

# Analyzing and Saving a Signal

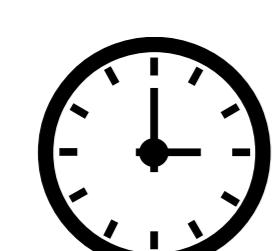
LabVIEW includes a set of Express VIs that help you analyze signals. This chapter teaches you how to use LabVIEW to perform a basic analysis of a signal and how to save the analyzed data to a file.



**Note** The exercises in this chapter use Express VIs that are available only in the LabVIEW Full and Professional Development Systems.

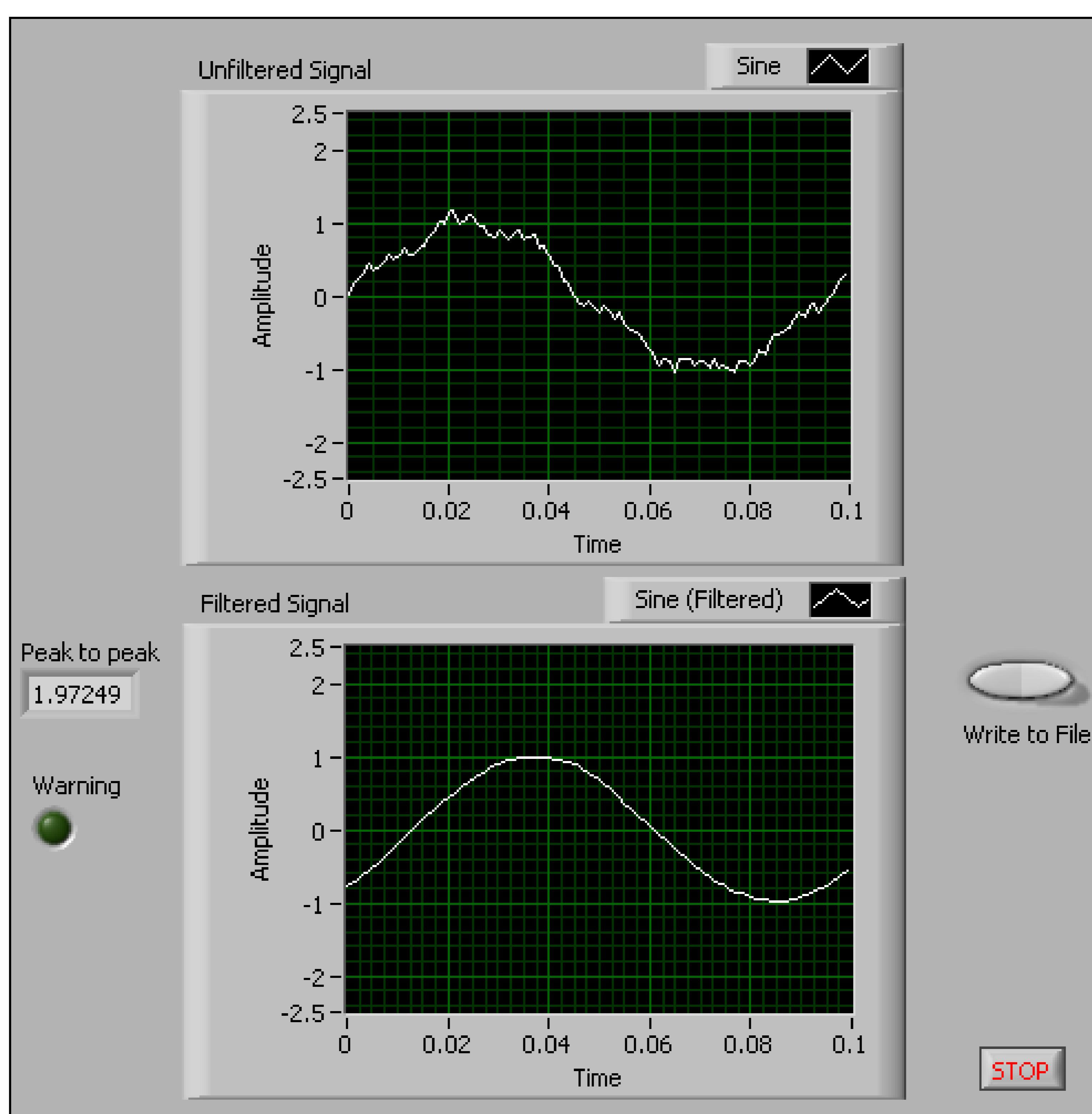
## Building an Analysis VI

In the following exercises, you will build a VI that generates a signal, filters the signal, indicates if the signal exceeds a certain limit, and records the data. After you complete the exercises, the front panel of the VI will look similar to the front panel in the following figure.



You can complete the exercises in this chapter in approximately 40 minutes.

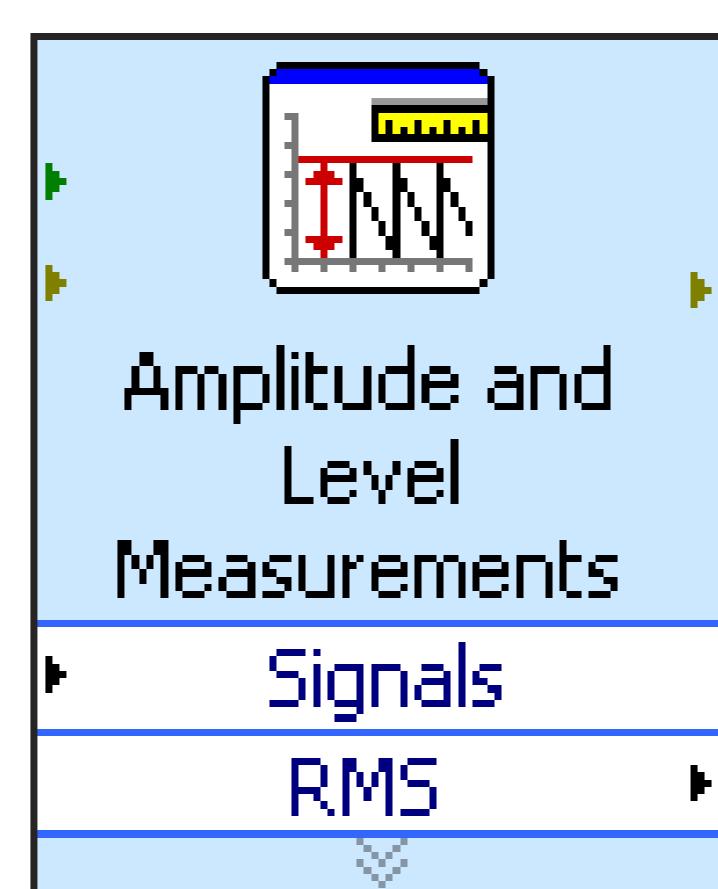
**Figure 3-1.** Front Panel of the Save Data VI



