the sample rate of the input signal.

- **Export signals (THD)**—Specifies which signals to display on the two graphs and to export to the Project View. Options include Input Signal, Fundamental Tone, Residual Signal, Harmonics Only, Noise and Spurs or None (no signal). The default is Input Signal.
- **Export signals (SINAD)**—Specifies which signals to display on the two graphs and to export to the Project View. Options include Input Signal, Fundamental Tone, Residual Signal, or None (no signal). The default is Input Signal.

- **Measurement Results**—Contains the following options:
  - **Fund. Frequency**—Returns the detected fundamental frequency of the input signal.
  - **THD (%)**—Returns the measured percentage of total harmonic distortion up to and including the highest harmonic or limited by the Nyquist frequency.
  - **SINAD (dB)**—Returns the measured sine in noise and distortion value in decibels.

	– **THD + Noise (%)**—Returns the measured total harmonic distortion plus noise. Notice that this result always includes all harmonics and is therefore independent of the value you specify for **Highest harm**.
Advanced	Contains the following options: • **Approx. fund. freq. (Hz)**—Returns the center frequency to use in the frequency-domain search for the fundamental tone. A negative value specifies to search for the tone with the highest amplitude. The default is–1. • **Exclude aliased harmonics**—Exclude aliased harmonics should be set to TRUE (default) to include only frequencies less than the Nyquist frequency, or half the sampling rate, in the harmonic search. When set to FALSE, this step continues searching the frequency domain beyond Nyquist by assuming that higher frequency components have aliased according to the following equation. Aliased f = Fs – (f modulo Fs) where Fs = 1/dt = sampling rate.

Power Spectrum Express VI

Owning Palette: Frequency-Domain Measurements Express VIs

Installed With: LabVIEW SignalExpress

Computes the averaged magnitude spectrum, power spectrum, or power spectral density for a single or multiple channels. This step can return the spectra in root-mean-square, peak, and peak-to-peak units.

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Graph	**Spectra**—Displays the spectra for all channels. Use the **Zoom** button to zoom in and out of the display. **Signals**—Displays the time-domain signals for all channels. Use the **Zoom** button to zoom in and out of the display.
View	Specifies if the graph displays time domain signals or the computed power spectra.
Autoscale	Automatically adjusts the scales of the graph to display the data.
Input	Contains the following option: • **Input signal**—Specifies the analog waveform for the step.
Configuration	Contains the following options: • **Window**—Specifies the window to apply to the input signal. Choose from one of the following window options: – **None** – **Hanning** – **Hamming** – **Blackman-Harris** – **Exact Blackman** – **Blackman** – **Flat Top** – **4 Term B-Harris** (Four Term Blackman-Harris) – **7 Term B-Harris** (Seven Term Blackman-Harris) – **Low Sidelobe** – **Gaussian** • **Scaling**—Specifies the scaling parameter for the

	step. – **Spectrum type**—Specifies if the spectrum is in units of **Magnitude** or **Power**, where power equals magnitude squared. The default is **Power**. – **Magnitude scale**—Specifies if zoom power is in linear units or in decibels. The default is decibels. – **Peak conversion**—Specifies the peak scaling of the converted spectrum. You can select **RMS** (default), **Peak**, or **Peak to Peak**. – **Spectral density**—Specifies if the spectrum is returned as power spectral density (PSD). The default is **Off**.
Averaging	Specifies the averaging parameters. • **Averaging mode**—Specifies the averaging mode from the following options: – **No Averaging** (Default) – **Vector Averaging** – **RMS Averaging** – **Peak Hold** • **Weighting mode**—Specifies either **Exponential** or **Linear** weighting. Exponential averaging applies more weight to the most recent data, and linear averaging applies equal weighting to all the data. • **Number of averages**—Specifies the number of averages used by the selected **Weighting mode**. • **Auto-restart**—Specifies if the averaging process automatically restarts once the step reaches the **Number of averages** value. When you set **Weighting mode** to linear, use **Auto-restart** to configure averaging to automatically restart when **Averages completed** equals the **Number**

	of averages. • **Averages completed**—Displays the number of averages completed. • **Averaging done**—Indicates when the number of **Averages completed** equals or exceeds the **Number of averages**. **Averaging done** is always TRUE if the selected **Averaging** mode is **No averaging**.

