LabView Exercises[1]

Exercise 1 - Open and Run a Virtual Instrument

Examine the **Signal Generation and Processing** VI and run it. Change the frequencies and types of the input signals and notice how the display on the graph changes. Change the Signal Processing Window and Filter options. After you have examined the VI and the different options you can change, stop the VI by pressing the Stop button.

1. Select **Start»Programs»National Instruments»LabVIEW 7.0»LabVIEW** to launch LabVIEW. The **LabVIEW** dialog box appears.

2. Select **Help»Find Examples.** The dialog box that appears lists and links to all available LabVIEW example VIs.

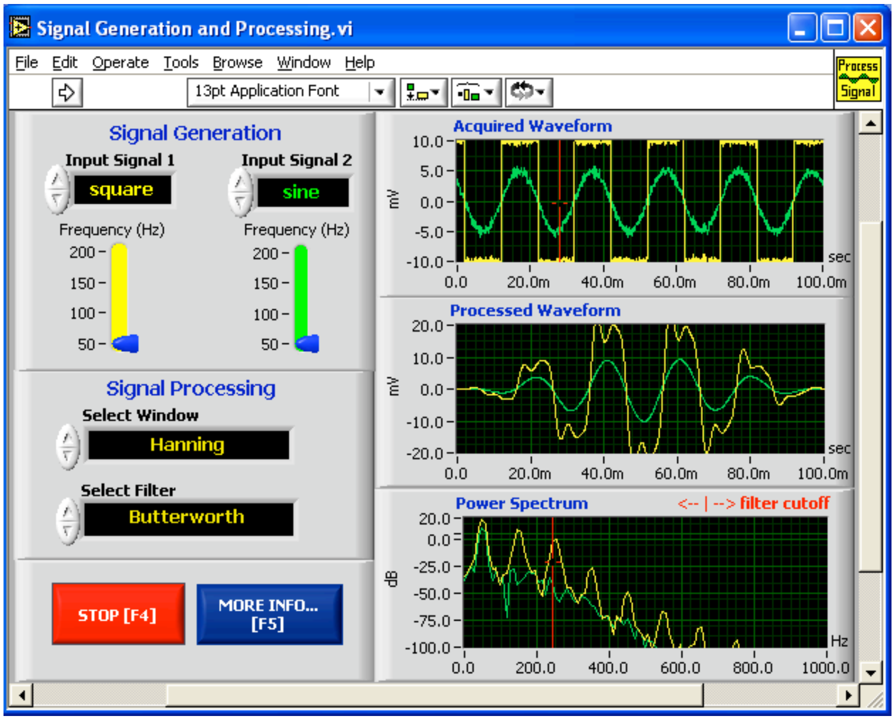
3. On the Browse Tab, select browse according to task. Choose **Analyzing and Processing Signals**, then **Signal Processing**, then **Signal Generation and Processing.vi.** This will open the Signal Generation and Processing VI Front Panel.

Note You also can open the VI by clicking the **Open VI** button and navigating to labview\examples\apps\demos.llb\Signal Generation and Processing.vi.

Front Panel

4. Click the **Run** button on the toolbar, shown at left, to run this VI.
This VI determines the result of filtering and windowing a generated signal. This example also displays the power spectrum for the generated signal. The resulting signals are displayed in the graphs on the front panel, as shown in the following figure.

[1] Ref. [1-3]



5. Use the Operating tool, shown at left, to change the Input Signal and the Signal Processing, use the increment or decrement arrows on the control, and drag the pointer to the desired Frequency.

6. Press the **More Info…** button or [F5] to read more about the analysis functions.

7. Press the **Stop** button or [F4] to stop the VI.

Block Diagram

8. Select **Window»Show Diagram** or press the <Ctrl-E> keys to display the block diagram for the Signal Generation and Processing VI.

(**MacOS**) Press the <Command-E> keys. (**Sun**) Press the <Meta-E> keys. (**Linux**) Press the <Alt-E> keys.

This block diagram contains several of the basic block diagram elements, including subVIs, functions, and structures, which you will learn about later in this course.

9. Select **Window»Show Panel** or press the <Ctrl-E> keys to return to the Front Panel.

10. Close the VI and do not save changes.

End of Exercise

Exercise 2 – Convert °C to °F

Build a VI

Complete the following steps to create a VI that takes a number representing degrees Celsius and converts it to a number representing degrees Fahrenheit.

In wiring illustrations, the arrow at the end of this mouse icon shows where to click and the number on the arrow indicates how many times to click.

Front Panel

1. Select **File»New** to open a new front panel.

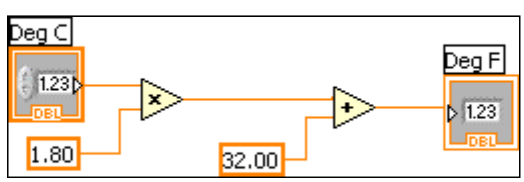
2. (Optional) Select **Window»Tile Left and Right** to display the front panel and block diagram side by side.
3. Create a numeric digital control. You will use this control to enter the value for degrees Centigrade.
    a. Select the digital control on the **Controls»Numeric Controls** palette. If the **Controls** palette is not visible, right-click an open area on the front panel to display it.
    b. Move the control to the front panel and click to place the control.
    c. Type `deg C` inside the label and click outside the label or click the **Enter** button on the toolbar. If you do not type the name immediately, LabVIEW uses a default label. You can edit a label at any time by using the Labeling tool.
4. Create a numeric digital indicator. You will use this indicator to display the value for degrees Fahrenheit.
    a. Select the digital indicator on the **Controls»Numeric Indicators** palette.
    b. Move the indicator to the front panel and click to place the indicator.
    c. Type `deg F` inside the label and click outside the label or click the **Enter** button.

