LabVIEW™

Getting Started with LabVIEW

April 2003 Edition
Part Number 323427A-01

Contents

About This Manual

Chapter 1
Getting Started with LabVIEW Virtual Instruments

Chapter 2
Analyzing and Saving a Signal

About This Manual

Use this manual to familiarize yourself with the LabVIEW graphical programming environment and the basic LabVIEW features you use to build data acquisition and instrument control applications.

This manual contains exercises to teach you how to develop basic applications in LabVIEW. These exercises take a short amount of time to complete and help you get started with LabVIEW.

Throughout each exercise, you will see illustrations that provide information about the concepts associated with that step. The end of each chapter includes a summary of the main concepts taught in that chapter. Use these summaries to review what you learned.

In addition to this manual, LabVIEW includes the *LabVIEW User Manual*, *LabVIEW Help*, other reference manuals, Application Notes, and examples. If you select the **Complete** install option, LabVIEW installs PDF versions of all LabVIEW manuals, which you can access by selecting **Help»Search the LabVIEW Bookshelf** in LabVIEW.

Conventions

The following conventions appear in this manual:

» The **»** symbol leads you through nested menu items and dialog box options to a final action. The sequence **File»Page Setup»Options** directs you to pull down the **File** menu, select the **Page Setup** item, and select **Options** from the last dialog box.

This icon denotes a tip, which alerts you to advisory information.

This icon denotes a note, which alerts you to important information.

bold Bold text denotes items that you must select or click in the software, such as menu items and dialog box options. Bold text also denotes parameter names, controls and buttons on the front panel, dialog boxes, sections of dialog boxes, menu names, and palette names.

italic Italic text denotes variables, emphasis, a cross reference, or an introduction to a key concept. This font also denotes text that is a placeholder for a wordor value that you must supply.

`monospace` Text in this font denotes text or characters that you should enter from the keyboard, sections of code, programming examples, and syntax examples. This font is also used for the proper names of disk drives, paths, directories, programs, subprograms, subroutines, device names, functions, operations, variables, filenames and extensions, and code excerpts.

`monospace bold` Bold text in this font denotes the messages and responses that the computer automatically prints to the screen. This font also emphasizes lines of code that are different from the other examples.

1

Getting Started with LabVIEW Virtual Instruments

LabVIEW programs are called virtual instruments, or VIs, because their appearance and operation imitate physical instruments, such as oscilloscopes and multimeters. LabVIEW contains a comprehensive set of tools for acquiring, analyzing, displaying, and storing data, as well as tools to help you troubleshoot your code.

In LabVIEW, you build a user interface, or front panel, with controls and indicators. Controls are knobs, push buttons, dials, and other input devices. Indicators are graphs, LEDs, and other displays. After you build the user interface, you add code using VIs and structures to control the front panel objects. The block diagram contains this code.

Use LabVIEW to communicate with hardware such as data acquisition, vision, and motion control devices and GPIB, PXI, VXI, RS-232, and RS-485 instruments.

Building a Virtual Instrument

In the following exercises, you will build a VI that generates a signal and displays that signal in a graph. When you complete the exercises, the front panel of the VI will look similar to the front panel in Figure 1-1.

You can complete the exercises in approximately 40 minutes.

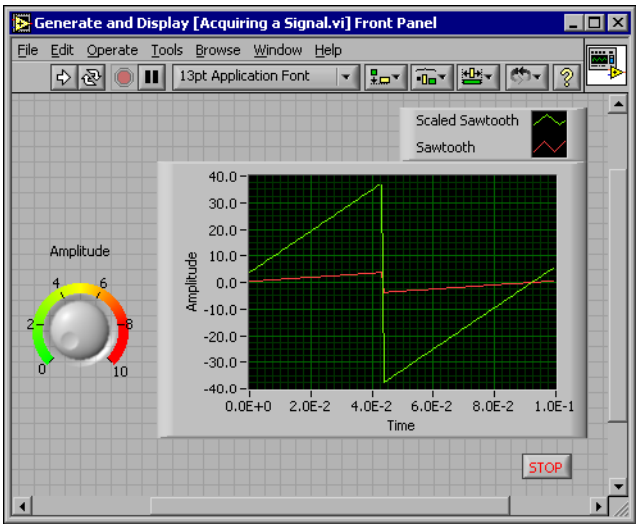


Figure 1-1. Front Panel for the Acquiring a Signal VI

Opening a New VI from a Template

LabVIEW provides templates containing information from which you can build your VI. These templates help you get started with LabVIEW. Complete the following steps to create a VI that generates a signal and displays it on the front panel.

1. Launch LabVIEW.
2. In the **LabVIEW** dialog box, shown in Figure 1-2, click the **New** button to display the **New** dialog box.

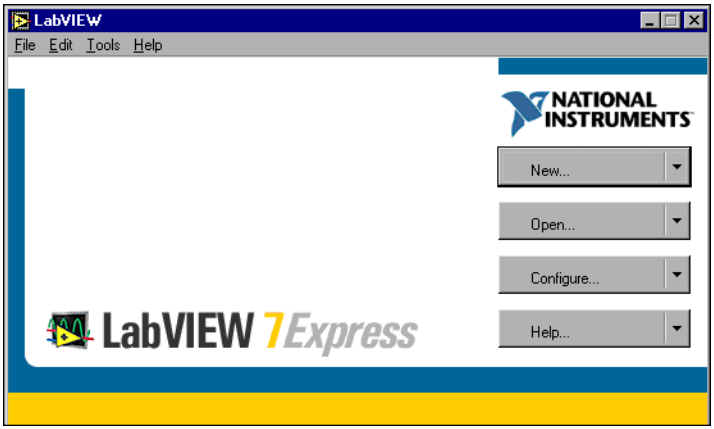


Figure 1-2. LabVIEW Dialog Box

3. Select **VI from Template»Tutorial (Getting Started)»Generate and Display** in the **Create new** list. This template VI generates and displays a signal.

   Notice that previews of the template VI appear in the **Front panel preview** and the **Block diagram preview** sections. Figure 1-3 shows the **New** dialog box and the Generate and Display template VI.