Block Diagram Inputs

Parameter	Description
input signals	Specifies the scaled input signal.
error in (no error)	Describes error conditions that occur before this VI or function runs.
restart averaging	Specifies whether to restart the selected averaging process. The default is FALSE. When you call this Express VI for the first time, the averaging process restarts automatically.

Block Diagram Outputs

Parameter	Description
spectrum	Returns the power spectrum computed by the Express VI.
spectrum info	Returns spectrum information. This information is needed by the Peak Search Express VI and the Power in Band Express VI.
unit labels	Returns the unit labels.
averages completed	Returns the number of averages completed by the Express VI.
averaging done	Returns TRUE when the number of averages completed equals or exceeds the number of averages specified in averaging parameters. Otherwise, averaging done returns FALSE. **averaging done** is always TRUE if the selected averaging mode is No averaging.
error out	Contains error information. If **error in** indicates that an error occurred before this VI or function ran, **error out** contains the same error information. Otherwise, it describes the error status that this VI or function produces. Right-click the **error out** front panel indicator and select **Explain Error** from the shortcut menu for more information about the error.

Tone Extraction Express VI

Owning Palette: Frequency-Domain Measurements Express VIs

Installed With: LabVIEW SignalExpress

Finds the single tone with the highest amplitude or searches a frequency range you specify to find the single tone with the highest amplitude and returns the frequency, amplitude, and absolute phase for the detected tone. The step also can export **Export signals** to the Project View.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Exported Time Signal	Displays the time signal that **Export signals** specifies. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Exported Power Spectrum	Displays the power spectrum in decibels that **Exported Time Signal** specifies.
Autoscale magnitude	Scales the magnitude scale of the power spectrum result graph. The default is to autoscale the magnitude.
Input/Output	Contains the following options: • **Input signal**—Specifies the signal from which to extract tonal data. • **Export time signal**—Exports the time signal that **Exported Time Signal** displays to the Project View. • **Export power spectrum**—Exports the power spectrum result that **Exported Power Spectrum** displays to the Project View.
Configuration	Contains the following option: • **Measurement Setup and Results**—Contains the following options: – **Export signals**—Specifies the signal to display on the graphs and optionally export to the Project View. Options include None, Input Signal, Extracted Tone, or Residual Signal, which is the input signal minus the extracted single

	tone. The default is Input Signal. – **Detected frequency**—Returns the frequency of the detected single tone in hertz. – **Detected amplitude**—Returns the amplitude of the detected single tone. – **Detected phase (deg)**—Returns the phase of the detected single tone in degrees.
Advanced	Contains the following option: • **Advanced Measurement Setup**—Contains the following options: – **Approximate frequency (Hz)**—Specifies the center frequency to use in the frequency-domain search for the single tone. A negative value corresponds to search automatically for the tone with the highest amplitude. The default is –1. – **Search range (% of sample rate)**—Specifies the frequency span as a percentage of the sampling rate for the frequency-domain search for the single tone frequency. The default is 0.25%.

Test and Compare Express VIs

Owning Palette: Analysis Express VIs

Use the Test and Compare Express VIs to compare an input signal to user-specified limits.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Palette Object	Description
Digital Compare	Compares a reference and test signal to determine the number of sample errors.
Limit Test	Tests an input signal or value against user-specified limits and returns information on whether the test passed or failed and, in the case of a failure, where it failed. Limit Test accepts time-domain signals, frequency-domain signals, and scalar values as inputs. You can specify either signals or scalar values for the limits, and you can define the limits or use other signals in the project as the limits.

Limit Test Express VI

Owning Palette: Test and Compare Express VIs

Installed With: LabVIEW SignalExpress

Tests an input signal or value against user-specified limits and returns information on whether the test passed or failed and, in the case of a failure, where it failed. Limit Test accepts time-domain signals, frequency-domain signals, and scalar values as inputs. You can specify either signals or scalar values for the limits, and you can define the limits or use other signals in the project as the limits.