## Modifying a VI Created from a Template

Complete the following steps to create a VI that generates, analyzes, and displays a signal.

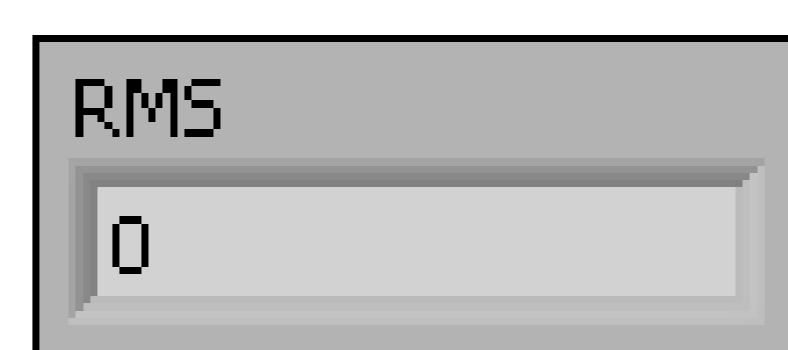
1. Select **File»New** to display the **New** dialog box.
2. From the **Create New** list, select **VI»From Template»Tutorial (Getting Started)»Generate, Analyze, and Display**. This template VI simulates a signal and analyzes it for its root mean square (RMS) value.
3. Click the **OK** button or double-click the name of the template to create a VI from the template.
4. If the **Context Help** window is not visible, press the <Ctrl-H> keys to display the window. (**Mac OS X**) Press the <Command-Shift-H> keys.
5. Display the block diagram by pressing the <Ctrl-E> keys.
6. Move the cursor over the Amplitude and Level Measurements Express VI, shown below.



The **Context Help** window displays information about the behavior of the Express VI.

Keep the **Context Help** window open. It will provide useful information as you complete the rest of this exercise.

7. Display the front panel and remove the **RMS** indicator, shown below.



You will not use the RMS functionality of the Amplitude and Level Measurements Express VI for this exercise. However, you can use the Generate, Analyze, and Display template VI with the RMS functionality in the future to reduce development time.

8. Display the block diagram and remove any broken wires that result from removing the **RMS** indicator. To remove all broken wires from the block diagram, you can press the <Ctrl-B> keys.
9. Then return to the front panel window and right-click the waveform graph indicator. Select **Properties** from the shortcut menu. The **Graph Properties** dialog box appears.
10. On the **Appearance** page, place a checkmark in the **Visible** checkbox in the **Label** section and enter **Unfiltered Signal** in the text box.
11. Click the **OK** button to save the configuration and close the **Graph Properties** dialog box.

12. Run the VI.  
The signal appears in the graph.
13. Click the **STOP** button to stop the VI.

## Adding a Signal

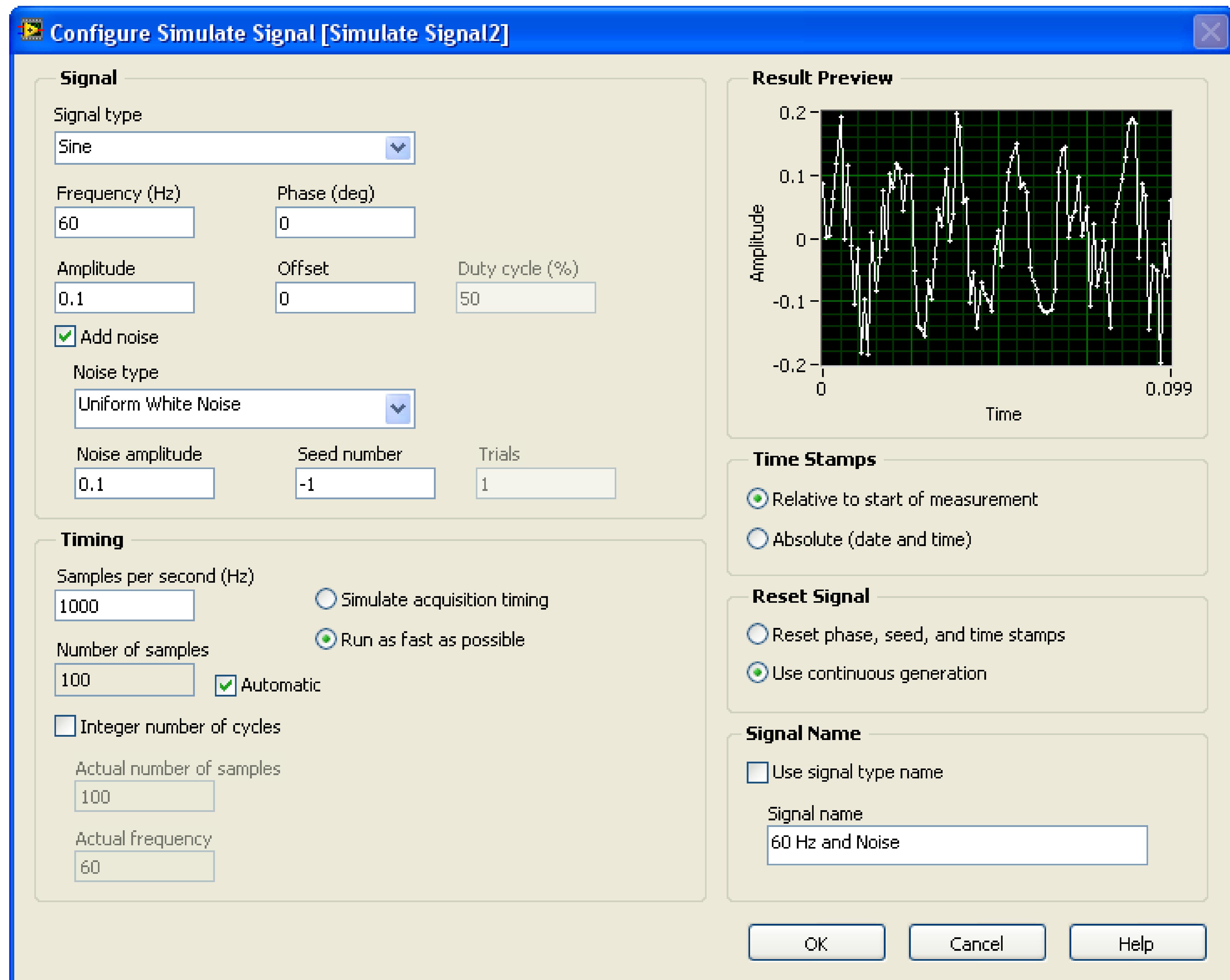
The Simulate Signal Express VI simulates a sine wave by default. You can customize the simulated signal by changing the options in the **Configure Simulate Signal** dialog box.