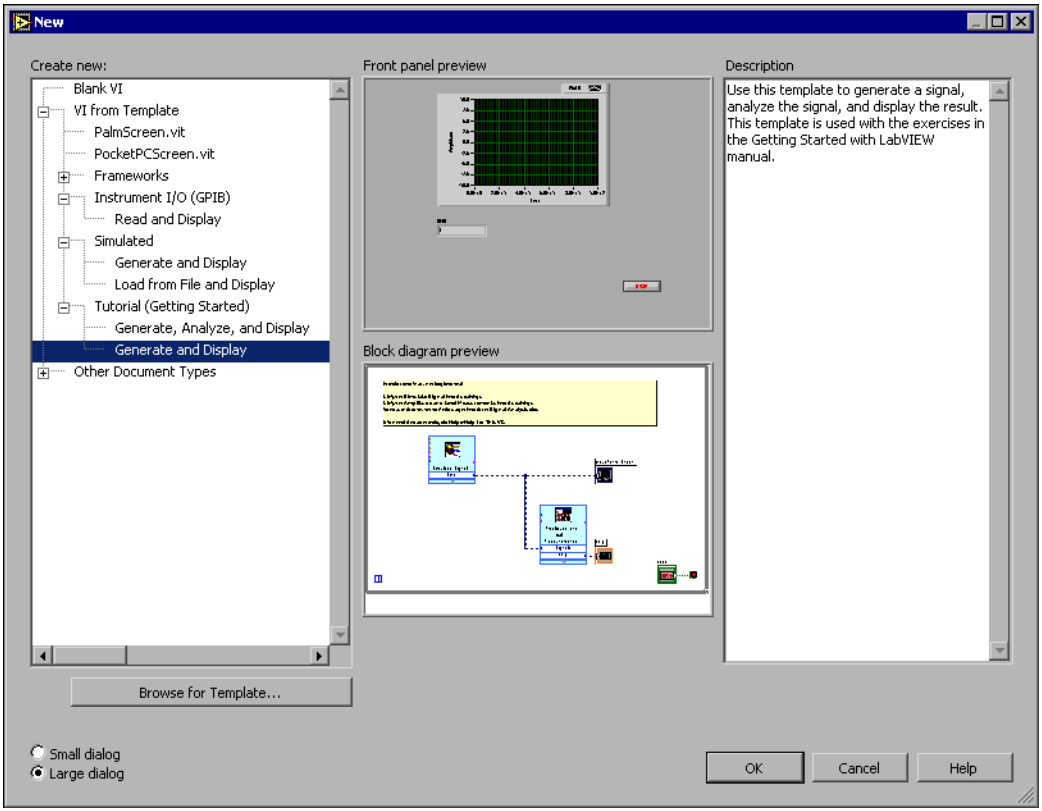


Figure 1-3. New Dialog Box

4. Click the **OK** button to open the template. You also can double-click the name of the template VI in the **Create new** list to open the template.

5. Examine the front panel of the VI.

   The user interface, or front panel, appears with a gray background and includes controls and indicators. The title bar of the front panel indicates that this window is the front panel for the Generate and Display VI.

Note If the front panel is not visible, you can display the front panel by selecting **Window»Show Front Panel**.

6. Examine the block diagram of the VI.

   The block diagram appears with a white background and includes VIs and structures that control the front panel objects. The title bar of the block diagram indicates that this window is the block diagram for the Generate and Display VI.

Note If the block diagram is not visible, you can display the block diagram by selecting **Window»Show Block Diagram**.

7. In the front panel toolbar, click the **Run** button, shown at left.

   Notice that a sine wave appears on the graph.

8. Stop the VI by clicking the **STOP** button, shown at left, on the front panel.

Adding a Control to the Front Panel

Controls on the front panel simulate the input devices on a physical instrument and supply data to the block diagram of the VI. Many physical instruments have knobs you can turn to change an input value. Complete the following steps to add a knob control to the front panel.

Tip Throughout these exercises, you can undo recent edits by selecting **Edit»Undo** or pressing the <Ctrl-Z> keys.

1. If the **Controls** palette, shown in Figure 1-4, is not visible on the front panel, select **Window»Show Controls Palette** to display it.

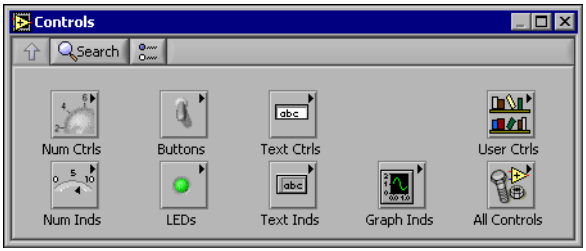


Figure 1-4. Controls Palette

2. Move the cursor over the icons on the **Controls** palette to locate the **Numeric Controls** palette.

   Notice that when you move the cursor over icons on the **Controls** palette, the name of that subpalette appears in the gray space above all the icons on the palette. When you idle the cursor over any icon on any palette, the full name of the subpalette, control, or indicator appears.

3. Click the **Numeric Controls** icon to access the **Numeric Controls** palette.

4. Select the knob control on the **Numeric Controls** palette and place it on the front panel to the left of the waveform graph.

   You will use this knob in a later exercise to control the amplitude of a signal.

5. Select **File»Save As** and save this VI as `Acquiring a Signal.vi` to an easily accessible location.

Changing the Signal Type

The block diagram has a blue icon labeled **Simulate Signal**. This icon represents the Simulate Signal Express VI. The Simulate Signal Express VI simulates a sine wave by default. Complete the following steps to change this signal to a sawtooth wave.

1. Display the block diagram by selecting **Window»Show Block Diagram** or by clicking the block diagram.

   Notice the Simulate Signal Express VI, shown at left. An Express VI is a component of the block diagram that you can configure to perform common measurement tasks. The Simulate Signal Express VI simulates a signal based on the configuration that you specify.

2. Right-click the Simulate Signal Express VI and select **Properties** from the shortcut menu to display the **Configure Simulate Signal** dialog box.

3. Select **Sawtooth** from the **Signal type** pull-down menu.

   Notice that the waveform on the graph in the **Result Preview** section changes to a sawtooth wave. The **Configure Simulate Signal** dialog box should appear similar to Figure 1-5.