   LabVIEW creates corresponding control and indicator terminals on the block diagram. The terminals represent the data type of the control or indicator. For example, a DBL terminal represents a double-precision, floating-point numeric control or indicator.

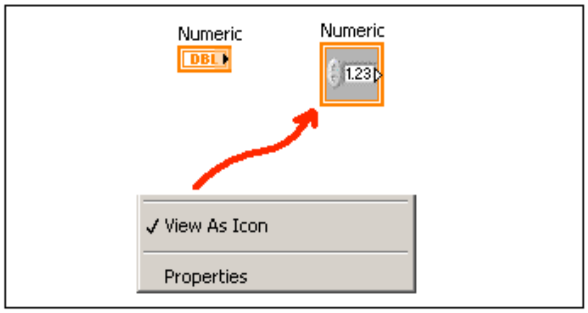
Note Control terminals have a thicker border than indicator terminals.

Block Diagram

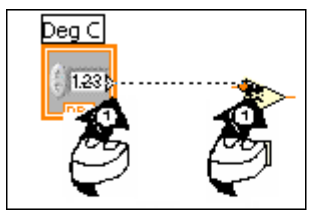
5. Display the block diagram by clicking it or by selecting **Window»Show Diagram**.



Note: Block Diagram terminals can be viewed as icons or as terminals. To change the way LabVIEW displays these objects right click on a terminal and select **View As Icon**.



6. Select the Multiply and Add functions on the **Functions»Numeric** palette and place them on the block diagram. If the **Functions** palette is not visible, right-click an open area on the block diagram to display it.
7. Select the numeric constant on the **Functions»Numeric** palette and place two of them on the block diagram. When you first place the numeric constant, it is highlighted so you can type a value.
8. Type `1.8` in one constant and `32.0` in the other.
   If you moved the constants before you typed a value, use the Labeling tool to enter the values.
9. Use the Wiring tool to wire the icons as shown in the previous block diagram.
   - To wire from one terminal to another, use the Wiring tool to click the first terminal, move the tool to the second terminal, and click the second terminal, as shown in the following illustration. You can start wiring at either terminal.



   - You can bend a wire by clicking to tack the wire down and moving the cursor in a perpendicular direction. Press the spacebar to toggle the wire direction.
   - To identify terminals on the nodes, right-click the Multiply and Add functions and select **Visible Items»Terminals** from the shortcut menu to display the connector pane. Return to the icons after wiring by right-clicking the functions and selecting **Visible Items»**
   - **Terminals** from the shortcut menu to remove the checkmark.
   - When you move the Wiring tool over a terminal, the terminal area blinks, indicating that clicking will connect the wire to that terminal and a tip strip appears, listing the name of the terminal.
   - To cancel a wire you started, press the <Esc> key, right-click, or click the source terminal.
10. Display the front panel by clicking it or by selecting **Window»Show Panel**.
11. Save the VI because you will use this VI later in the course.
    a. Select **File»Save**.
    b. Navigate to `c:\exercises\LV Intro`.

Note Save all the VIs you edit in this course in `c:\exercises\LV Intro`.

    c. Type `Convert C to F.vi` in the dialog box.
    d. Click the **Save** button.

12. Enter a number in the digital control and run the VI.
    a. Use the Operating tool or the Labeling tool to double-click the digital control and type a new number.
    b. Click the **Run** button to run the VI.
    c. Try several different numbers and run the VI again.
13. Select **File»Close** to close the Convert C to F VI.

End of Exercise

Exercise 2a – Create a SubVI

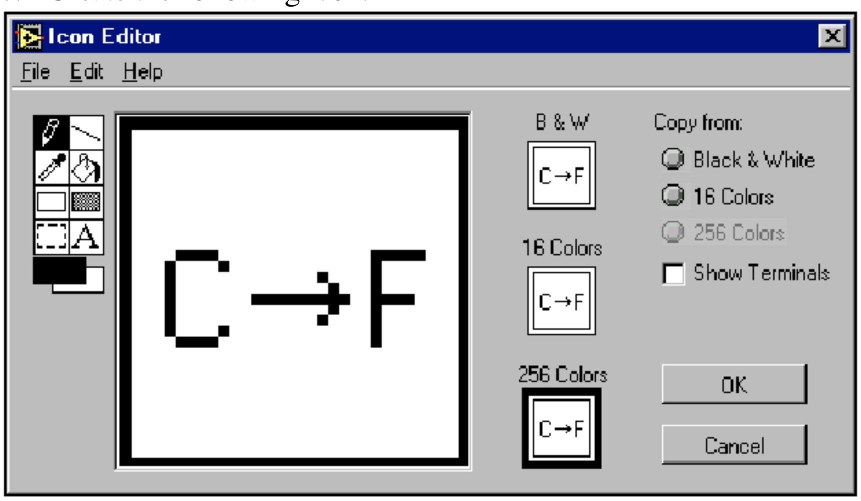
Front Panel

1. Select **File»Open** and navigate to `c:\exercises\LV Intro` to open the Convert C to F VI.
   If you closed all open VIs, click the **Open…** button on the **LabVIEW** dialog box.