Complete the following steps to create an additional simulated signal that adds uniform white noise to the sine wave.

1. On the block diagram, use the Positioning tool to select the Simulate Signal Express VI. Hold down the <Ctrl> key and click and drag to create an additional Simulate Signal Express VI on the block diagram. (**Mac OS X**) Hold down the <Option> key and drag. (**Linux**) You also can hold down the middle mouse button and drag.
2. Release the mouse button to place the copied Simulate Signal Express VI below the original Simulate Signal Express VI. LabVIEW updates the name of the copied Simulate Signal Express VI to Simulate Signal2.
3. Double-click the Simulate Signal2 Express VI to display the **Configure Simulate Signal** dialog box.
4. Select **Sine** from the **Signal type** pull-down menu.
5. Enter 60 in the **Frequency (Hz)** text box.
6. Enter 0.1 in the **Amplitude** text box.
7. Place a checkmark in the **Add noise** checkbox to add noise to the sine signal.
8. Select **Uniform White Noise** from the **Noise type** pull-down menu.
9. Enter 0.1 in the **Noise amplitude** text box.
10. Enter -1 in the **Seed number** text box.
11. In the **Timing** section, select the **Run as fast as possible** option.
12. In the **Signal Name** section, remove the checkmark from the **Use signal type name** checkbox.
13. Enter 60 Hz and Noise in the **Signal name** text box.

When you change the signal name in the **Configure Simulate Signal** dialog box, LabVIEW changes the name of the signal output on the block diagram. Changing the signal name makes it easier for you to identify the signal type when you view the Express VI on the block diagram.

The **Result Preview** section displays a random signal. The **Configure Simulate Signal** dialog box should appear similar to the following figure.

**Figure 3-2.** Configure Simulate Signal Dialog Box

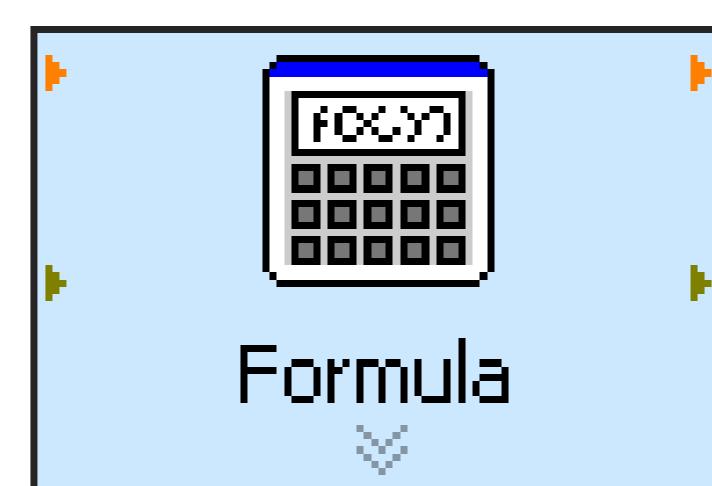
- Click the **OK** button to save the current configuration and close the **Configure Simulate Signal** dialog box.

## Adding Two Signals

To add two signals together to create one signal, you can use the Formula Express VI. Rather than displaying two signals on one graph, the Formula Express VI adds both signals together to create a single signal on the graph. You can use this Express VI to add noise to a signal.

Complete the following steps to add the **60 Hz and Noise** signal to the **Sine** signal.

- In the block diagram window, triple-click the wire that connects the **Sine** output of the Simulate Signal Express VI to the **Signals** input of the Amplitude and Level Measurements Express VI and to the **Unfiltered Signal** indicator. Remove the wire.
- On the **Functions** palette, click the **Search** button to search for the Formula Express VI, shown below, and add it to the block diagram between the Simulate Signal Express VIs and the Amplitude and Level Measurements Express VI. The **Configure Formula** dialog box appears.



**Note** LabVIEW displays a folder glyph to the left of subpalettes in the search results and displays a light blue glyph to the left of Express VIs in the search results.

