

# Getting Started with LabVIEW Virtual Instruments

---

## Approximate Time

You can complete this exercise in approximately 30 minutes.

## Background

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 code you write.

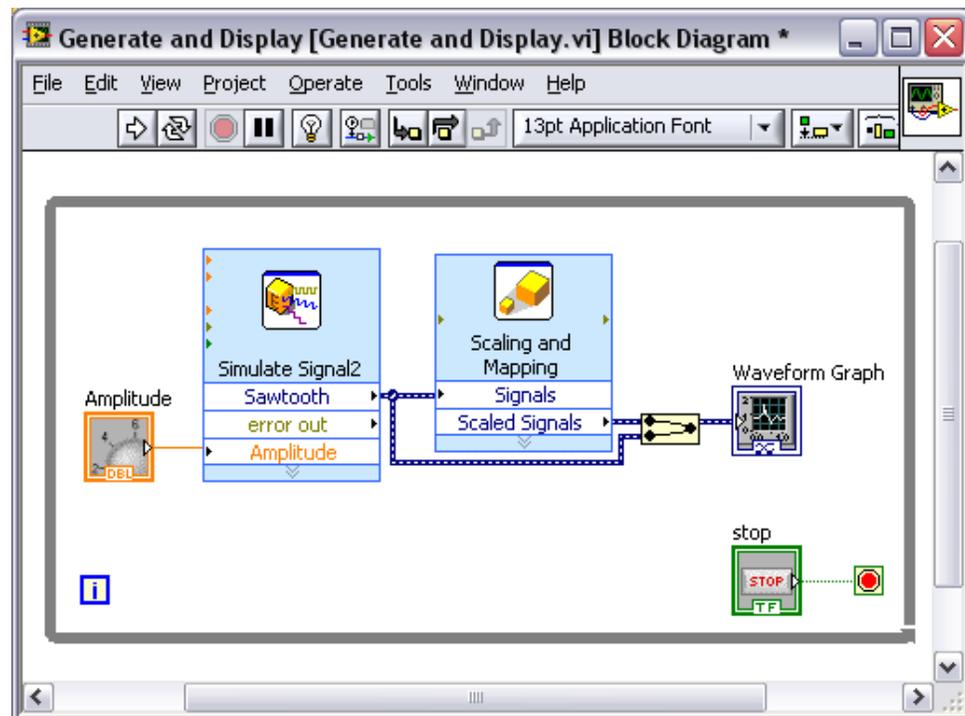
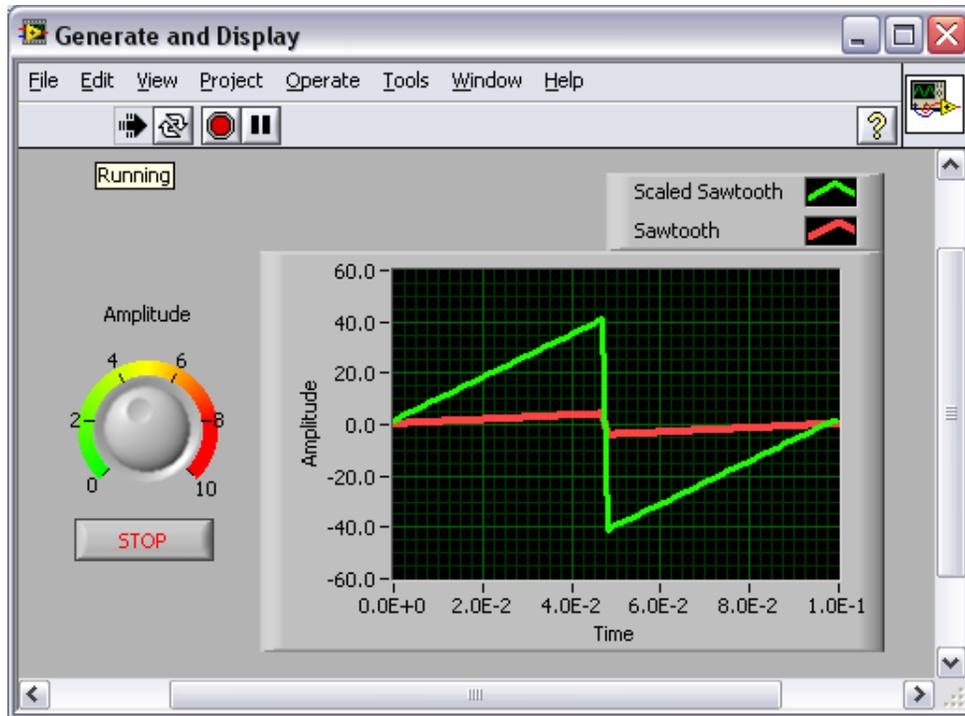
In LabVIEW, you build a user interface, or front panel, with controls and indicators. Controls are knobs, push buttons, dials, and other input mechanisms. Indicators are graphs, LEDs, and other output displays. After you build the user interface, you add code using VIs and structures to control the front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