Tip Click the arrow next to **Open…** button on the **LabVIEW** dialog box to open recently opened files, such as `Convert C to F.vi`.

The following front panel appears.

2. Right-click the icon in the upper right corner of the front panel and select **Edit Icon** from the shortcut menu. The **Icon Editor** dialog box appears.
3. Double-click the Select tool on the left side of the **Icon Editor** dialog box to select the default icon.
4. Press the <Delete> key to remove the default icon.
5. Double-click the Rectangle tool to redraw the border.
6. Create the following icon.



   a. Use the Text tool to click the editing area.
   b. Type `C` and `F`.
   c. Double-click the Text tool and change the font to **Small Fonts**.
   d. Use the Pencil tool to create the arrow.

Note To draw horizontal or vertical straight lines, press the <Shift> key while you use the Pencil tool to drag the cursor.

   e. Use the Select tool and the arrow keys to move the text and arrow you created.
   f. Select the **B&W** icon and select **256 Colors** in the **Copy from** field to create a black and white icon, which LabVIEW uses for printing unless you have a color printer.
   g. When the icon is complete, click the **OK** button to close the **Icon Editor** dialog box. The icon appears in the upper right corner of the front panel and block diagram.

7. Right-click the icon on the front panel and select **Show Connector** from the shortcut menu to define the connector pane terminal pattern.
   LabVIEW selects a connector pane pattern based on the number of controls and indicators on the front panel. For example, this front panel has two terminals, **deg C** and **deg F**, so LabVIEW selects a connector pane pattern with two terminals.

8. Assign the terminals to the digital control and digital indicator.
    a. Select **Help»Show Context Help** to display the **Context Help** window. View each connection in the **Context Help** window as you make it.
    b. Click the left terminal in the connector pane. The tool automatically changes to the Wiring tool, and the terminal turns black.
    c. Click the **deg C** control. The left terminal turns orange, and a marquee highlights the control.
    d. Click an open area of the front panel. The marquee disappears, and the terminal changes to the data type color of the control to indicate that you connected the terminal.
    e. Click the right terminal in the connector pane and click the **deg F** indicator. The right terminal turns orange.
    f. Click an open area on the front panel. Both terminals are orange.
    g. Move the cursor over the connector pane. The **Context Help** window shows that both terminals are connected to floating-point values.
9. Select **File»Save** to save the VI because you will use this VI later in the course.
10. Select **File»Close** to close the Convert C to F VI.

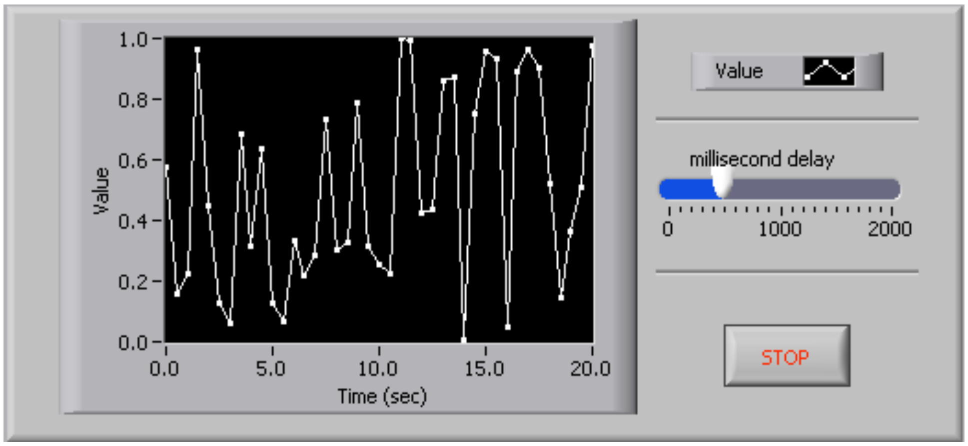
End of Exercise

Exercise 3 – Using Loops

Use a while loop and a waveform chart to build a VI that demonstrates software timing.

Front Panel

1. Open a new VI.
2. Build the following front panel.



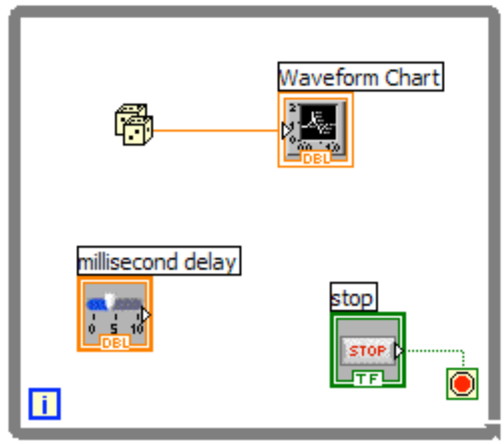
   a. Select the horizontal pointer slide on the **Controls»Numeric Controls** palette and place it on the front panel. You will use the slide to change the software timing.
   b. Type `millisecond delay` inside the label and click outside the label or click the **Enter** button on the toolbar, shown at left.
   c. Place a Stop Button from the **Controls»Buttons** palette.
   d. Select a waveform chart on the **Controls»Graph Indicators** palette and place it on the front panel. The waveform chart will display the data in real time.
   e. Type `Value History` inside the label and click outside the label or click the **Enter** button.
   f. The waveform chart legend labels the plot `Plot 0`. Use the Labeling tool to triple-click `Plot 0` in the chart legend, type `Value`, and click outside the label or click the **Enter** button to relabel the legend.
   g. The random number generator generates numbers between 0 and 1, in a classroom setting you could replace this with a data acquisition VI. Use the Labeling tool to double-click `10.0` in the y-axis, type `1`, and click outside the label or click the **Enter** button to rescale the chart.
   h. Change `–10.0` in the y-axis to `0`.
   i. Label the y-axis `Value` and the x-axis `Time (sec)`.