3. In the **Label** column, change the label for **X1** to **Sine** and the label for **X2** to **60 Hz and Noise**.  
The Formula Express VI automatically enters the first input, **Sine**, in the **Formula** text box.
4. Click the **+** button and then the **X2** button to add **Sine** and **60 Hz and Noise** together in the **Formula** text box.
5. Click the **OK** button to save the current configuration and close the **Configure Formula** dialog box.
6. Use the Wiring tool to wire the **Sine** output of the Simulate Signal Express VI to the **Sine** input of the Formula Express VI.
7. Wire the **60 Hz and Noise** output of the Simulate Signal2 Express VI to the **60 Hz and Noise** input of the Formula Express VI.
8. Wire the **Result** output of the Formula Express VI to the **Unfiltered Signal** indicator and to the **Signals** input of the Amplitude and Level Measurements Express VI.
9. Display the front panel by pressing the <Ctrl-E> keys.
10. Run the VI.  
The signal with added noise appears in the graph.
11. Click the **STOP** button to stop the VI.
12. Select **File»Save As** and save the VI as **Analysis.vi** in an easily accessible location.

## Filtering a Signal

You can use the Filter Express VI to process signals through filters and windows.

Complete the following steps to configure the Filter Express VI to filter the signal using an infinite impulse response (IIR) filter.

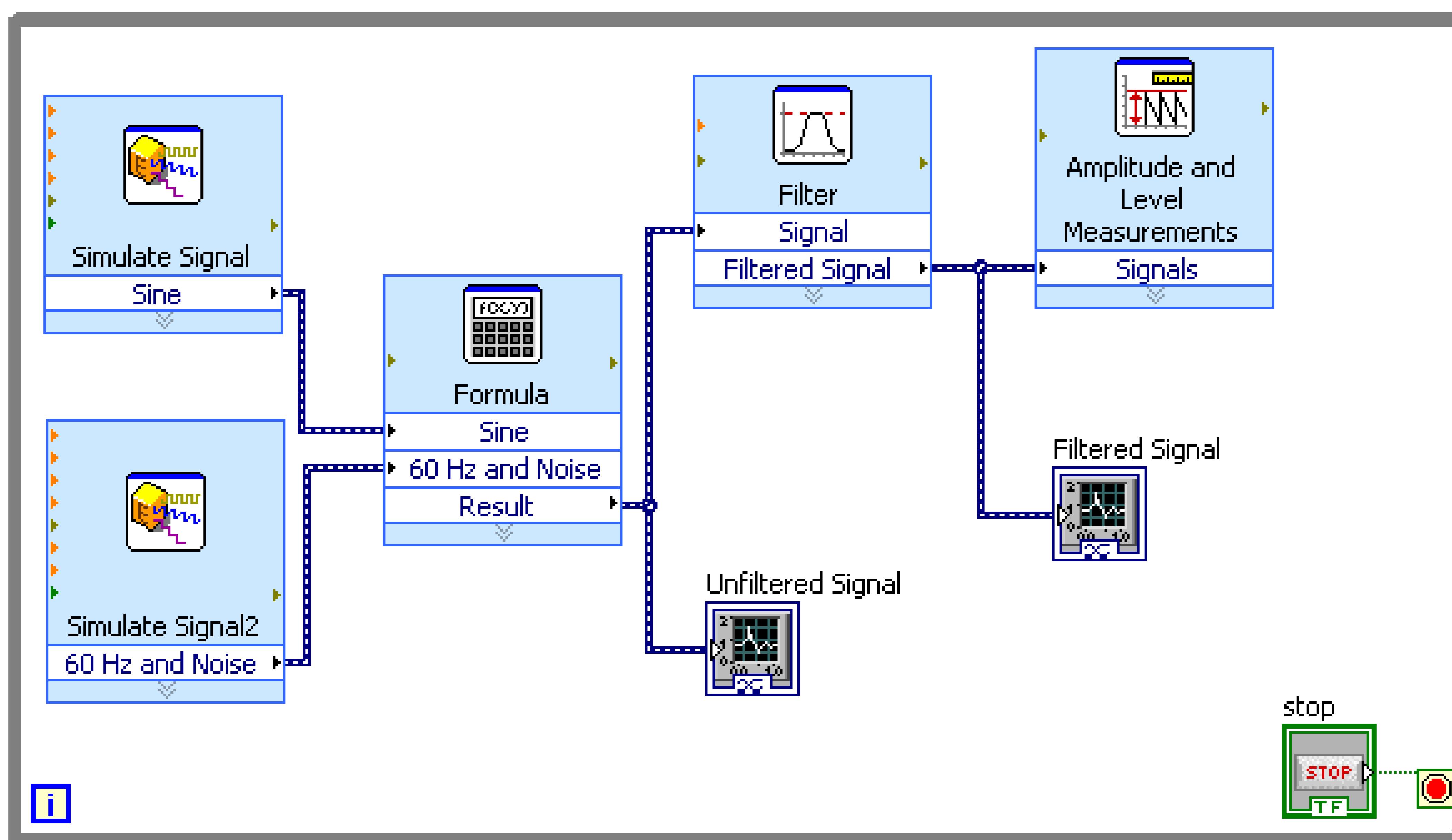
1. Display the block diagram window and remove the wire that connects the **Result** output of the Formula Express VI to the **Signals** input of the Amplitude and Level Measurements Express VI.
2. Remove all broken wires that result from removing the wire.

3. Search for the Filter Express VI, shown below, and add it to the block diagram between the Simulate Signal2 Express VI and the Amplitude and Level Measurements Express VI. The **Configure Filter** dialog box appears.



4. In the **Filter Specifications** section, change the **Cutoff Frequency (Hz)** to 25.
5. Click the **OK** button to save the configuration and close the **Configure Filter** dialog box.
6. Display the front panel.
7. Click the **Unfiltered Signal** waveform graph indicator and press the <Ctrl> key while you drag with the Positioning tool to create an additional waveform graph indicator.
8. Triple-click the **Unfiltered Signal 2** label above the new waveform graph indicator and enter **Filtered Signal** to change the label of the indicator. You also can change the label on the **Appearance** page of the **Graph Properties** dialog box.
9. On the block diagram, wire the **Result** output of the Formula Express VI to the **Signal** input of the Filter Express VI.
10. Wire the **Filtered Signal** output of the Filter Express VI to the **Signals** input of the Amplitude and Level Measurements Express VI and to the input of the **Filtered Signal** waveform graph indicator.
11. Select **File»Save**. The block diagram of the Analysis VI should appear similar to the following figure.