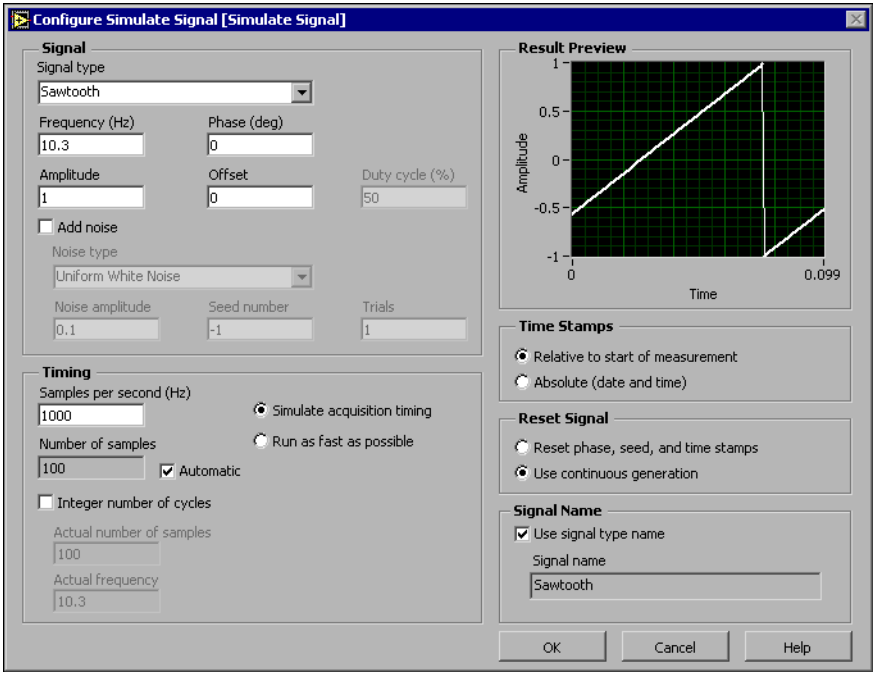
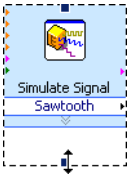


Figure 1-5. Configure Simulate Signal Dialog Box

4. Click the **OK** button to apply the current configuration and close the **Configure Simulate Signal** dialog box.
5. Move the cursor over the down arrows at the bottom of the Simulate Signal Express VI.



6. When a double-headed arrow appears, shown at left, click and drag the border of the Express VI until the **Amplitude** input appears.

   Notice how you expanded the Simulate Signal Express VI to display a new input. Because the **Amplitude** input appears on the block diagram, you can configure the amplitude of the sawtooth wave on the block diagram.

   In Figure 1-5, notice how **Amplitude** is an option in the **Configure Simulate Signal** dialog box. When inputs, such as **Amplitude**, appear on the block diagram and in the configuration dialog box, you can configure the inputs in either location.

Wiring Objects on the Block Diagram

To use the knob control to change the amplitude of the signal, you must connect the two objects on the block diagram. Complete the following steps to wire the knob to the **Amplitude** input on the Simulate Signal Express VI.

1. Move the cursor over the **Knob** terminal, shown at left, until the Positioning tool appears.

   Notice how the cursor becomes an arrow, or the Positioning tool, shown at left. Use the Positioning tool to select, position, and resize objects.

2. Click the **Knob** terminal to select it, then drag the terminal to the left of the Simulate Signal Express VI. Make sure the **Knob** terminal is inside the loop, shown at left.

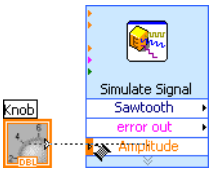
   The terminals are representations of front panel controls and indicators. Terminals are entry and exit ports that exchange information between the front panel and block diagram.

3. Deselect the **Knob** terminal by clicking a blank space on the block diagram.

4. Move the cursor over the arrow of the **Knob** terminal, shown at left.

   Notice how the cursor becomes a wire spool, or the Wiring tool, shown at left. Use the Wiring tool to wire objects together on the block diagram.

Note The cursor does not switch to another tool while an object is selected.



5. When the Wiring tool appears, click the arrow and then click the **Amplitude** input of the Simulate Signal Express VI, shown at left, to wire the two objects together.

   Notice that a wire appears and connects the two objects. Data flows along this wire from the terminal to the Express VI.

6. Select **File»Save** to save this VI.

Running the VI

Running a VI executes your solution. Complete the following steps to run the Acquiring a Signal VI.

1. Display the front panel by selecting **Window»Show Front Panel** or by clicking the front panel.

Tip Press the <Ctrl-E> keys to switch from the front panel to the block diagram or from the block diagram to the front panel.

2. Click the **Run** button.
3. Move the cursor over the knob control.

Notice how the cursor becomes a hand, or the Operating tool, shown at left. Use the Operating tool to change the value of a control or select the text within a control.

4. Using the Operating tool, turn the knob to adjust the amplitude of the sawtooth wave.

Notice how the amplitude of the sawtooth wave changes as you turn the knob. Also notice that the y-axis on the graph autoscales to account for the change in amplitude.

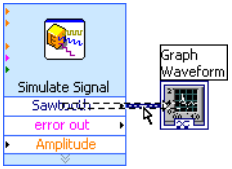
To indicate that the VI is running, the **Run** button changes to a darkened arrow, shown at left. You cannot edit the front panel or block diagram while the VI runs.

5. Click the **STOP** button, shown at left, to stop the VI.

Note Although the **Abort Execution** button  looks like a stop button, the **Abort Execution** button does not always properly close the VI. National Instruments recommends stopping your VIs using the **STOP** button on the front panel. Use the **Abort Execution** button only when errors prevent you from terminating the application using the **STOP** button.

Modifying the Signal

Complete the following steps to add scaling to the signal and display the results in the graph on the front panel.



1. On the block diagram, use the Positioning tool to double-click the wire that connects the Simulate Signal Express VI to the **Waveform Graph** terminal, shown at left.
2. Press the <Delete> key to delete this wire.
3. If the **Functions** palette, shown in Figure 1-6, is not visible, select **Window»Show Functions Palette** to display it.