You can use LabVIEW to communicate with hardware such as data acquisition, vision, and motion control devices, as well as GPIB, PXI, VXI, RS232, and RS485 instruments.

## Overview

In the following exercise, you will build a VI that generates a signal and displays that signal in a graph. You will build and customize a user interface that allows you to control the amplitude of the signal and easily display your data.

# Completed Exercise



# Step-by-Step Instructions

---

## 1. Launching LabVIEW

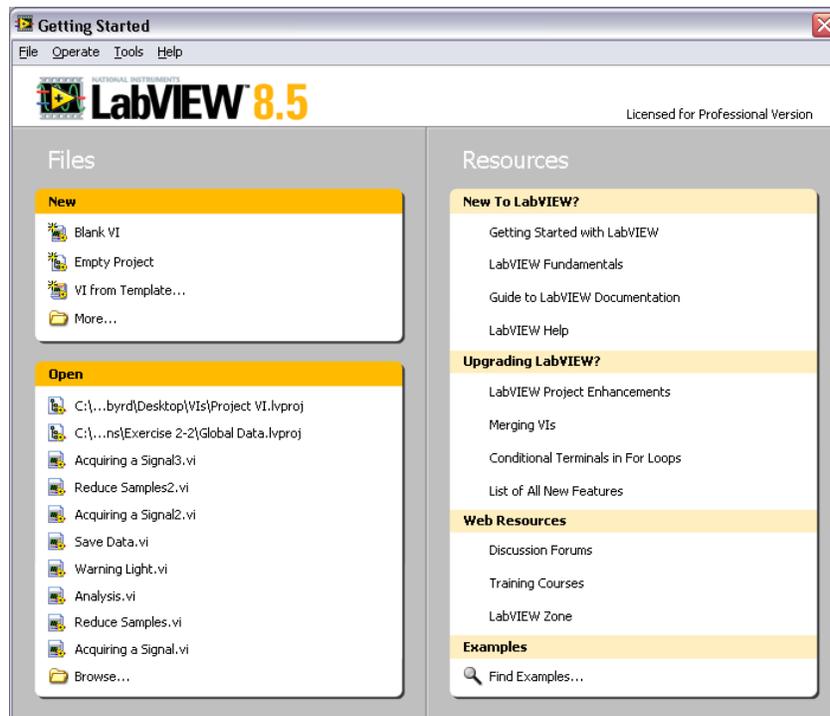
The **Getting Started** window appears when you launch LabVIEW. You can open the LabVIEW Getting Started window by doing either of the following:

- Double-click the LabVIEW icon on your desktop, or
- In the Windows **Start** menu, navigate to **Programs » National Instruments »**

**LabVIEW 8.5 » LabVIEW**



The **Getting Started** window is used to create new VIs, view the most recently opened LabVIEW files, find examples, and launch the *LabVIEW Help*. You also can access information and resources to help you learn about LabVIEW, such as specific manuals, help topics, and resources on the National Instruments Web site, ni.com.