Block Diagram

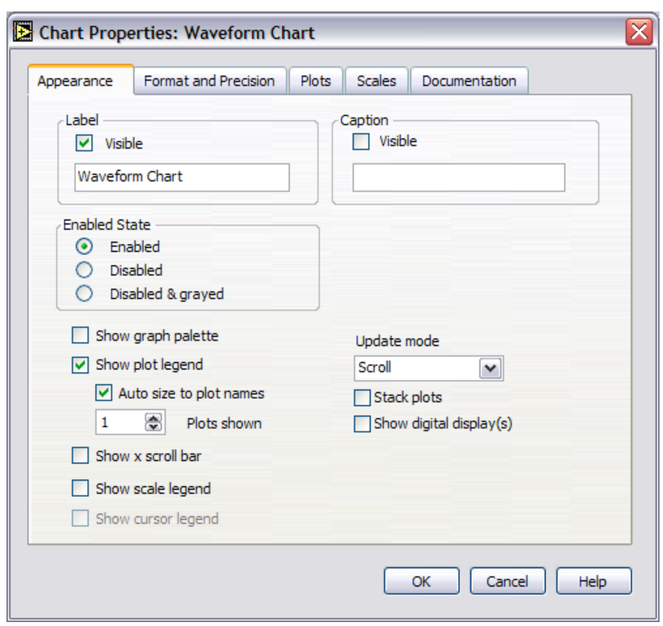
3. Select **Window»Show Diagram** to display the block diagram.

4. Enclose the two terminals in a While Loop, as shown in the following block diagram.

a. Select the While Loop on the **Functions»Execution Control** palette.
b. Click and drag a selection rectangle around the two terminals.
c. Use the Positioning tool to resize the loop, if necessary.



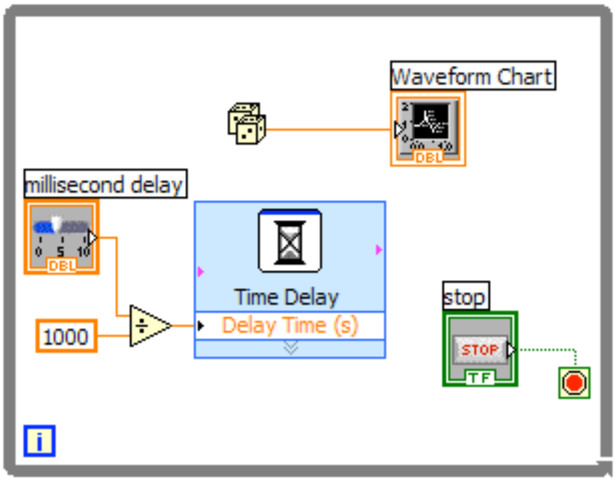
5. Select the Random Number (0-1) on the **Functions»Arithmetic and Comparison»Numeric** palette. Alternatively you could use a VI that is gathering data from an external sensor.
6. Wire the block diagram objects as shown in the previous block diagram.
7. Save the VI as `Use a Loop.vi` because you will use this VI later in the course.
8. Display the front panel by clicking it or by selecting **Window»Show Panel**.
9. Run the VI.
   The section of the block diagram within the While Loop border executes until the specified condition is TRUE. For example, while the STOP button is not pressed, the VI returns a new number and displays it on the waveform chart.
10. Click the STOP button to stop the acquisition. The condition is FALSE, and the loop stops executing.
11. Format and customize the X and Y scales of the waveform chart.
    a. Right-click the chart and select **Properties** from the shortcut menu. The following dialog box appears.
    b. Click the **Scale** tab and select different styles for the y-axis. You also can select different mapping modes, grid options, scaling factors, and formats and precisions. Notice that these will update interactively on the waveform chart
    c. Select the options you desire and click the **OK** button.



12. Right-click the waveform chart and select **Data Operations»Clear Chart** from the shortcut menu to clear the display buffer and reset the waveform chart. If the VI is running, you can select **Clear Chart** from the shortcut menu.

Adding Timing

When this VI runs, the While Loop executes as quickly as possible. Complete the following steps to take data at certain intervals, such as once every half-second, as shown in the following block diagram.



a. Place the Time Delay Express VI located on the **Functions»Execution Control** palette. In the dialog box that appears, insert 0.5. This function would make sure that each iteration occurs every half-second (500 ms).
b. Divide the millisecond delay by 1000 to get time in seconds. Connect the output of the divide function to the Delay Time (s) input of the Time Delay Express VI. This will allow you to adjust the speed of the execution from the pointer slide on the front panel.

13. Save the VI, because you will use this VI later in the course.
14. Run the VI.
15. Try different values for the millisecond delay and run the VI again. Notice how this effects the speed of the number generation and display.
16. Close the VI.

End of Exercise

Exercise 4 – Arrays [1,2]

Complete the following steps to build a VI that creates an array of random numbers, scales the resulting array, and takes a subset of that final array. You create a For Loop that runs for 10 iterations. Each iteration generates a random number and stores it at the output tunnel. **Random Array** displays an array of 10 random numbers. The VI multiplies each value in **Random Array** by a **Scaling Factor** to create another array called **Final Array**. The VI then takes a subset of the **Final Array** starting at **Start Subset** for **# of Elements** and displays the subset in Subset Array.