Dialog Box Options

Parameter	Description
View	Specifies how to display the results of the limit test. You can select from the following options: • **Graph**—(Default) Displays the results of the limit test as a graph. • **Results table**—Displays the results of the limit test as a table.
Limit Test	[View: Graph] Displays the result of the limit test operation. The four plots show the original input signal, the points where the limit test operation failed, and the two limit signals, respectively. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Limit test results	[View: Results table] Displays each signal in the limit test and whether or not the signal passed the test.
Autoscale y-axis	[View: Graph] Specifies whether to autoscale the y-axis on the **Limit Test** graph.
selected test	Indicates whether the test of the signal displayed on the **Limit Test** graph passed or failed. This indicator appears only when you test a group of signals.
all tests	Indicates if the tests passed or failed.
Input	The following options apply only to the Limit Test step in LabVIEW SignalExpress: • **Input signal**—Specifies the input signal or scalar value. • **Upper limit**—[Limits source: Input Signals, Input Scalars] Specifies the upper limit signal or value.

	Note When a LabVIEW SignalExpress project runs continuously, LabVIEW SignalExpress reads the **Upper limit** and **Lower limit** only on the first iteration of the project. If you select a limit signal that continuously changes, LabVIEW SignalExpress uses only the first iteration of the signal to perform the limit test. • **Lower limit**—[Limits source: Input Signals, Input Scalars] Specifies the lower limit signal or value. • **Limit**—[Limits source: Input Signals, Input Scalars AND Limits window based on: Single Limit & Range] Specifies the single limit signal or value.
Input type	The following options apply only to the Limit Test Express VI in LabVIEW: • **Time Waveform**—Specifies to perform limit testing on a time domain signal. • **Frequency Waveform**—Specifies to perform limit testing on a frequency spectrum. • **Scalar**—Specifies to perform limit testing on a single scalar value.
Configuration	Contains the following options for configuring the limits for the limit test: • **Limit Setup**—Contains the following options: – **Limits source**—Specifies the source of the limits for the limit test. You can select from the following options: • **Input Signals**—Uses output signals from previous steps or Express VIs as the limits. • **Input Scalars**—Uses output scalar values from previous steps or Express VIs as the limits. You specify the scalar values on the **Input** page.

- **User Defined Signals**—Uses signals you define for the limits. If you set **Limits window based on** to **Two Limits**, click the **Define upper limit** and **Define lower limit** buttons to display the **Define Signal** dialog box and interactively define the limit signals. If you set **Limits window based on** to **Single Limit & Range**, click the **Define single limit** button to display the **Define Signal** dialog box and interactively define the limit signal.
- **User Defined Constants**—(Default) Uses constant values that you specify for the limits. If you set **Limits window based on** to **Two Limits**, use the **Upper constant** and **Lower constant** fields to specify the limits. If you set **Limits window based on** to **Single Limit & Range**, use the **Limit constant** field to specify the limit.

– **Compare mode**—Specifies the comparison mode Limit Test uses to compare the input signal to the limits you specify. You can select from the following options:

- **Between Limits**—(Default) Determines whether the input signal is between the limits you specify.
- **Outside Limits**—Determines whether the input signal is outside the limits you specify.

- **>Lower Limit**—Determines whether the input signal is above the lower limit you specify.
- **<Upper Limit**—Determines whether the input signal is below the upper limit you specify.

- **Limits window based on**—Specifies how you define the limits used for the limit test. You can select from the following options:
  - **Two Limits**—Specifies that you define two limits for the limit test.
  - **Single Limit & Range**—Specifies that you define a single limit and a range of gain and offset values for the limit test.
- **Upper constant**—[Limits source: User Defined Constants AND Limits window based on: Two Limits] Specifies the value of the upper limit constant. The default is 1.
- **Lower constant**—[Limits source: User Defined Constants AND Limits window based on: Two Limits] Specifies the value of the lower limit constant. The default is –1.
- **Limit constant**—[Limits source: User Defined Constants AND Limits window based on: Single Limit & Range] Specifies the single constant value that, in conjunction with the **Relative Range Specs**, defines the limit values.
- **Define upper limit**—[Limits source: User Defined Signals AND Limits window based on: Two Limits] Displays the **Define Signal** dialog box, which you can use to define the upper limit signal