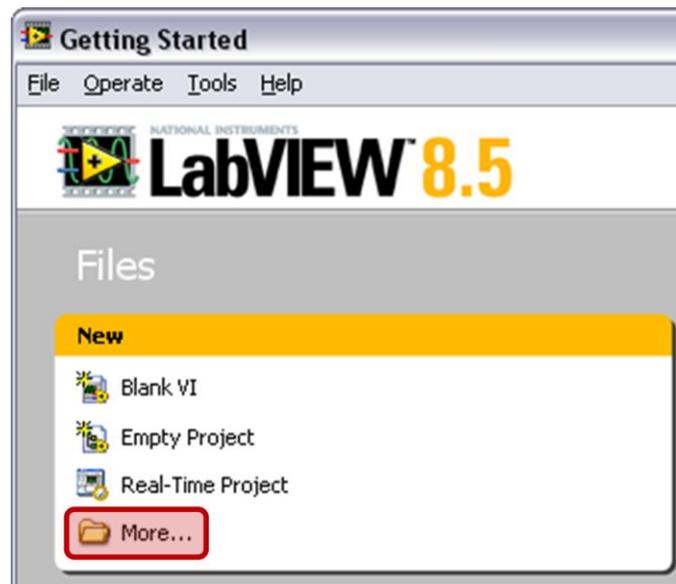


The **Getting Started** window disappears when you open a VI and reappears when you close all open files. You can also display the window from an open VI by selecting **View » Getting Started Window**.

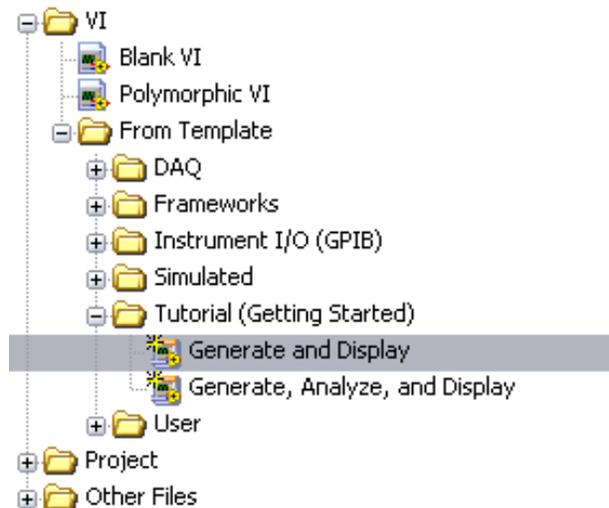
## 2. Opening a New VI from a Template

LabVIEW provides built-in template VIs that include the functions, structures, and front panel objects you need to get started building common measurement applications. Complete the following steps to create a VI that generates a signal and displays it on the front panel:

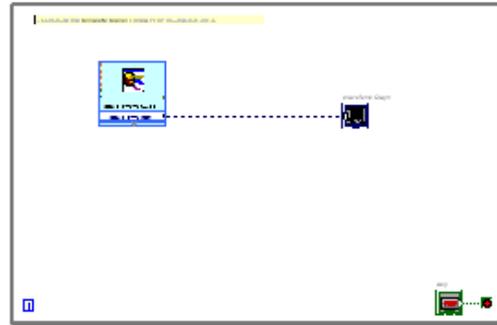
- a. In the **Getting Started** window, click the **More...** link to display the **New** dialog box.



- b. From the **Create New** list, select **VI » From Template » Tutorial (Getting Started) » Generate and Display**. This template VI generates and displays a signal.



- c. A preview and a brief description of the template VI appear in the **Description** section. The following figure shows the **New** dialog box and the preview of the Generate and Display template VI.



Use this template to simulate the generation and display of data.

Note: LabVIEW enables automatic error handling by default for VIs you create from this template.

- d. Click the **OK** button to create a VI from the template.
- e. LabVIEW displays two windows: the Front panel window (gray background) and the block diagram window (white background). Examine the Front panel window.



The user interface, or Front panel, includes controls and indicators. The title bar indicates that this window is the Front panel for the Generate and Display VI.



You can switch between the front panel window and block diagram window at any time by pressing the <Ctrl-E> keys. You can also select **Window » Show Front Panel** or **Window » Show Block Diagram** to switch between the two.

- f. Select **Window » Show Block Diagram** and 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 indicates that this window is the block diagram for the Generate and Display VI.

- g. On the front panel toolbar, click the **Run** button. You also can press the <Ctrl-R> keys to run a VI. A sine wave appears on the front panel graph.



- h. Stop the VI by clicking the **STOP** button, on the front panel.