**Figure 3-3.** Block Diagram of the Analysis VI



## Modifying the Appearance of Graphs

You can use the **Display Format** page of the **Graph Properties** dialog box to specify how the scales of the x-axis and y-axis appear on the graph.

Complete the following steps to change the format of the x-axis and y-axis of the **Unfiltered Signal** and **Filtered Signal** graphs.

1. In the front panel window, right-click the **Unfiltered Signal** graph indicator and select **Properties** from the shortcut menu. The **Graph Properties** dialog box appears.
2. On the **Display Format** page, select **Time (X-Axis)** from the top pull-down menu.
3. Select the **Default editing mode** option.
4. In the **Type** list, select **Automatic formatting**.
5. In the **Digits** field, enter 6 and select **Significant digits** from the **Precision Type** pull-down menu.
6. Place a checkmark in the **Hide trailing zeros** checkbox.
7. Select **Amplitude (Y-Axis)** from the top pull-down menu and repeat steps 3 through 6 so the y-axis configuration matches the x-axis configuration.
8. On the **Scales** page, select **Amplitude (Y-Axis)**.
9. Remove the checkmark from the **Autoscale** checkbox.
10. Enter  $-2 . 5$  in the **Minimum** text box and  $2 . 5$  in the **Maximum** text box.
11. Click the **OK** button to save the configuration and close the **Graph Properties** dialog box.
12. Repeat steps 1 through 11 to configure the **Filtered Signal** graph indicator.

The x-axis and the y-axis on the **Unfiltered Signal** and **Filtered Signal** graph indicators change to reflect the new configuration.

## Analyzing the Amplitude of a Signal

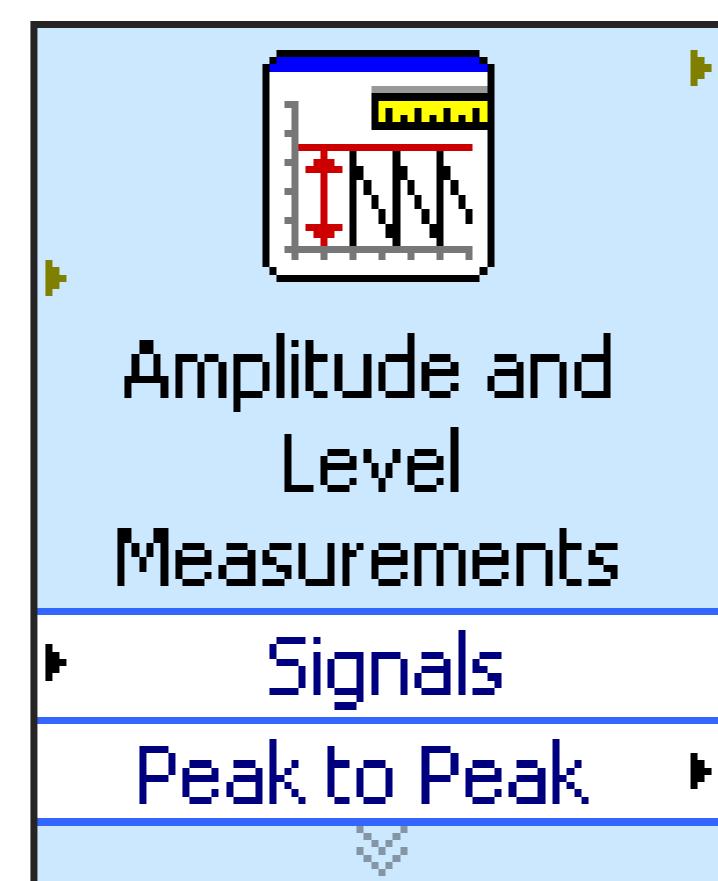
You can use the Amplitude and Level Measurements Express VI to analyze the voltage characteristics of a signal.

Complete the following steps to reconfigure the Express VI to measure the peak-to-peak amplitude values of the signal.

1. On the block diagram, double-click the Amplitude and Level Measurements Express VI to display the **Configure Amplitude and Level Measurements** dialog box.
2. In the **Amplitude Measurements** section, remove the checkmark from the **RMS** checkbox.
3. Place a checkmark in the **Peak to peak** checkbox. Peak to peak appears in the **Results** section with the corresponding value of the measurement.

4. Click the **OK** button to save the current configuration and close the **Configure Amplitude and Level Measurements** dialog box.

The **RMS** output of the Amplitude and Level Measurements Express VI changes to reflect the new **Peak to Peak** output, shown below.



You will use the **Peak to Peak** output in a later exercise.

## Controlling the Speed of Execution

To plot the points on the waveform graphs more slowly, you can add a time delay to the block diagram. A time delay slows the speed at which a VI runs.

Complete the following steps to control the speed at which the VI runs.

1. On the block diagram, search for the Time Delay Express VI.
2. Place the Time Delay Express VI inside the While Loop. The **Configure Time Delay** dialog box appears.
3. Enter **1.000** in the **Time delay (seconds)** text box and click the **OK** button.
4. Display the front panel and run the VI. The VI runs more slowly.  
The loop iterates once every second.
5. Stop the VI.

Another way to control the speed of the VI is to alter the rate of data acquisition. On the block diagram, double click the **Simulate Signal** Express VI to display the **Configure Simulate Signal** dialog box. Locate the **Timing** section in the dialog box. The **Timing** section contains a number of ways to alter the rate of data acquisition and the speed at which a VI runs.

For example, one of the default settings of the VI is **Simulate Acquisition Timing**. This means that the VI mimics the acquisition rate of a hardware device. You can select **Run as fast as possible** to display data more quickly. In the **Samples per second (Hz)** text box, the default value is 1000, while the default value in the **Number of Samples** text box is 100. This means that the VI will output 100 data points spanning 0.1 second. You can change these values to change the amount of data the VI displays, as well as the rate at which the VI displays the data.

## Adding a Warning Light

If you want a visual cue to indicate when a value exceeds a specified limit, you can use a warning light.

Complete the following steps to add a warning light to the VI.