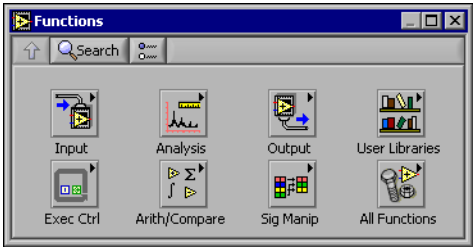


Figure 1-6. Functions Palette

4. Select the Scaling and Mapping Express VI, shown at left, on the **Arithmetic & Comparison** palette and place it on the block diagram inside the loop between the Simulate Signal Express VI and the **Waveform Graph** terminal. If there is no room between the Express VI and the terminal, move the **Waveform Graph** terminal to the right.

   Notice that the **Configure Scaling and Mapping** dialog box automatically opens when you place the Express VI on the block diagram.

5. Define the value of the scaling factor by entering `10` in the **Slope (m)** text box.

   The **Configure Scaling and Mapping** dialog box should appear similar to Figure 1-7.

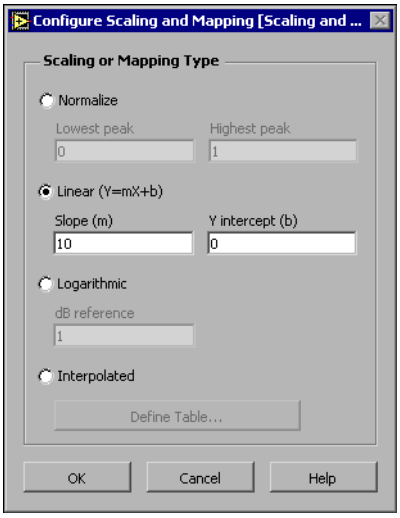
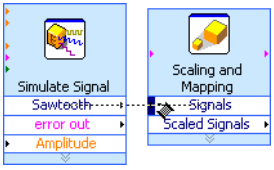


Figure 1-7. Configure Scaling and Mapping Dialog Box

6. Click the **OK** button to apply the current configuration and close the **Configure Scaling and Mapping** dialog box.
7. Move the cursor over the arrow on the **Sawtooth** output of the Simulate Signal Express VI.
8. When the Wiring tool appears, click the arrow and then click the arrow on the **Signals** input of the Scaling and Mapping Express VI, shown at left, to wire the two objects together.



9. Using the Wiring tool, wire the **Scaled Signals** output of the Scaling and Mapping Express VI to the **Waveform Graph** terminal.

   Notice the wires connecting the Express VIs and terminals. The arrows on the Express VIs and terminals indicate the direction that the data flows along these wires. The block diagram should appear similar to Figure 1-8.

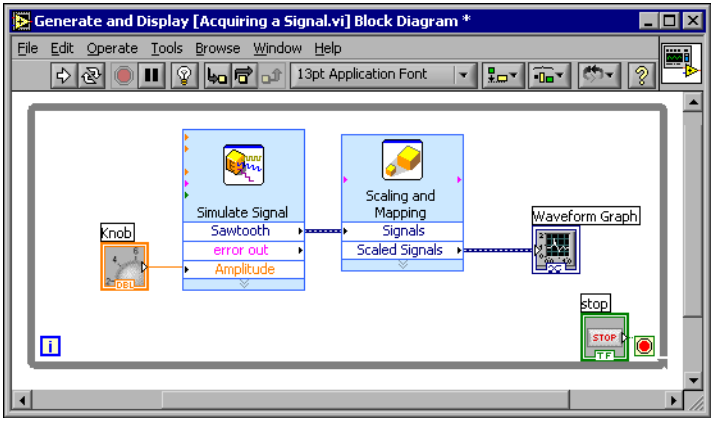


Figure 1-8. Block Diagram for the Acquiring a Signal VI

10. Select **File»Save** to save this VI.

Displaying Two Signals on the Graph

To compare the signal generated by the Simulate Signal Express VI and the signal modified by the Scaling and Mapping Express VI on the same graph, use the Merge Signals function. Complete the following steps to display two signals on the same graph.

1. Move the cursor over the arrow on the **Sawtooth** output of the Simulate Signal Express VI.

2. Using the Wiring tool, wire the **Sawtooth** output to the **Waveform Graph** terminal.

The Merge Signals function, shown at left, appears where the two wires connect. This function takes the two separate signals and combines them so that both can be displayed on the same graph. The block diagram should appear similar to Figure 1-9.

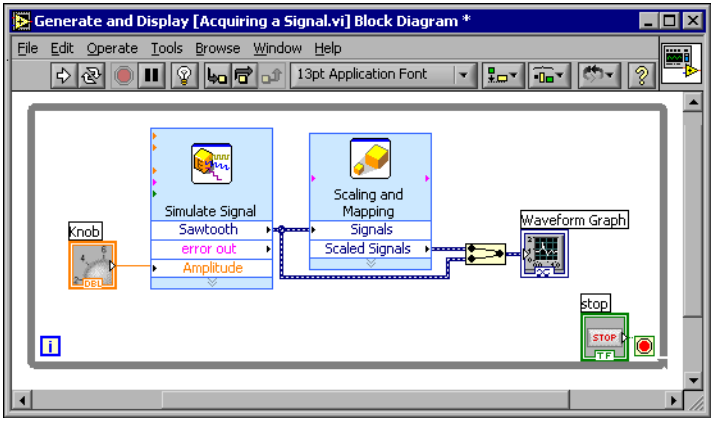


Figure 1-9. Block Diagram Showing the Merge Signals Function

3. Select **File»Save** to save this VI. You also can press the <Ctrl-S> keys to save a VI.
4. Return to the front panel, run the VI, and turn the knob control.

   Notice that the graph plots the sawtooth wave and the scaled signal. Also notice that the maximum value on the y-axis automatically changes to be 10 times the knob value. This scaling occurs because you set the slope to 10 in the Scaling and Mapping Express VI.
5. Click the **STOP** button.

Customizing the Knob

The knob control changes the amplitude of the sawtooth wave so labeling it **Amplitude** accurately describes the function of the knob. Complete the following steps to customize the appearance of a control on the front panel.