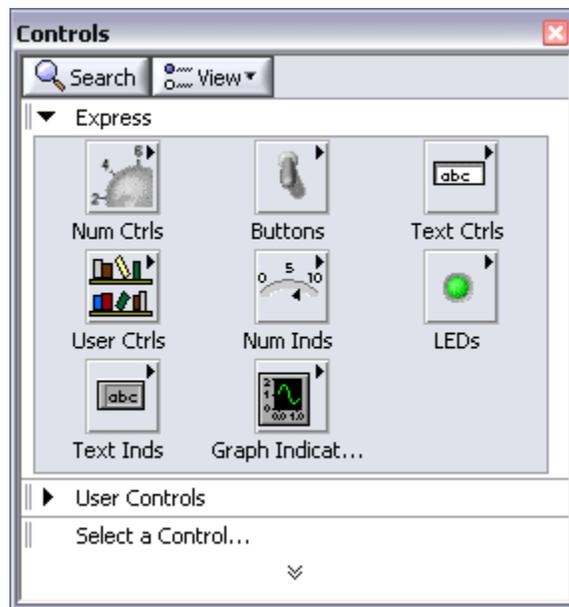
### 3. Adding a Control to the Front Panel

Controls on the front panel simulate the input mechanisms 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:



Throughout the exercise, you can undo recent edits by selecting **Edit » Undo** from the toolbar or pressing the <Ctrl-Z> keys.

- a. If the **Controls** palette is not visible on the front panel select **View » Controls Palette** from the toolbar.



- b. You can right-click any blank space on the front panel to display a temporary version of the **Controls** palette. The **Controls** palette appears with a thumbtack icon in the upper left corner. Click the thumbtack to pin the palette so it is not temporary.



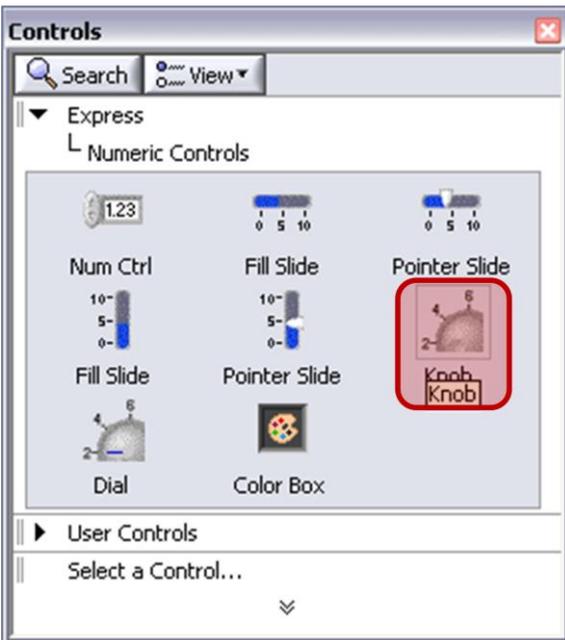
- c. The **Controls** palette opens with the **Express** subpalette visible by default. Click the double arrows at the bottom of the **Controls** palette if this subpalette is not visible.



- d. Move your cursor over the icons on the **Express** subpalette. Notice that the name of each subpalette, control, or indicator appears in a tip strip below the icon.



- e. Click the **Numeric Controls** icon to display the **Numeric Controls** palette.

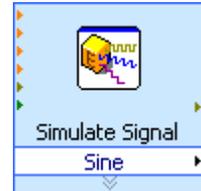


- f. Left-click the knob control on the **Numeric Controls** palette to attach the control to the cursor
- g. Place the knob on your front panel to the left of the waveform graph. You will use this knob in a later exercise to control the amplitude of a signal.
- h. Select **File » Save As** and save the VI to the desktop as **Acquiring a Signal.vi**

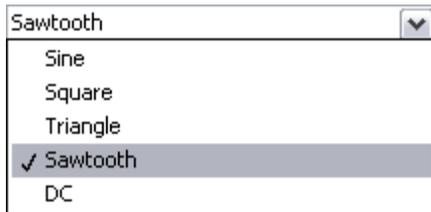
## 4. Changing a Signal Type

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

- a. Display the block diagram by pressing the <Ctrl-E> keys or clicking the block diagram.
- b. Locate the Simulate Signal Express VI. 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 specified.

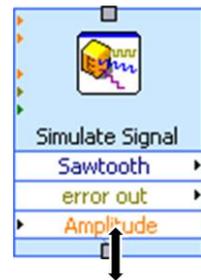


- c. Double-click the Simulate Signal Express VI to display the **Configure Simulate Signal** dialog box. This can also be done by right-clicking the VI and selecting **Properties**.