1.a) Front Panel

1. Open a blank VI and build the front panel shown in Figure 1.

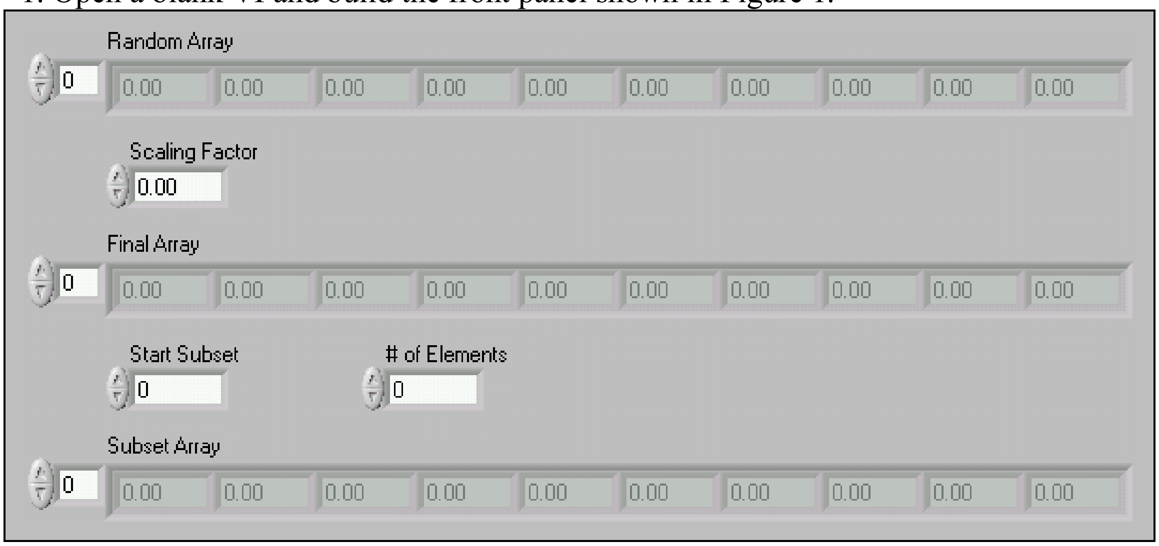


Figure 1

1. Place an array, located on the **Controls>>All Controls>>Array & Cluster** palette, on the front panel.
2. Label the array Random Array.
3. Place a numeric indicator, located on the **Controls>>Numeric Indicators** palette, in the array shell.
4. Use the Positioning tool to resize the array control to contain **10 numeric indicators**.
5. Press the **<Ctrl>** key while you click and drag the **Random Array** control to create two copies of the control.
6. Label the copies Final Array and Subset Array.
7. Place three numeric controls, located on the **Controls>>Numeric Controls** palette, and label them Scaling Factor, Start Subset, and # of Elements.
8. Right-click the **Start Subset** and **# of Elements** controls and select **Representation>>I32** from the shortcut menu.
9. Do not change the values of the front panel controls.

1.b) Block Diagram

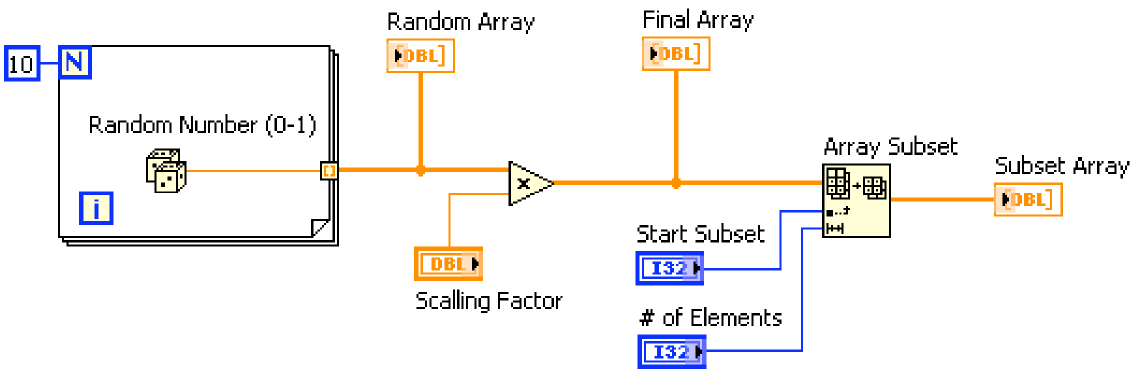
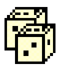
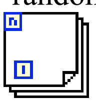
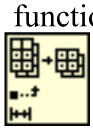


Figure 2

a.  Place the Random Number (0-1) function, located on the **Functions>>Arithmetic & Comparison>>Express Numeric palette**, on the block diagram. This function generates a random number between 0 and 1.

b.  Place the For Loop, located on the **Functions>>All Functions>>Structures palette**, on the block diagram. The loop accumulates an array of 1010 random numbers at the output tunnel. Create a constant of 1010 for the count terminal.

c.  Place the Multiply function, located on the **Functions>>Arithmetic & Comparison>>Express Numeric palette**, on the block diagram. In this exercise this function multiplies Random Array by Scaling Factor and returns Final Array.

d.  Place the Array Subset function, located on **the Functions>>All Functions>>Array** palette, on the block diagram. This function returns a portion of an array starting at St**art Subset** and containing **# of Elements elements**.