1. Right-click the knob and select **Properties** from the shortcut menu to display the **Knob Properties** dialog box.
2. In the **Label** section on the **Appearance** tab, delete the label **Knob**, and enter `Amplitude` in the text box.

The **Knob Properties** dialog box should appear similar to Figure 1-10.

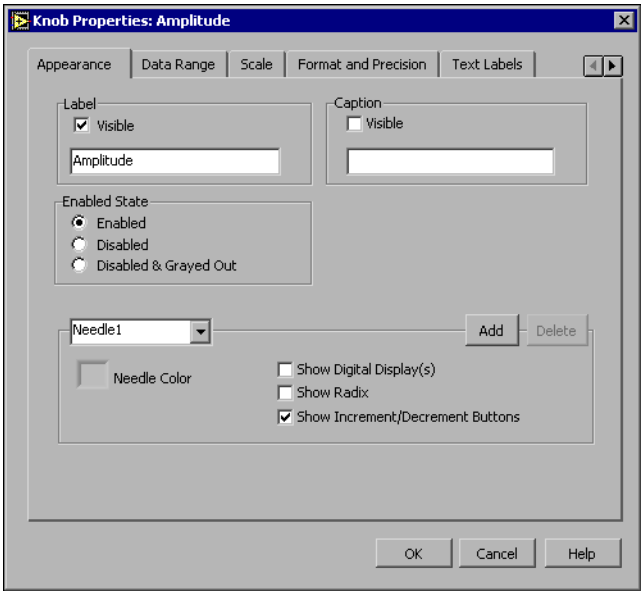


Figure 1-10. Knob Properties Dialog Box

3. Click the **Scale** tab and in the **Scale Style** section, place a checkmark in the **Ramp Visible** checkbox.

   Notice how the knob on the front panel instantly updates to reflect these changes.

4. Click the **OK** button to apply the current configuration and close the **Knob Properties** dialog box.

5. Save this VI.

Tip As you build your VIs, you can experiment with different properties and configurations. You also can add and delete objects. Remember you can undo recent edits by selecting **Edit»Undo** or pressing the <Ctrl-Z> keys.

6. Experiment with other properties of the knob by using the **Knob Properties** dialog box. For example, try changing the colors for the **Marker Text Color** by clicking the color box located on the **Scale** tab.

7. Click the **Cancel** button to avoid applying the changes you made while experimenting. If you want to keep the changes you made, click the **OK** button.

Customizing the Waveform Graph

The waveform graph indicator displays the two signals. To indicate which plot is the scaled signal and which is the simulated signal, you customize the plots. Complete the following steps to customize the appearance of an indicator on the front panel.

1. Move the cursor over the top of the plot legend on the waveform graph.

   Notice that while there are two plots on the graph, the plot legend displays only one plot.

2. When a double-headed arrow appears, shown in Figure 1-11, click and drag the border of the plot legend until the second plot name appears.

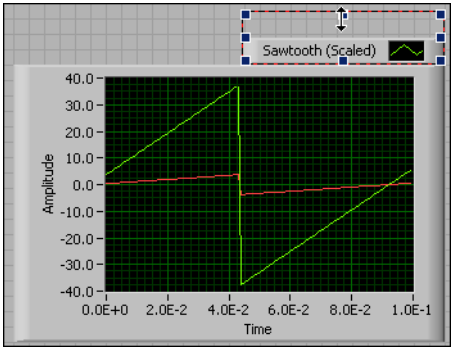


Figure 1-11. Expanding a Plot Legend

3. Right-click the waveform graph and select **Properties** from the shortcut menu to display the **Graph Properties** dialog box.
4. On the **Plots** tab, select **Sawtooth** from the pull-down menu. Click the **Line Color** color box to display the color picker. Select a new line color.
5. Select **Sawtooth (Scaled)** from the pull-down menu.
6. Place a checkmark in the **Don't use waveform names for plot names** checkbox.
7. In the **Name** text box, delete the current label and change the name of this plot to `Scaled Sawtooth`.

8. Click the **OK** button to apply the current configuration and close the **Graph Properties** dialog box.

   Notice how the plot color on the front panel changes.

9. Experiment with other properties of the graph by using the **Graph Properties** dialog box. For example, try disabling the autoscale feature located on the **Scales** tab.

10. Click the **Cancel** button to avoid applying the changes you made while experimenting. If you want to keep the changes you made, click the **OK** button.

11. Save and close this VI.

Summary

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

New Dialog Box and VI Templates

The **New** dialog box contains many LabVIEW VI templates, including the ones used in this manual. The VI templates help you start building VIs for common measurements and other tasks. The VI templates include the Express VIs, functions, and front panel objects you need to get started building common measurement applications.

You can use one of the following methods to access the **New** dialog box.

- Click the **New** button on the **LabVIEW** dialog box.
- Click the arrow on the **New** button in the LabVIEW dialog box and select **New** from the **New** pull-down menu.
- Select **File»New** from the front panel or block diagram menu bar.

Front Panel

The front panel is the user interface of a VI. You build the front panel by using controls and indicators, which are the interactive input and output terminals of the VI, respectively. Controls and indicators are located on the **Controls** palette.

Controls are knobs, push buttons, dials, and other input devices. Controls simulate the input devices on a physical instrument and supply data to the block diagram of the VI.

Indicators are graphs, LEDs, and other displays. Indicators simulate the output devices on a physical instrument and display data the block diagram acquires or generates.

Property Dialog Boxes

Use property dialog boxes or shortcut menus to configure how controls and indicators appear or behave on the front panel. Right-click a control or indicator on the front panel and select **Properties** from the shortcut menu to access the property dialog box for that object. You cannot access property dialog boxes for a control or indicator when the VI is running.

Block Diagram

The block diagram contains the graphical source code for how the VI runs. Front panel objects appear as icon terminals on the block diagram. Wires connect control and indicator terminals to Express VIs. Data flows from controls to Express VIs, from Express VIs to Express VIs, and from Express VIs to indicators through the wires.

Express VIs

Use Express VIs located on the **Functions** palette for common measurement tasks. When you place an Express VI on the block diagram, the dialog box you use to configure that Express VI appears by default. Set the options in this configuration dialog box to specify how an Express VI behaves.