- d. Select **Sawtooth** from the **Signal type** pull-down menu. Notice that the waveform graph in the **Result Preview** changes to a sawtooth wave.
- e. Click the **OK** button to save the current configuration and close the dialog box.

- f. Move the cursor over the down arrows at the bottom of the Simulate Signal Express VI. When the cursor changes to a double-headed arrow and “handles” appear on the VI, click and drag the bottom handle down to display hidden inputs and outputs.
- g. When you release the mouse, the **Amplitude** input appears. You can configure the amplitude of the sawtooth wave programmatically now that it appears on the block diagram.

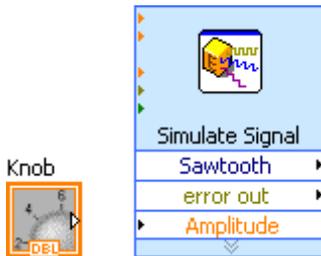


**Amplitude** is also an option in the **Configure Simulate Signal** dialog box. You can configure several inputs for Express VIs in either location.

## 5. Wiring Objects on the Block Diagram

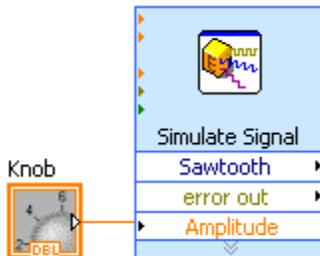
To use the knob to change the amplitude of the signal, you must connect two objects on the block diagram. Complete the following steps to wire the knob to the **Amplitude** input of the Simulate Signal Express VI:

- a. On the block diagram, move the cursor over the **Knob** terminal.
- b. The cursor becomes an arrow, or the Positioning tool. Use the Positioning tool to select, position, and resize objects.



- c. Use the Positioning tool to select the **Knob** terminal and make sure it is to the left of the Simulate Signal Express VI and inside the gray loop.
- d. Deselect the **Knob** terminal by clicking a blank space on the block diagram.

- e. Move the cursor over the arrow on the right-hand side of the **Knob** terminal.
- f. The cursor becomes a wire spool, or the Wiring tool. The Wiring tool is used to connect objects together on the block diagram.



- g. When the Wiring tool appears, click the arrow on the **Knob** terminal and then click the arrow on the **Amplitude** input of the Simulate Signal Express VI, this wires the objects together.
- h. A wire appears and connects the two objects. Data flows along this wire from the **Knob** Terminal to the Express VI.

- i. Select **File » Save** to save the VI.

## 6. Running a VI

What you have done so far is add a knob control that allows you to programmatically change the amplitude of a Sawtooth wave. Running a VI executes the LabVIEW graphical code you just created. Complete the following steps to run the Acquiring a Signal VI:

- a. Display the front panel by pressing the <Ctrl-E> keys or by clicking the front panel.
- b. Click the **Run** button or press the <Ctrl-R> keys to run the VI.
- c. Move the cursor over the knob control.
- d. The cursor becomes a hand, or the Operating tool. Use the Operating tool to change the value of a control.
- e. Using the Operating tool, turn the knob to adjust the amplitude of the sawtooth wave.



The amplitude of the sawtooth wave changes as you turn the knob. The y-axis on the graph scales automatically to account for the change in amplitude when autoscale is selected.

- f. To indicate that the VI is running, the **Run** button changes to a darkened arrow. You can change the value of the controls while a VI runs, but you cannot edit the VI.



- g. Click the **STOP** button to stop the VI. The **STOP** button stops the VI after the VI completes the current iteration.



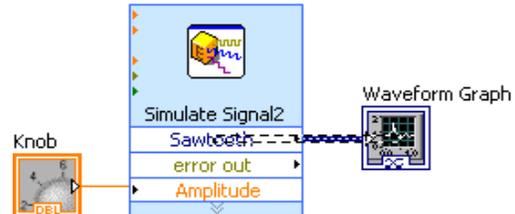
The **Abort Execution** button in the toolbar stops the VI instantly, before the VI finishes the current iteration. Aborting a VI that uses external resources, such as external hardware, might leave the resources in an unknown state. Design VIs with a Stop button to avoid this problem.