3. Save the VI as Array Exercise.vi in the C:\Exercises\ LabVIEW Basics I directory.

Run the VI

4. Display the front panel, change the values of the controls, and run the VI a few times.

5. Close the VI.

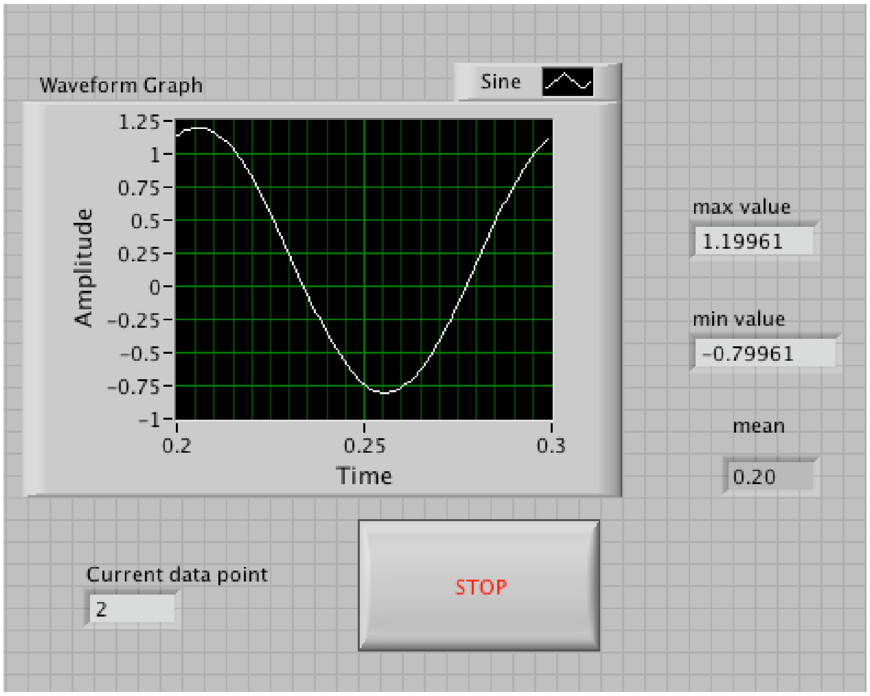
End of Exercise

Exercise 5 - Analyzing and Logging Data

Complete the following steps to build a VI that measures temperature every 0.25 s for 10s. During the acquisition, the VI displays the measurements in real time on a waveform chart. After the acquisition is complete, the VI plots the data on a graph and calculates the minimum, maximum, and average temperatures. The VI displays the best fit of the temperature graph.

Front Panel

1. Open a new VI and build the following front panel using the following tips.



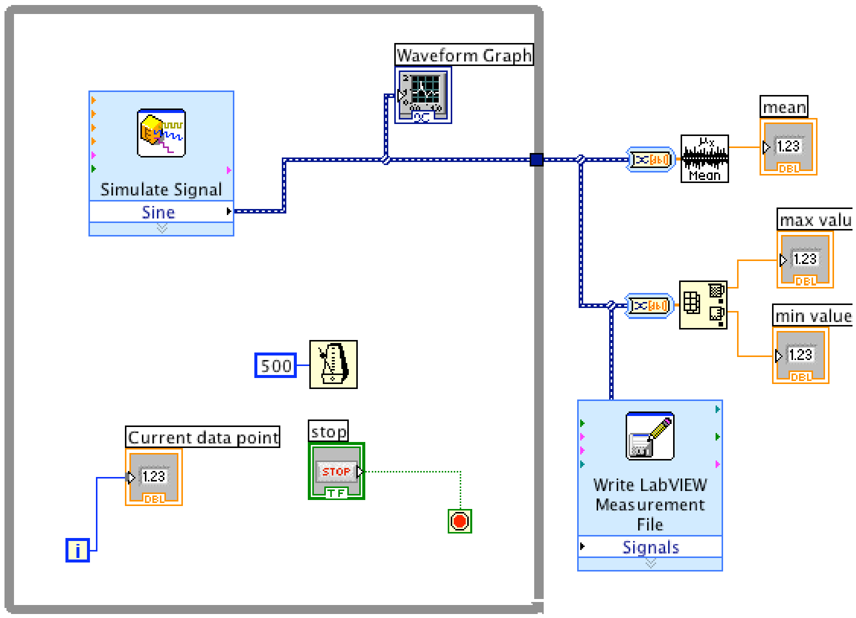
- Do not create the **Mean**, **Max**, and **Min** indicators yet. Create them on the Block Diagram by right clicking on the functions and choosing Create Indicator. Then position them on the Front Panel.