	interactively. – **Define lower limit**—[Limits source: User Defined Signals AND Limits window base on: Two Limits] Displays the **Define Signal** dialog box, which you can use to define the lower limit signal interactively. – **Define single limit**—[Limits source: User Defined Signals AND Limits window base on: Single Limit & Range] Displays the **Define Signal** dialog box, which you can use to define the limit signal interactively. • **Relative Range Specs**—Contains the following options: – **Upper gain**—[Limits window base on: Single Limit & Range] Specifies the gain value to apply to the single limit to calculate the upper limit. The default is 1.1. – **Lower gain**—[Limits window base on: Single Limit & Range] Specifies the gain value to apply to the single limit to calculate the lower limit. The default is 900m. – **Upper offset**—[Limits window base on: Single Limit & Range] Specifies the offset value to add to the single limit to calculate the upper limit. The default is 0. – **Lower offset**—[Limits window base on: Single Limit & Range] Specifies the offset value to add to the single limit to calculate the lower limit. The default is 0.
Advanced	Contains the following options: • **Limits Inclusion**—Contains the following options: – **Upper inclusive**—Specifies whether a value that is exactly on or equal to the

	upper limit passes the limit test. Place a checkmark in this checkbox to pass a value that is on or equal to the upper limit. – **Lower inclusive**—Specifies whether a value that is exactly on or equal to the lower limit passes the limit test. Place a checkmark in this checkbox to pass a value that is on or equal to the lower limit. • **Timing information**—Contains the following options for time-domain and frequency-domain signals: – **Freq. axis is logarithmic**—Sets the display graph frequency axis to logarithmic and, when the **Limits source** is **User Defined Signals**, computes the limit values between the definition points so the resulting segment appears as a straight line in a logarithmic frequency representation. For example, you can use this to create asymptotic limits fitting filter roll-off in decibels per decade. This parameter appears only if the input signal is a frequency-domain signal. – **Ignore timestamp**—Forces the timestamp of the input signal to 0 so you can define the limit signals relative to the beginning of the input signal. This parameter appears only if the input signal is a time-domain signal.
Actions	Contains the following options: • **Action on failed**—Contains the following options: – **Stop project after failed occurs**—Specifies to stop running the project after

the signal fails the limit test the number of **times** you specify.

- **times**—Specifies the number of times for the signal to pass or fail the limit test before LabVIEW SignalExpress stops the project.
- **Action on failed**—Specifies the action to perform when the signal fails the limit test the number of **times** you specify.
    - **none**—Specifies to take no additional action.
    - **snapshot of inputs**—Specifies to take a snapshot of the current inputs of the Limit Test step.
    - **snapshot of all signals in project**—Specifies to take a snapshot of all signals in the project.

- **Action on passed**—Contains the following options:
    - **Stop project after passed occurs**—Specifies to stop running the project after the signal passes the limit test the number of **times** you specify.
    - **times**—Specifies the number of times for the signal to pass or fail the limit test before LabVIEW SignalExpress stops the project.
    - **Action on passed**—Specifies the action to perform when the signal passes the limit test the number of **times** you specify.
        - **none**—Specifies to take no additional action.
        - **snapshot of inputs**—Specifies to take a snapshot of the current inputs of the Limit Test step.

	• **snapshot of all signals in project**—Specifies to take a snapshot of all signals in the project.

Block Diagram Inputs

Parameter	Description
Input	Specifies the input signal or scalar value.
error in (no error)	Describes error conditions that occur before this VI or function runs.
Lower	Specifies the frequency spectrum that the Express VI uses as the lower limit.
LowerScalar	Specifies the scalar value that this Express VI uses as the lower limit.
reset	Specifies if the limits have to be reset. The default is FALSE. Set **reset** to TRUE each time you provide new limits to this VI.
Upper	Specifies the frequency spectrum that this Express VI uses as the upper limit.
UpperScalar	Specifies the scalar value that this Express VI uses as the upper limit.

Block Diagram Outputs

Parameter	Description
test passed	Indicates the result of limit mask testing.
error out	Contains error information. If **error in** indicates that an error occurred before this VI or function ran, **error out** contains the same error information. Otherwise, it describes the error status that this VI or function produces. Right-click the **error out** front panel indicator and select **Explain Error** from the shortcut menu for more information about the error.
Input Signals	Returns the input signal, the upper and lower limits, and the failures.