1. Display the **Controls** palette by right-clicking any blank space in the front panel window.
2. On the **Express** palette, select the **LEDs** palette.
3. Select the round LED indicator and add it to the front panel to the left of the waveform graphs.
4. Double-click the **Boolean** label above the LED and enter **Warning** to change the label of the LED.

You will use this LED in a later exercise to indicate when a value has exceeded its limit.

5. Select **File»Save As** to display the **Save As** dialog box.
6. Read the various dialog box options. Select the **Copy** and **Substitute copy for original** options to create a copy of the original VI and immediately edit the copy.
7. Click the **Continue** button and save the VI as **Warning Light.vi** in an easily accessible location.

## Setting a Warning Level Limit

To specify the value at which you want the LED to light, use the Comparison Express VI.

Complete the following steps to compare the peak-to-peak value to a limit you set.

1. On the block diagram, search for the Comparison Express VI and place it to the right of the Amplitude and Level Measurements Express VI. The **Configure Comparison** dialog box appears.
2. In the **Compare Condition** section, select the **> Greater** option.
3. In the **Comparison Inputs** section, select **Value** and enter **2** in the **Value** numeric control to assign a constant value at which you want the LED to light.
4. Click the **OK** button to save the current configuration and close the **Configure Comparison** dialog box.

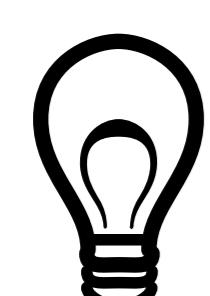
The name of the Comparison Express VI changes to reflect the operation of the Express VI, shown below. **Greater** indicates that the Express VI does a greater than comparison.



5. Wire the **Peak to Peak** output of the Amplitude and Level Measurements Express VI to the **Operand 1** input of the Greater Express VI.
6. Move the cursor over the wire that connects the **Peak to Peak** output to the **Operand 1** input.

7. When the Positioning tool appears, right-click the wire that connects the **Peak to Peak** output to the **Operand 1** input and select **Create»Numeric Indicator** from the shortcut menu.

A **Peak to Peak** terminal, shown below, appears on the block diagram. If the **Peak to Peak** terminal appears to be on top of the wires between the Express VIs, move the Express VIs and **Peak to Peak** terminal around to create more space. For example, move the **Peak to Peak** terminal into blank space above the Express VIs.



**Tip** You also can press <Ctrl> and drag a rectangle in open space to add more working space to the front panel or block diagram.

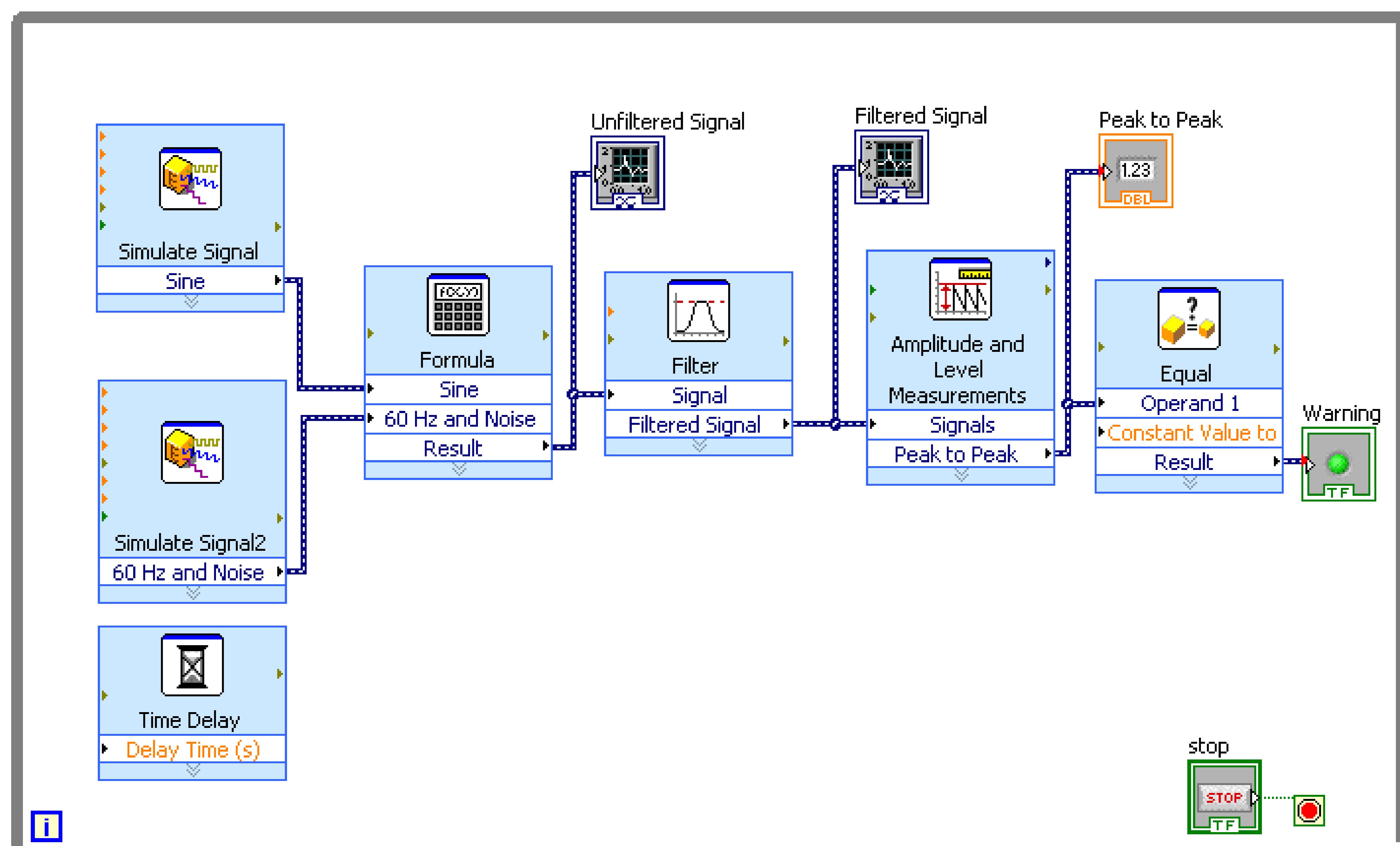
## Warning the User

After you specify the values at which you want the LED to light, you must wire the LED to the Greater Express VI.