Block Diagram

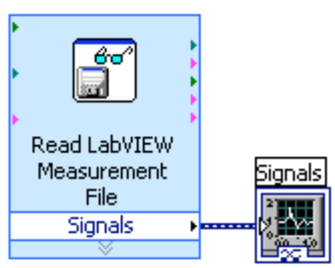
2. Build the following block diagram.

   a. Select the **Functions»Input»Simulate Signal.**
   b. Choose a frequency, Amplitude, and an offset
   c. Place the Wait Until Next ms Multiple function located on the **Functions»All Functions »Time & Dialog** palette and create a constant of `250`. Much like the Time Delay Express VI, this function causes the For Loop to execute every 0.25 s (250 ms).
   d. Place the Array Max & Min function located on the **Functions»All Functions »Array** palette. This function returns the maximum and minimum amplitude of the wave.

e. Place the Mean VI located on the **Functions»All Functions» Analyze»Mathematics»Probability and Statistics** palette. This VI returns the average of the amplitude of the wave.
f. Right-click the output terminals of the Array Max & Min function and Mean VI and select **Create»Indicator** from the shortcut menu to create the **Max**, **Min**, and **Mean** indicators.
g. Place the Write LabVIEW Measurements File Express VI located on the **Functions»Output** palette. Configure the VI to ask the user to choose the file to write and change the delimiter to Tab



3. Save the VI as `Temperature Logger.vi`.
4. Display the front panel and run the VI.
5. After pressing STOP a dialog box will appear. Enter the name of the file to save the spreadsheet.
6. Open the spreadsheet file to make sure the file was properly created by using Notepad or Excel, or by creating a VI to read the file as follows.
   - Create the following block diagram



   - Place the Read LabVIEW Measurement File Express VI located on the **Functions»Input** palette.
   - Configure the VI to ask the user to choose the file to read and change the delimiter to Tab
   - Right click on the Signals Output and choose create graph indicator

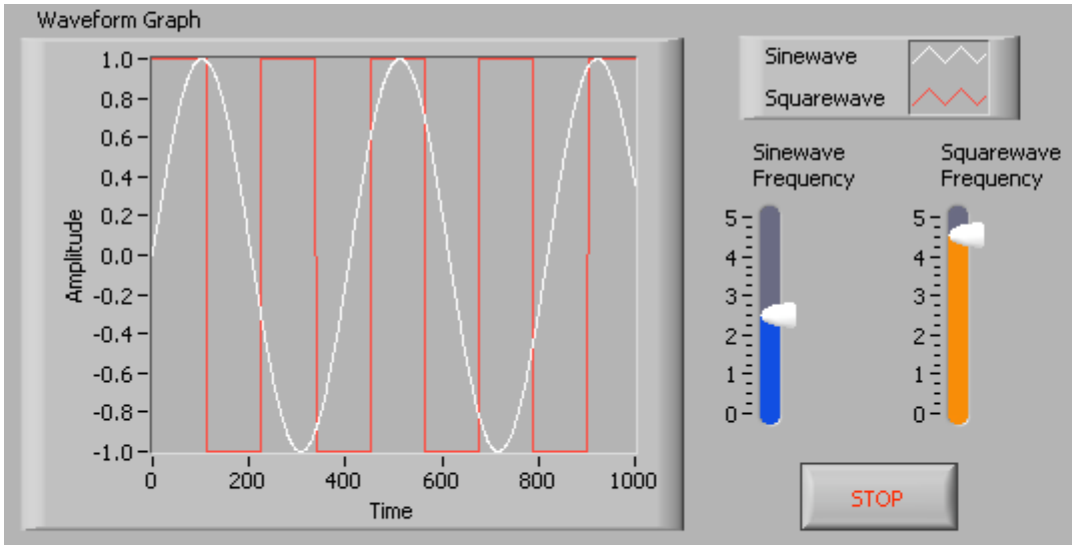
7. Run the VI
8. Save and close both of the VIs.

End of Exercise

Exercise 6 - Using Waveform Graphs

Front Panel

1. Open a new VI and build the following front panel using the following tips.



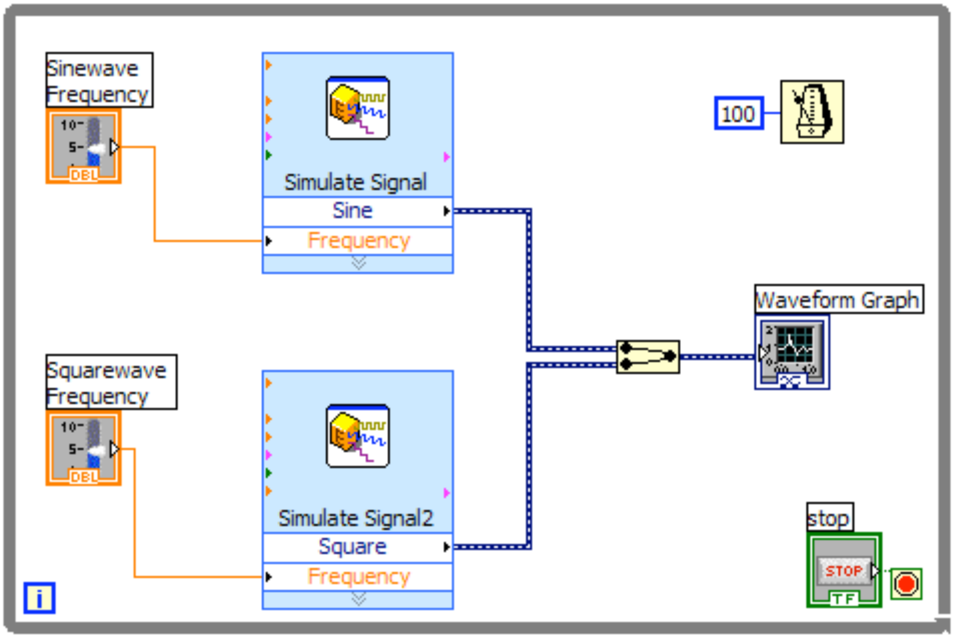
a. Create a waveform graph indicator from the **Controls»Graph Indicators** palette. Use the position/size/select tool to move the plot legend to the side, and expand it to display two plots. Use the labeling tool to change the plot names and the properties page to choose different colors for your plots.

b. Place a Stop button on the front panel**.**

c. Place two vertical pointer slides from the **Controls»Numeric Controls** palette. Use the properties page again to change the slide fill color.