Limit Test Details

Output Signal Types

Limit Test returns a group of signals named **limit test results**. This group of signals contains the following elements:

- **failed signal**—The input signal(s) or value(s). If you graph **failed signal**, the graph includes the input signal, the limits, and the portions of the signal that failed the limit test.
- **upper limit**—The upper limit used to perform the limit test.
- **lower limit**—The lower limit used to perform the limit test.

Limits

The type of limits you can define for the Limit Test depend on the input signal type.

If the input is a time-domain or frequency-domain signal, you can use the following types of limits:

- Output signals of the same type as the input signal—You can use output signals from other LabVIEW SignalExpress steps or LabVIEW VIs as the limits. For example, you can compare a time-domain signal to two other time-domain signals.
- Output scalar values—You can use output scalar values from other LabVIEW SignalExpress steps or LabVIEW VIs as the limits. Limit Test compares the input signal to the scalar values element by element. For example, you can compare a time-domain signal to two measured DC values.
- User-defined signals of the same type as the input signal—You can define limit signals by clicking the **Define Upper Limit**, **Define Lower Limit**, or **Define Single Limit** buttons. The **Define Signal** dialog box appears, and you can define a limit signal based on user-defined points. Use this dialog box to create a limit signal made up of a series of line segments that connect these points.
- User-defined constants—You can define constant values for the limits. Limit Test compares the input signal to the constant values element by element.

If the input is a scalar value, you can use the following types of limits:

- Output scalar values—You can use output scalar values from other LabVIEW SignalExpress steps or LabVIEW VIs as the limits. Limit Test compares the input signal to the scalar values element by element. For example, you can compare a time-domain signal to two measured DC values.
- User-defined constants—You can define constant values for the limits. Limit Test compares the input value to the constant values.

Compare Mode and Limits Inclusion

Limit Test has four compare modes. These modes indicate if a signal or value is between limits, outside limits, greater than a lower limit, or lower than an upper limit. You can choose the exact limit values to include or not include in the test. These limit values result in a failing or passing test where the input value equals the limit value.

Defining a Limit Range from a Single Limit

You can define a set of upper and lower limits from a single limit using the gain and offset scaling parameters. Select the limits based on **Single Limit & Range** to enable the user-defined scaling parameters.

Limits Defined in a Logarithmic Frequency Scale

When you use user-defined signals as limits for a frequency-domain signal, Limit Test defines the limits as a series of line segments that connect user-defined points. By default, Limit Test assumes the frequency axis is linear so a linear relationship exists between the frequency and the magnitude or phase values. The **Freq. axis is logarithmic** checkbox specifies whether to display a logarithmic frequency axis and define the limit signals so the connection between the points appears as straight lines in the logarithmic frequency scale. You can define the limits in a logarithmic frequency scale if you want to test the asymptotic roll-off of a filter, typically a straight line in a decibel versus logarithmic frequency scale.

Digital Compare Express VI

Owning Palette: Test and Compare Express VIs

Installed With: LabVIEW SignalExpress

Compares a reference and test signal to determine the number of sample errors.

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Digital waveform preview	Displays the two waveforms to be compared.
Signal name preview	Displays the signal names associated with the signals in the **Digital waveform preview**.
Digital preview Y scrollbar	Scrollbar that allows you to scroll through the **Digital waveform preview**.
Input	Contains the following options: • **Reference waveform**—Displays the signal that will serve as the reference for your comparison. Sample errors are produced when the test waveform differs from this waveform. • **Test waveform**—Displays the signal you want to compare to the reference signal. Sample errors are produced when the test waveform differs from the reference waveform.
Configuration	Contains the following options: • **Comparison Start Position**—Contains the following options: – **Reference waveform start**—Specifies the position of the sample where you want to start the signal comparison. – **Test waveform start**—Specifies the position of the sample where you want to start the signal comparison. • **Compare subset**—Specifies whether you want to compare the entire test signal to the reference signal or whether you only want to compare a subset of the samples. Selecting this checkbox means that a subset that is the number of samples specified in **Number Samples** will be compared, starting at the reference waveform

	start position you specify. • **Number of samples**—Specifies the number of samples in the subset the VI compares. • **Passed**—Displays the result of the waveform compare. If all specified samples match, the compare passes. Otherwise, the compare fails, and the number of sample errors is returned in **Number of Sample Errors**. • **Number of Sample Errors**—Returns the number of sample errors in the test signal.