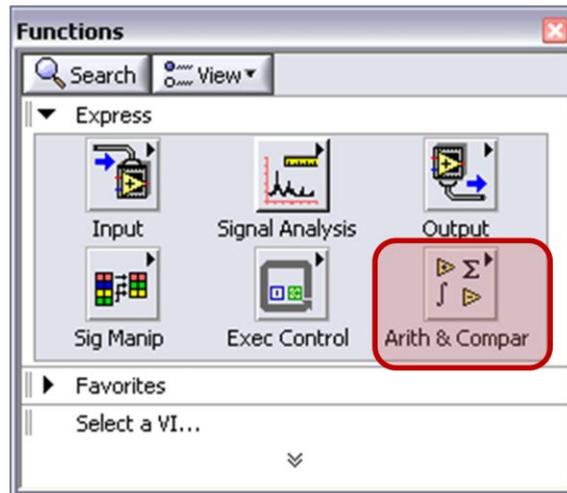
## 7. Modifying a Signal

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

- On the block diagram, left-click the wire that connects the Simulate Signal Express VI to the Waveform Graph to highlight the wire.



- Press the <Delete> key to remove this wire.
- If the **Functions** Palette is not visible, select **View » Functions Palette** to display it or simply right-click on the block diagram.



The **Functions** Palette opens with the **Express** subpalette visible by default. If you have selected another subpalette, you can return to the **Express** subpalette by clicking **Express** on the **Functions** palette.

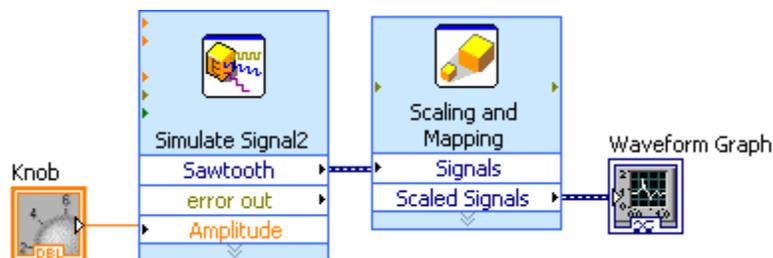
- On the **Arithmetic & Comparison** subpalette, select the Scaling and Mapping Express VI. Place it on the block diagram inside the loop between the Simulate Signal Express VI and the **Waveform Graph** terminal. Move the **Waveform Graph** terminal if necessary.



- e. The **Configure Scaling and Mapping** dialog box opens when you place the Express VI on the block diagram. Define the scaling factor by entering **10** in the Slope (m) text box.
- f. Click the **OK** button to save the current configuration and close the **Configure Scaling and Mapping** dialog box.



- g. Move the cursor over the arrow on the **Sawtooth** output of the Simulate Signal Express VI. When the Wiring tool appears, click the arrow on the **Sawtooth** output and then click the arrow on the **Signals** input of the Scaling and Mapping Express VI to wire the two objects together.
- h. Use the Wiring tool to wire the Scaled Signals output of the Scaling and Mapping Express VI to the Waveform Graph terminal.

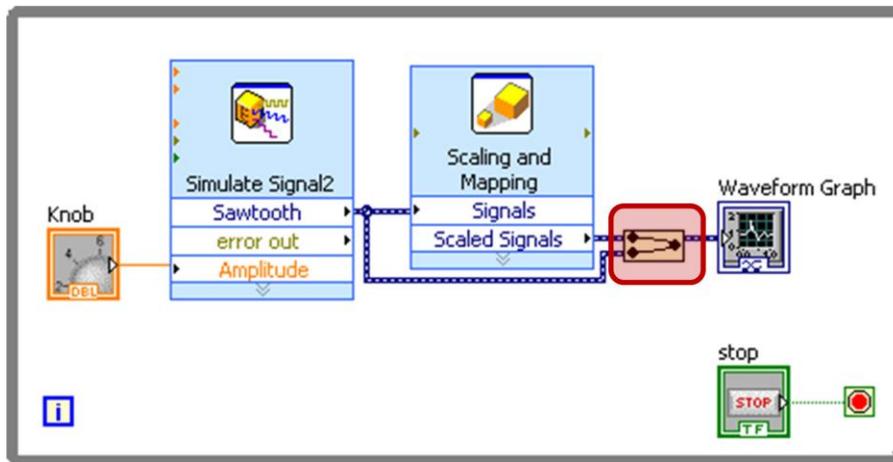


- i. Examine the wires connecting the Express VIs and terminals. The arrows on the Express VIs and terminals indicate the direction of data flow along these wires.