Complete the following steps to provide a visual cue when the peak-to-peak value of the signal exceeds a specified limit.

1. In the block diagram window, move the **Warning** terminal to the right of the Greater Express VI. Make sure the **Warning** terminal is inside the While Loop, as shown in the following figure.

**Figure 3-4.** Block Diagram of the Warning Light VI



2. Wire the **Result** output of the Greater Express VI to the **Warning** terminal.

The block diagram should appear similar to Figure 3-4.



**Note** Red coercion dots appear automatically at the **Peak to Peak** and **Warning** input terminals to alert you that you wired two different data types together and LabVIEW converted the value passed into the node to a different representation. The block diagram places the coercion dots on the border of the terminal where the conversion takes place. For this exercise, the conversion does not affect how the VI runs. Refer to the *LabVIEW Help* for more information about coercion dots.

3. Display the front panel.

A numeric indicator labeled **Peak to Peak** appears in the front panel window. This indicator displays the peak-to-peak value of the signal.

4. Run the VI.

When the peak-to-peak value exceeds 2 . 0, the **Warning** indicator lights.

5. Click the **STOP** button to stop the VI.

6. Save the VI.

## Configuring a VI to Save Data to a File

To store information about the data a VI generates, use the Write To Measurement File Express VI.

Complete the following steps to build a VI that saves peak-to-peak values and other information to a LabVIEW data file.

1. Search for the Write To Measurement File Express VI and add it to the block diagram below and to the right of the Amplitude and Level Measurements Express VI.

The **Configure Write To Measurement File** dialog box appears.

The **Filename** text box displays the full path to the output file, `test.lvm`. A `.lvm` file is a tab-delimited text measurement file you can open with a spreadsheet application or a text-editing application. LabVIEW saves data with up to six digits of precision in a `.lvm` file. LabVIEW saves the `.lvm` file in the default LabVIEW Data directory. LabVIEW installs the LabVIEW Data directory in the default file directory of the operating system.

When you want to view the data, use the file path displayed in the **Filename** text box to access the `test.lvm` file.

2. In the **Configure Write to Measurement File** dialog box, locate the **If a file already exists** section and select the **Append to file** option to write all the data to the `test.lvm` file without erasing any existing data in the file.
3. In the **Segment Headers** section, select the **One header only** option to create only one header in the file to which LabVIEW writes the data.

4. Enter the following text in the **File Description** text box: Sample of peak to peak values. LabVIEW appends the text you enter in this text box to the header of the file.
5. Click the **OK** button to save the current configuration and close the **Configure Write To Measurement File** dialog box.

## Saving Data to a File

When you run the VI, LabVIEW saves the data to the `test.lvm` file.

Complete the following steps to generate the `test.lvm` file.

1. On the block diagram, wire the **Peak to Peak** output of the Amplitude and Level Measurements Express VI to the **Signals** input of the Write To Measurement File Express VI.
2. Select **File»Save As** and save the VI as `Save Data.vi` in an easily accessible location.
3. Display the front panel and run the VI.
4. Click the front panel **STOP** button.
5. To view the data you saved, open the `test.lvm` file in the LabVIEW Data directory with a spreadsheet or text-editing application.  
The file has a header that contains information about the Express VI.
6. Close the file after you finish looking at it and return to the Save Data VI.

## Adding a Button That Stores Data When Clicked

If you want to store only certain data points, you can configure the Write To Measurement File Express VI to save peak-to-peak values only when a user clicks a button.

Complete the following steps to add a button to the VI and configure how the button responds when a user clicks it.

1. Display the front panel and search the **Controls** palette for a rocker button. Select one of the rocker buttons and place it to the right of the waveform graphs.
2. Right-click the rocker button and select **Properties** from the shortcut menu to display the **Boolean Properties** dialog box.
3. Change the label of the button to `Write to File`.
4. On the **Operation** page of the **Boolean Properties** dialog box, select **Latch when pressed** from the **Button behavior** list.

Use the **Operation** page to specify how a button behaves when a user clicks it. To see how the button reacts to a click, click the button in the **Preview Selected Behavior** section.

5. Click the **OK** button to save the current configuration and close the **Boolean Properties** dialog box.
6. Save the VI.

# Saving Data When Prompted by a User

Complete the following steps to build a VI that logs data to a file when the user clicks a button on the front panel.