Block Diagram Inputs

Parameter	Description
error in (no error)	Describes error conditions that occur before this VI or function runs.

Block Diagram Outputs

Parameter	Description
error out	Contains error information. If **error in** indicates that an error occurred before this VI or function ran, **error out** contains the same error information. Otherwise, it describes the error status that this VI or function produces. Right-click the **error out** front panel indicator and select **Explain Error** from the shortcut menu for more information about the error.

Time-Domain Measurements Express VIs

Owning Palette: Analysis Express VIs

Use the Time-Domain Measurements Express VIs to perform time domain analysis. The Time-Domain Measurements VIs implement some operations commonly used in signal processing.

If a LabVIEW SignalExpress palette is empty or appears to be missing VIs, you are missing components that are not installed by LabVIEW SignalExpress. Reinstall the National Instruments Device Drivers DVD and include NI-DAQmx and Modular Instruments support.

The VIs on this palette can return general LabVIEW error codes.

Palette Object	Description
Amplitude and Levels	Measures DC, RMS, positive and negative peak, and peak-to-peak values of a signal.
Histogram	Calculates the discrete histogram of the input signal. The **Histogram** result runs continuously and accumulates the data from all incoming signals until you click the **Reset Histogram** button on the title bar of the **Histogram** configuration view or you change a configuration parameter. You can compute the resulting bin values as an absolute number of occurrence or as a percentage of the total number of occurrences. You also can display the accumulated bin values in a logarithmic scale. Click the **Auto-config** button below the **Histogram** graph to display standard start-up configuration options.
Statistics	Performs statistical calculations on time-domain, scalar, or array of scalar data. You can select up to six statistical measurements to perform on your data, and Statistics creates a scalar output for each specified measurement. If the input signal is a waveform, by default Statistics returns a statistical measurement on the current input signal. For scalar data, the Statistics step returns a statistical measurement that represents the entire signal history since you started the project or you reset the step.

Timing and Transition	Measures timing and transition parameters on single pulses and on rising and falling edges.

Amplitude and Levels Express VI

Owning Palette: Time-Domain Measurements Express VIs

Installed With: LabVIEW SignalExpress

Measures DC, RMS, positive and negative peak, and peak-to-peak values of a signal.

You can measure DC and RMS values with linear or exponential averaging. If you select **Linear**, you can apply a window to the signal.

You also can individually export the different measurement results.

Dialog Box Options

Block Diagram Inputs

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Autoscale amplitude	Autoscales the preview graph along the y-axis. The default is to autoscale the amplitude.
Input/Output	Contains the following options: • **Input signal**—Specifies the input signal to be measured. • **Export DC value**—Exports the DC value to the Project View. • **Export RMS value**—Exports the RMS value to the Project View. • **Export +Peak value**—Exports the positive peak value to the Project View. • **Export -Peak value**—Exports the negative peak value to the Project View. • **Export Peak-Peak value**—Exports the difference between the positive and negative peak values to the Project View.
Configuration	Contains the following options: • **DC-RMS Setup**—Contains the following options: – **Averaging type**—Sets the type of averaging to linear or exponential. The default is Linear. – **Window**—Sets the type of window when you use linear averaging. Windowing can

	sometimes help increase measurement accuracy for signals that are dominated by periodic components **Window** is not available if you select Exponential as the averaging type. **Window** contains the following options: Rectangular (none), Hanning, and Low side lobe. The default is Rectangular (none). • **Peak Setup**—Contains the following option: – **Hold peaks**—Specifies to hold the peak levels until you click the **Reset Amplitude and Levels** button or the **Reset All** button or restart your measurement. The default is to not hold the peak levels. • **DC-RMS Results**—Contains the following options: – **DC value**—Returns the measured DC value. – **RMS value**—Returns the measured RMS value. • **Peak Results**—Contains the following options: – **+Peak value**—Returns the positive peak value of the input signal. – **-Peak value**—Returns the negative peak value of the input signal. – **Peak-peak value**—Returns the difference between the positive and negative peak values of the input signal.