## 8. Displaying Two Signals on Your Graph

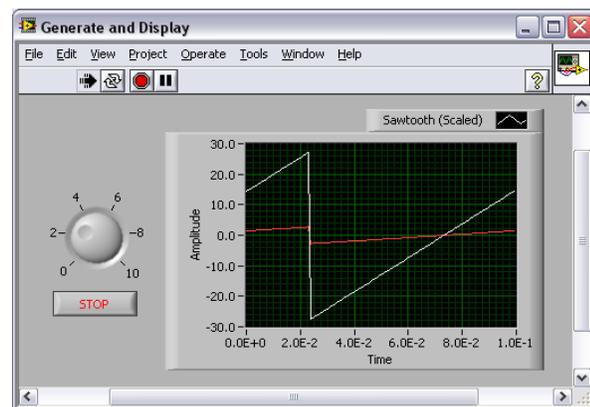
To compare the signal generated by the Simulate Signal Express VI and the signal modified by the Formula Express VI on the same graph, use the Merge Signals function. Complete the following steps to display two signals on the same graph:

- Wire the **Sawtooth** output to the **Waveform Graph** terminal.
- The Merge Signals function automatically appears where the two wires connect.



A function is a built-in execution element, comparable to an operator or statement in a text-based programming language. The Merge Signals function takes two signals and combines them so that both can display on the same graph. Be sure to start the wiring at the **Sawtooth** output to avoid creating a feedback node.

- Press the <Ctrl-S> keys or select **File » Save** to save the VI.
- From the front panel, run the VI, and turn the knob control.
- Click the **STOP** button to stop the VI.

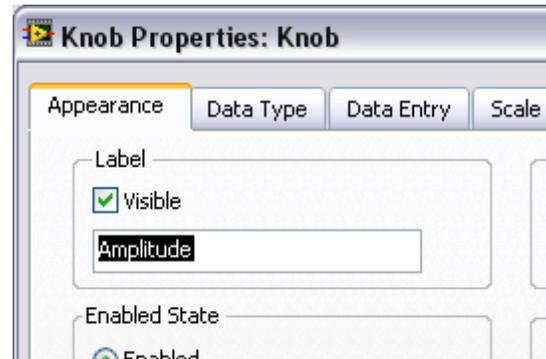


The graph plots the wave and the scaled wave. The maximum value on the y-axis automatically changes to be 10 times the knob value. This scaling occurs because you configured the Scaling and Mapping Express VI to generate a slope of 10.

## 9. Customizing a Knob Control

The knob control changes the amplitude of the sawtooth wave, so labeling it *Amplitude* accurately describes the behavior of the knob. Complete the following steps to customize the appearance of the knob:

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



- c. Click the **Scale** tab and in the **Scale Style** section, place a checkmark in the **Show color ramp** checkbox.



- d. Click the **OK** button to save the current configuration and close the **Knob Properties** dialog box. The knob on the front panel updates to reflect these changes.
- e. Save the VI.
- f. Reopen the **Knob Properties** dialog box and experiment with other properties of the knob. For example, on the **Scale** page, try changing the colors for the **Marker text color** by clicking the color box.



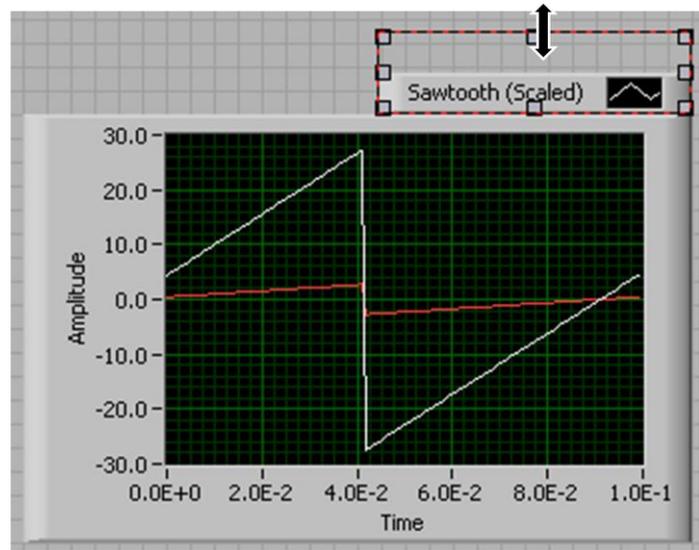
Click the **Cancel** button to avoid applying any changes you made while experimenting. If you want to keep the changes you made, click the **OK** button.