Express VIs appear on the block diagram as expandable nodes with icons surrounded by a blue field. You can resize an Express VI to display its inputs and outputs. Inputs accept data, and outputs return data. The inputs and outputs for the Express VI depend on how you configure the VI.

2

Analyzing and Saving a Signal

LabVIEW includes a set of Express VIs to 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.

Building a VI from a Template

In the following exercises, you will build a VI that generates a signal, extracts the DC value of the signal, indicates if the signal exceeds a certain limit, and records the data. When you complete the exercises, the front panel of the VI will look similar to the front panel in Figure 2-1.

You can complete the exercises in approximately 40 minutes.

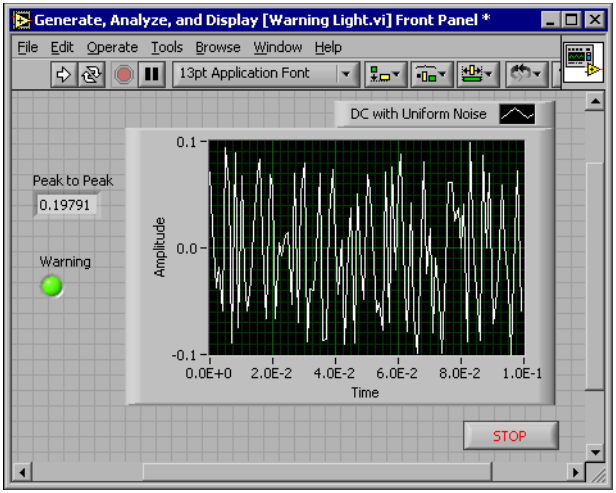


Figure 2-1. Front Panel for the Warning Light VI

Opening a New VI from a Template

To build this VI, you can start from the **New** dialog box. Complete the following steps to select a new template VI that generates, analyzes, and displays a signal.

1. In the **LabVIEW** dialog box, click the **New** button to display the **New** dialog box.

Note You also can access the **New** dialog box by clicking the arrow on the **New** button and selecting **New** from the **New** pull-down menu or by selecting **File»New** from the front panel or block diagram menu bar.

2. Select the **VI from Template»Tutorial (Getting Started)»Generate, Analyze, and Display** template in the **Create new** list.

   The template VI simulates a signal and analyzes it for its root mean square (RMS) value.

3. Click the **OK** button to open the template. You also can double-click the name of the template VI in the **Create new** list to open the template.
4. Display the block diagram by pressing the <Ctrl-E> keys.
5. If the **Context Help** window, shown in Figure 2-2, is not visible, select **Help»Show Context Help** from the block diagram menu bar to display the **Context Help** window.

Note You also can press the <Ctrl-H> keys to display the **Context Help** window.

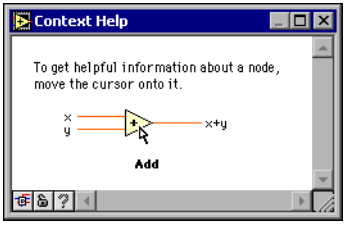
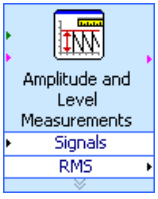


Figure 2-2. Context Help Window



6. Move the cursor over the Amplitude and Level Measurements Express VI, shown at left.

   Notice when you move the cursor over the Express VI, the **Context Help** window displays information about the Express VI, including information about how it is configured.

Keep the **Context Help** window open and notice how it provides useful information as you complete the rest of this exercise.

Modifying the Block Diagram

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 change the simulated signal from a sine wave to a DC signal with uniform white noise.

1. Right-click the Simulate Signal Express VI and select **Properties** from the shortcut menu to display the **Configure Simulate Signal** dialog box.
2. Select **DC** from the **Signal type** pull-down menu.
3. Place a checkmark in the **Add noise** checkbox to add noise to the DC signal.
4. Type `0.1` in the **Noise amplitude** text box.

   Notice that the **Result Preview** section displays a random signal. The **Configure Simulate Signal** dialog box should appear similar to Figure 2-3.

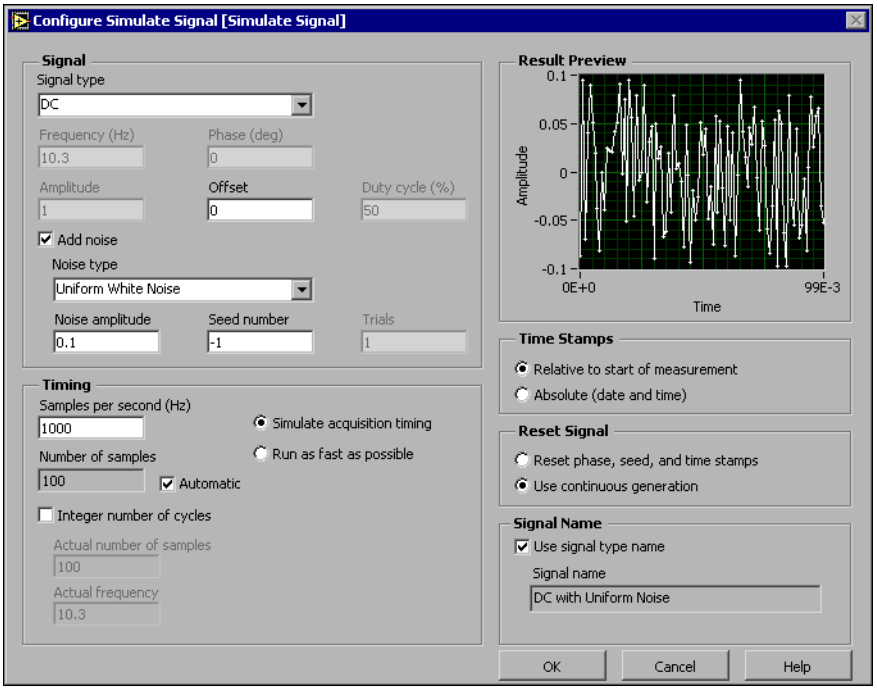


Figure 2-3. Configure Simulate Signal Dialog Box

5. Click the **OK** button to save the current configuration and close the **Configure Simulate Signal** dialog box.
6. Display the front panel by pressing the <Ctrl-E> keys.
7. Run the VI.