Block Diagram Inputs

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).

Histogram Express VI

Owning Palette: Time-Domain Measurements Express VIs

Installed With: LabVIEW SignalExpress

Calculates the discrete histogram of the input signal. The **Histogram** result runs continuously and accumulates the data from all incoming signals until you click the **Reset Histogram** button on the title bar of the **Histogram** configuration view or you change a configuration parameter. You can compute the resulting bin values as an absolute number of occurrence or as a percentage of the total number of occurrences. You also can display the accumulated bin values in a logarithmic scale. Click the **Auto-config** button below the **Histogram** graph to display standard start-up configuration options.

Dialog Box Options

Block Diagram Inputs

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Histogram	Displays the histogram of the input signal.
Auto-config.	Sets the values in **Number of bins**, **Minimum value**, and **Maximum value** based on the input signal.
Log. bin values	Scales the bin values axis to logarithmic scale. The default is to not scale the bin values axis to logarithmic scale.
Input	Contains the following option: • **Input signal**—Specifies the analog waveform for the step.
Configuration	Contains the following option: • **Histogram Specifications**—Contains the following options: – **Number of bins**—Specifies the number of bins. The default is 20. – **Minimum value**—Specifies the minimum value. The default is –1. – **Maximum value**—Specifies the maximum value. The default is 1. – **Bin value in percent**—Configures the histogram result to scale by percent. The default is to scale the result by percent. – **Calculation enabled**—Enables calculation of the histogram. The default

	is to enable the calculation.

Block Diagram Inputs

Parameter	Description
Input Signal	Displays the input signal. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).

Statistics Express VI

Owning Palette: Time-Domain Measurements Express VIs

Installed With: LabVIEW SignalExpress

Performs statistical calculations on time-domain, scalar, or array of scalar data. You can select up to six statistical measurements to perform on your data, and Statistics creates a scalar output for each specified measurement. If the input signal is a waveform, by default Statistics returns a statistical measurement on the current input signal. For scalar data, the Statistics step returns a statistical measurement that represents the entire signal history since you started the project or you reset the step.

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input Signal	Displays the input signal. If you wire data to the Express VI and run it, **Input Signal** displays real data. If you close and reopen the Express VI, **Input Signal** displays sample data until you run the Express VI again. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s). • **Number of points in the data**—Specifies the number of points in the data. The default is 4400.
Input	Contains the following option: • **Input signal**—Specifies the input value.
Configuration And Results	Contains the following options: • **Max**—Specifies to output the maximum value of the current input signal. • **Min**—Specifies to output the minimum value of the current input signal. • **Mean**—Specifies to output the mean value of the current input signal. • **Nb of samples**—Specifies to output the number of input samples the Statistics step performs statistical measurements on. • **Combine channels**—Specifies whether Statistics returns single outputs for measurements on groups of input signals. For example, if you configure Statistics to measure the maximum value of the signals, placing a checkmark in this checkbox returns the maximum value of all the samples of all the

	channels, instead of one maximum value per channel. This option appears only when **Input signal** is a group of signals. • **Standard deviation**—Specifies to output the standard deviation of the current input signal. • **Variance**—Specifies to output the variance of the current input signal. • **Sum**—Specifies to output the sum of the current input signal. • **Measurement duration (s)**—Indicates the duration of each measurement the Statistics step is returning. • **Restart measurement on each iteration**—Specifies to restart the statistical measurement on each iteration of the input signal. If you do not select this option, output statistics represent the cumulative values of the entire input signal.

Timing and Transition Express VI

Owning Palette: Time-Domain Measurements Express VIs

Installed With: LabVIEW SignalExpress

Measures timing and transition parameters on single pulses and on rising and falling edges.

If the signal includes a single positive or negative pulse or a pulse train, you can measure the pulse frequency, period (1/frequency), duration, and duty cycle. You also can use this step to measure the transition time, rise or fall time, the amount of undershoot and overshoot, and the slew rate on rising and falling edges. You can select the pulse polarity and the pulse and/or edge numbers on the **Advanced** page.