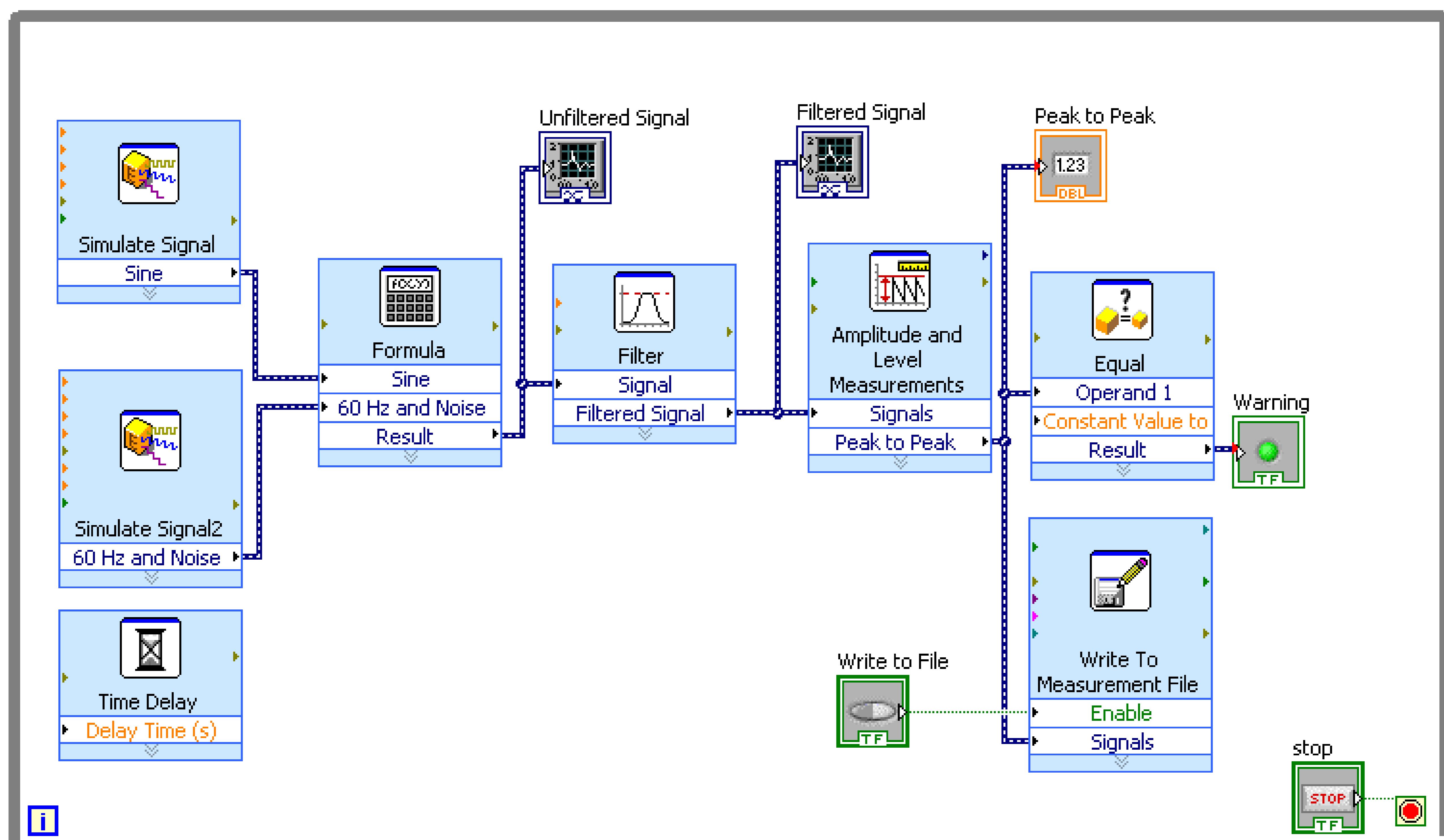
1. In the block diagram window, double-click the Write To Measurement File Express VI to display the **Configure Write To Measurement File** dialog box.
2. Change the filename `test.lvm` in the **Filename** text box to `Selected Samples.lvm` to save the data to a different file.
3. Click the **OK** button to save the current configuration and close the **Configure Write To Measurement File** dialog box.
4. Right-click the **Signals** input of the Write To Measurement File Express VI and select **Insert Input/Output** from the shortcut menu to insert the **Comment** input.
5. Right-click the **Comment** input of the Write To Measurement File Express VI and select **Select Input/Output»Enable** from the shortcut menu to replace the **Comment** input with the **Enable** input.

The inputs and outputs of an Express VI appear in a predetermined order when you add new inputs and outputs. To select a specific input, you might need to add an input first, and then change the input to the specific one you want to use by right-clicking the input and selecting **Select Input/Output** from the shortcut menu.

6. Move the **Write to File** terminal to the left of the Write To Measurement File Express VI.
7. Wire the **Write to File** terminal to the **Enable** input of the Write To Measurement File Express VI.

The block diagram should appear similar to the following figure.

**Figure 3-5.** Block Diagram of the Save Data VI



## Viewing Saved Data

Complete the following steps to view the data that you save to the `Selected Samples.lvm` file.

1. Display the front panel and run the VI. Click the **Write to File** button several times.
2. Click the **STOP** button.
3. Open the `Selected Samples.lvm` file with a spreadsheet or text-editing application.  
The `Selected Samples.lvm` file differs from the `test.lvm` file. `test.lvm` recorded all the data generated by the Save Data VI, whereas `Selected Samples.lvm` recorded data only when you clicked the **Write to File** button.
4. Close the file after you finish looking at it.
5. Save and close the VI.

## Summary

---

The following topics are a summary of the main concepts you learned in this chapter.

### Controls and Indicators

You can configure front panel controls and indicators to perform tasks depending on what you want a VI to do. In this chapter, you learned to use controls and indicators in the following ways:

- You can build VIs that perform a task when certain conditions occur, such as displaying a warning light when a value exceeds a certain limit.
- You can build VIs that let users control when an Express VI executes by using buttons and the **Enable** input. You can configure the buttons to operate in one of six ways using the **Operation** page of the **Boolean Properties** dialog box.

### Filtering Data

The Filter Express VI processes signals through filters and windows. You can use the Filter Express VI to remove noise from a signal.

### Saving Data

The Write To Measurement File Express VI saves data that a VI generates and analyzes to a `.lvm`, `.tdm`, or `.tdms` measurement file. The text-based measurement file (`.lvm`) is a tab-delimited text file you can open with a spreadsheet application or a text-editing application. LabVIEW saves data with up to six digits of precision in a `.lvm` file. In addition to the data an Express VI generates, the `.lvm` file includes headers that contain information about the data, such as the date and time LabVIEW generated the data. The binary measurement file (`.tdm`) is a binary file that contains waveform data. Binary `.tdm` files provide higher accuracy for floating-point numbers, take up less space on disk, and perform faster than text-based measurement files (`.lvm`). The TDM Streaming file (`.tdms`) is a binary file that provides faster writing performance than the `.tdm` file format and allows a simpler interface for defining properties.

LabVIEW installs the LabVIEW Data directory in the default file directory of the operating system to help you organize and locate the data files LabVIEW generates. Refer to the *LabVIEW Help* for more information about writing data to and reading data from .lvm and .tdm files.