   Notice that the signal appears in the graph and the RMS value for the signal appears in the digital indicator.
8. Click the **STOP** button.
9. Select **File»Save As** and save this VI as `Analysis.vi` to an easily accessible location.

Modifying the Front Panel

If you decide that you do not want to use an indicator that comes with the template, you can delete it. Complete the following steps to remove the **RMS** indicator from the front panel.

1. Move the cursor over the **RMS** indicator until the Positioning tool appears.

2. Click the **RMS** indicator, shown at left, to select it and press the <Delete> key.
3. Display the block diagram.

   The block diagram now has a dashed wire with a red X, shown at left. This is a broken wire. Also notice the **Run** button, shown at left, appears broken to indicate the VI cannot run.

4. Click the broken **Run** button to display the **Error list** window.

   The **Error list** window lists all errors in the VI and provides details about each error.
5. In the **Errors and Warnings** listbox, double-click the **`Wire: has loose ends`** error to highlight the broken wire.

   Notice how LabVIEW automatically displays the problem causing the error.
6. Press the <Delete> key to delete the broken wire.

Tip Press the <Ctrl-B> keys to delete *all* broken wires on the block diagram.

7. Select **Windows»Show Error List** to display the **Error list** window. Now there are no errors listed in the **Errors and Warnings** listbox.

Note You also can press the <Ctrl-L> keys to display the **Error list** window.

8. Click the **Close** button to close this window.

   Notice that the **Run** button is no longer broken.

Analyzing the Amplitude of a Signal

The Amplitude and Level Measurements Express VI includes options that you can use 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. Right-click the Amplitude and Level Measurements Express VI and select **Properties** from the shortcut menu to display the **Configure Amplitude and Level Measurements** dialog box.

Tip You also can double-click the Express VI to open the **Configure Amplitude and Level Measurements** dialog box.

2. Remove the checkmark in the **RMS** checkbox in the **Amplitude Measurements** section.

3. Click the **Help** button, shown at left, in the bottom right corner of the **Configure Amplitude and Level Measurements** dialog box to display the *LabVIEW Help* topic for this Express VI.

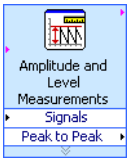
   The help topic describes the Express VI, the inputs and outputs for the Express VI, and the configuration options. Each Express VI has a corresponding help topic accessible by pressing the **Help** button.

4. In the *Amplitude and Level Measurements* topic, find the output parameter whose description indicates that it takes a measurement from the maximum peak to the minimum peak of the signal.

5. Minimize *LabVIEW Help* to return to the **Configure Amplitude and Level Measurements** dialog box.

6. Select the input or output you decided to use.

   Notice how the option you selected, **Peak to Peak**, appears in the **Results** section with the corresponding value of the measurement.



7. Click the **OK** button to close the **Configure Amplitude and Level Measurements** dialog box and return to the block diagram.

   Notice that the RMS output in the Amplitude and Level Measurements Express VI changed to reflect the new **Peak to Peak** parameter, shown at left.

Adding a Warning Light

If you want a visual cue indicating when a value exceeds a specified limit, use a warning light. Complete the following steps to add a warning light to the VI.

1. From the **Controls** palette, select the round LED indicator on the **LEDs** palette, shown in Figure 2-4, and place it on the front panel to the left of the waveform graph.

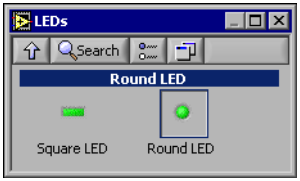


Figure 2-4. LEDs Palette

2. Right-click the LED and select **Properties** from the shortcut menu to display the **Button Properties** dialog box.
3. Change the label of the LED to `Warning`.
4. Click the **OK** button to save the current configuration and close the **Button Properties** dialog box.

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

5. Select **File»Save As** and save this VI as `Warning Light.vi` to an easily accessible location.

Setting the Warning Level Limit

To specify the value at which you want the warning light to turn on, 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, select the Comparison Express VI on the **Arithmetic & Comparison»Express Comparison** palette and place it to the right of the Amplitude and Level Measurements Express VI.
2. In the **Configure Comparison** dialog box, select the **> Greater than** option from the **Compare Condition** section.
3. In the **Comparison Inputs** section, select **Use constant value** and type `0.195` in the **Constant value** text box.
4. Close the configuration page and return to the block diagram.

   Notice how the name of the Comparison Express VI reflects the operation of the Express VI, shown at left. **Greater Than** 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 Comparison Express VI.
6. Move the cursor over the wire connecting the **Peak to Peak** output to the **Operand 1** input.
7. When the Positioning tool appears, right-click the wire connecting the **Peak to Peak** output to the **Operand 1** input and select **Create» Numeric Indicator** from the shortcut menu.

   Notice how a **Peak to Peak** terminal, shown at left, appears on the block diagram. If the **Peak to Peak** terminal appears to be on top of the wires of 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 the blank space above the Express VIs.

Warning the User

After specifying the values at which you want the warning light to turn on, you must wire the warning light to the Comparison Express VI. Complete the following steps to provide a visual cue to the user when the peak to peak value of the signal exceeds a specified limit.

1. On the block diagram, move the **Warning** terminal to the right of the Comparison Express VI. Make sure the **Warning** terminal is inside the loop, as shown in Figure 2-5.
2. Wire the **Result** output of the Comparison Express VI to the **Warning** terminal.

   The block diagram should appear similar to Figure 2-5.