## 10. Customizing a 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 can customize the plots. Complete the following steps to customize the appearance of the waveform graph indicator:

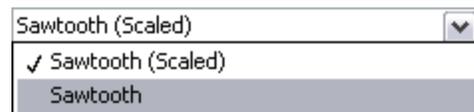
- a. Though the graph has two plots, the legend displays only one plot. On the front panel, move the cursor over the top of the plot legend on the waveform graph.

- b. When the cursor changes to a double-headed arrow and “handles” appear, click and drag the top handle of the legend to add one item. When you release the mouse button, the second plot name appears.

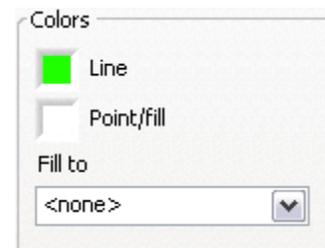


- c. Right-click the waveform graph and select **Properties** from the shortcut menu to display the **Waveform Graph Properties** dialog box.

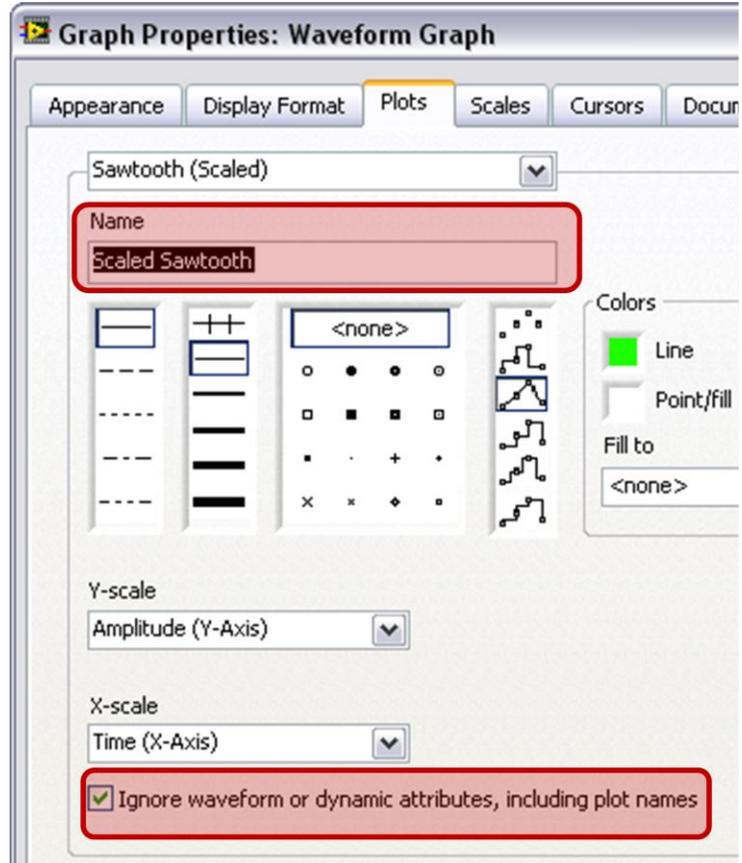
- d. On the **Plots** page, select **Sawtooth** from the pull-down menu.



- e. In the **Colors** section, click the **Line** color box to display the color picker. Select a new line color.



- f. Select **Sawtooth (Scaled)** from the pull-down menu.
- g. Place a checkmark in the **Ignore Waveform or Dynamic Attributes, Including Plot names** checkbox at the bottom of the dialog.
- h. In the **Name** text box, delete the current label and change the name of this plot to *Scaled Sawtooth*.



The reason for placing a checkmark in the **Ignore Waveform or Dynamic Attributes, Including Plot names** checkbox is because this will display *Scaled Sawtooth* in the legend of the Waveform Graph instead of *Sawtooth (Scaled)* which would be scaled without placing that checkmark.

- i. Click the **OK** button to save the current configuration and close the **Waveform Graph Properties** dialog box.
- j. Reopen the **Waveform Graph Properties** dialog box and experiment with other properties of the graph. For example, on the **Scales** page, try disabling automatic scaling and changing the minimum and maximum value of the y-axis.



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

- k. Run the VI. Notice the changes made.
- l. Save and close your VI.