Details

Dialog Box Options

- Place on the block diagram
- Find on the **Functions** palette

Dialog Box Options

Parameter	Description
Input Signal	Displays the input signal to measure. Cross-hair cursors indicate the pulse and edges to use for the measurements. This step marks edges at the pulse and mid-transition points at its center position using the color code in the three result tables. • **Displayed signal**—Specifies the signal(s) to display in the preview graph(s). This option appears only when you select a group of signals for the input.  **Note** If the input signals include scalar values that depend on the values of input waveform signals, you cannot specify to display all signals in the preview graph(s).
Autoscale amplitude	Scales the Amplitude (V) axis of the **Acquired Data** graph. The default is to autoscale the amplitude.
Input	Contains the following option: • **Input signal**—Specifies the analog waveform for the step.
Configuration and Results	Contains the following options: • **Pulse results**—Contains the results of the pulse measurements. If one or more measurements you select fail, this step highlights the background color of the failing measurements in red. To disable failing measurements, remove the checkmark from the corresponding checkboxes. Contains the following options: – **Export frequency**—Measures the frequency of a pulse train and exports the measurement result to the Project View. – **Export period**—Measures the period of a pulse train and exports the measurement result to the Project View.

- **Export duration**—Measures the duration of a pulse and exports the measurement result to the Project View.
- **Export duty cycle**—Measures the duty cycle of a pulse and exports the measurement result to the Project View.

- **Rising edge results**—Contains the results of the rising edge transition measurements. If one or more measurements you select fail, this step highlights the background color of the failing measurements in red. To disable failing measurements, remove the checkmark from the corresponding checkboxes. Contains the following options:
  - **Export rise time**—Measures the rising transition time (rise time) of an edge and exports the measurement result to the Project View. The rise time is the time it takes the signal to change from a low reference level (10% of the amplitude of the signal) to a high reference level (90% of the amplitude of the signal).
  - **Export rising undershoot**—Measures the percentage amount of undershoot that precedes a rising edge and exports the measurement result to the Project View.
  - **Export rising overshoot**—Measures the percentage amount of overshoot that follows a rising edge and exports the measurement result to the Project View.
  - **Export rising slew rate**—Measures the slew rate, or the ratio between (90% amplitude – 10% amplitude) and the rise time, of a rising edge and exports the measurement result to the Project View.
- **Falling edge results**—Contains the results of

	the falling edge transition measurements. If one or more measurements you select fail, this step highlights the background color of the failing measurements in red. To disable failing measurements, remove the checkmark from the corresponding checkboxes. Contains the following options: – **Export fall time**—Measures the falling transition time (fall time) of an edge and exports the measurement result to the Project View. The fall time is the time it takes the signal to change from a high reference level (90% of the amplitude of the signal) to a low reference level (10% of the amplitude of the signal). – **Export falling undershoot**—Measures the percentage amount of undershoot that follows a falling edge and exports the measurement result to the Project View. – **Export falling overshoot**—Measures the percentage amount of overshoot that precedes a falling edge and exports the measurement result to the Project View. – **Export falling slew rate**—Measures the slew rate, or the ratio between (90% amplitude – 10% amplitude) and the fall time, of a falling edge and exports the measurement result to the Project View.
Advanced	Contains the following options: • **Pulse Definition**—Contains the following options: – **Pulse polarity**—Specifies if the pulse to measure is positive (High Pulse) polarity or negative (Low Pulse) polarity. – **Pulse number**—Specifies which pulse number in a pulse train to use for the

	measurement. • **Transition Definition**—Contains the following options: – **Rising edge number**—Specifies which rising edge to use for the measurement. – **Falling edge number**—Specifies which falling edge to use for the measurement.

Timing and Transition Details

The following image shows a sample pulse. Timing and Transition uses a high reference level of 90% of the amplitude of the signal and a low reference level of 10% of the amplitude of the signal. In a timing and transition measurement, overshoot is the height of a local maximum preceding a rising or falling edge, depending on the **Pulse polarity** you specify. Undershoot is the height of the local minimum preceding a rising or falling edge, depending on the **Pulse polarity** you specify.