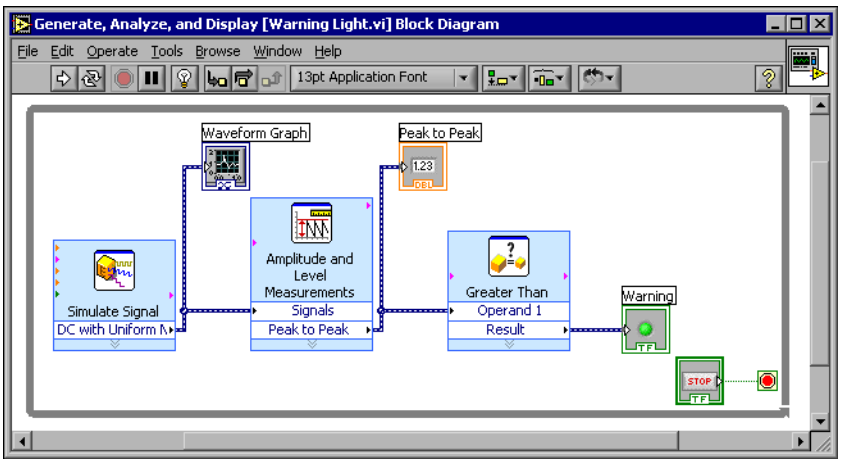


Figure 2-5. Block Diagram for the Warning Light VI

3. Display the front panel.

   Notice how a numeric indicator also appears on the front panel labeled **Peak to Peak**. This indicator displays the peak to peak value of the signal.
4. Run the VI.

   Notice that when the peak to peak value exceeds 0.195, the **Warning** indicator lights up.
5. Click the **STOP** button to stop the VI.
6. Select **File»Save** to save this VI.

Configuring the VI to Save Data to a File

To store information about the data your VI generates, use the Write LabVIEW 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. Select the Write LabVIEW Measurement File Express VI on the **Output** palette and place it on the block diagram below and to the right of the Amplitude and Level Measurements Express VI.

   Notice that the **File name** text box indicates that the output file is `test.lvm` and displays the full path to the `test.lvm` file. A `.lvm` file is a LabVIEW measurement data file, which LabVIEW places 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 **File name** text box to access the `test.lvm` file.

2. In the **Configure Write LabVIEW Measurement File** dialog box, select the **Append to file** option in the **If a file already exists** section.

   By selecting **Append to file**, LabVIEW writes all the data to the `test.lvm` file without erasing the existing data in the file if a file by that name exists already.

3. Select the **One header only** option in the **Segment Headers** section.

4. Enter the following text in the **File Description** text box: `Sample of peak to peak values.`

5. Close the **Configure Write LabVIEW Measurement File** dialog box and return to the block diagram.

Saving Data to a File

When you run this VI, LabVIEW saves the data to the `test.lvm` file. Complete the following steps to generate the `test.lvm` file.

1. Wire the **Peak to Peak** output of the Amplitude and Level Measurements Express VI to the **Signals** input of the Write LabVIEW Measurement File Express VI.

2. Select **File»Save As** and save this VI as `Save Data.vi` to an easily accessible location.

3. Display the front panel and run the VI.

4. Click the **STOP** button on the front panel.

5. To view the data you saved, open the `LabVIEW Data\test.lvm` file with a spreadsheet or word processing application.
6. Close the file when you finish looking at it and return to the Save Data VI.

Adding a Button that Stores Data when Pressed

If you want to store only certain data, you can configure the Write LabVIEW Measurement File Express VI to save peak to peak values only when a user presses 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. On the front panel, select the rocker button on the **Buttons & Switches** palette and place it to the right of the waveform graph.
2. Using the **Button Properties** dialog box, change the label of the button to `Write to File`.
3. On the **Operation** tab, select **Latched When Pressed** from the **Button Behavior** list.

   Use the **Operation** tab 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.
4. Close the **Button Properties** dialog box.
5. Save this 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. On the block diagram, double-click the Write LabVIEW Measurement File Express VI to access the **Configure Write LabVIEW Measurement File** dialog box.
2. Change the file name `test.lvm` to `Selected Samples.lvm` in the **File name** text box to save the data to a different file.
3. Close the **Configure Write LabVIEW Measurement File** dialog box.
4. Right-click the **Signal** input of the Write LabVIEW Measurement File Express VI. Select **Insert Input/Output** from the shortcut menu to insert the **Comment** input.

5. Right-click the **Comment** input of the Write LabVIEW Measurement File Express VI. Select **Select Input/Output»Enable** from the shortcut menu to insert the **Enable** input.

   In the previous exercise you learned to add inputs and outputs by expanding the Express VI using the down arrows. Notice that this method is a different way of displaying and selecting the inputs and outputs of an Express VI.

   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 may need to add an input first, then change the input to the specific one you want to use.

6. Move the **Write to File** terminal to the left of the Write LabVIEW Measurement File Express VI.

7. Wire the **Write to File** terminal to the **Enable** input of the Write LabVIEW Measurement File Express VI.

   The block diagram should appear similar to Figure 2-6.

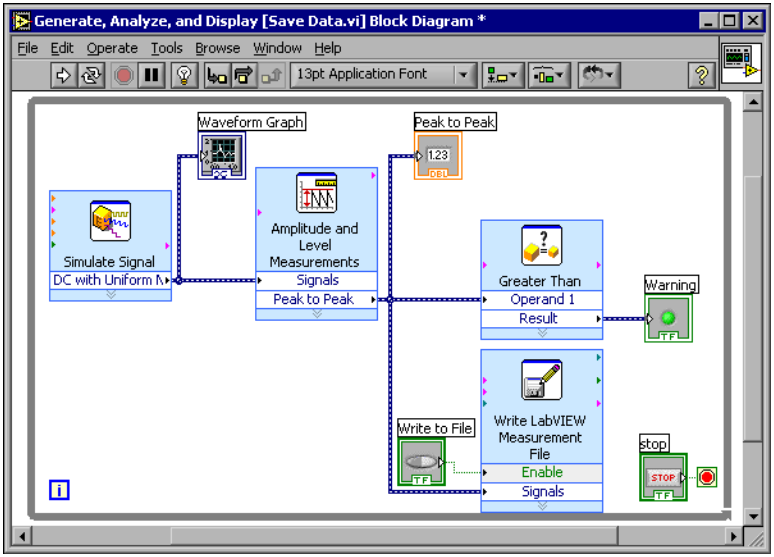


Figure 2-6. Block Diagram for the Save Data VI

8. Display the front panel and run the VI. Click the **Write to File** button several times.

9. Click the **STOP** button on the front panel.

10. To view the data you saved, open the `Selected Samples.lvm` file with a spreadsheet or word processing application.

    Notice how 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` only recorded the data when you pressed the **Write to File** button.

11. Save and close this VI.