Block Diagram

2. Build the following block diagram.



a. Place a While Loop from **Functions»Execution Control** palette.
b. Place a **Wait Until Next ms Multiple** from **Functions»All Functions »Time & Dialog** and create a constant with a value of 100.
c. Place two Simulate Signal Express VIs from the **Functions»Input** and leave the Signal type as Sine for the first Simulate Signal VI and change the Signal Type to Square for the second VI. Wire both of the outputs into the waveform graph. A Merge Signals function will automatically be inserted.
d. Expand the Simulate Signal Express VIs to show another Input/Output. By default, error out should appear. Change this to Frequency by clicking on error out and choosing **Frequency**.

3. Save the VI as `Multiplot Graph.vi`.

4. Display the front panel and run the VI.

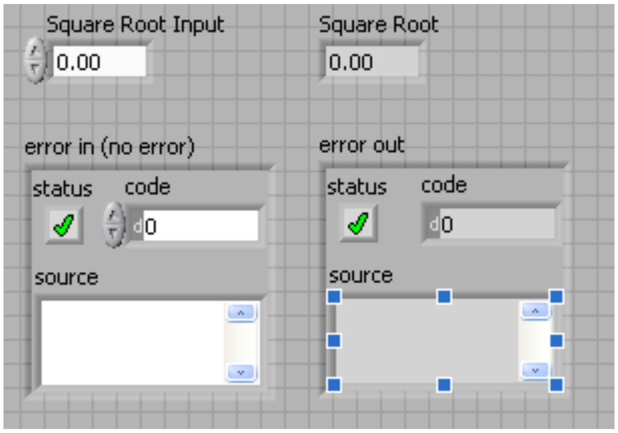
5. Save and close the VI.

End of Exercise

Exercise 7 - Error Clusters & Handling

Front Panel

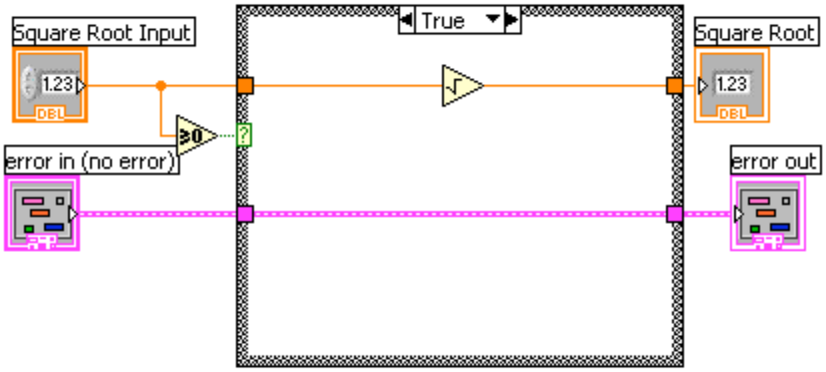
1. Open a new VI and build the following front panel using the following tips.



   a. Create a numeric control and change the Label to Square Root Input. Create a numeric indicator for Square Root.
   b. Place Error In 3D.ctl from **Controls»All Controls»Arrays & Clusters.**
   c. Place Error Out 3D.ctl from **Controls» All Controls»Arrays & Clusters.**

Block Diagram

2. Build the following block diagram.

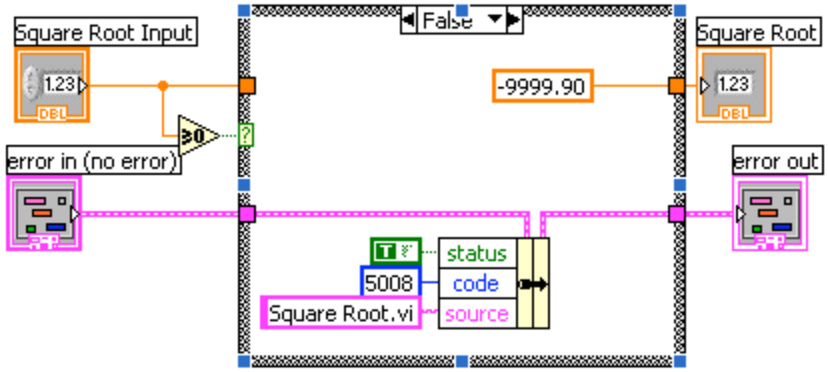


   a. Place a Case Structure from **Functions»Execution Control** palette.
   b. Place a **Greater or Equal to 0?** from the **Functions»Arithmetic and Comparison»Comparison** palette and wire it to the condition terminal of the case structure.

In the True Case:

   c. Place the Square Root function from **Functions»Arithmetic and Comparison»Numeric** palette.

In the False Case:



d. Create a numeric constant from **Functions»Arithmetic and Comparison»Numeric** palette and type -9999.90.
e. Place the Bundle By Name from **Functions»All Functions»Arrays & Clusters** palette. Wire from Error in to the center terminal of Bundle by Name to make status show up. Create constants. Wire from the Error Out indicator to the output of Bundle By name.

3. Save the VI as `Square Root.vi`.
4. Display the front panel and run the VI.
5. Save and close the VI.

End of Exercise

Exercise 8 - 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.

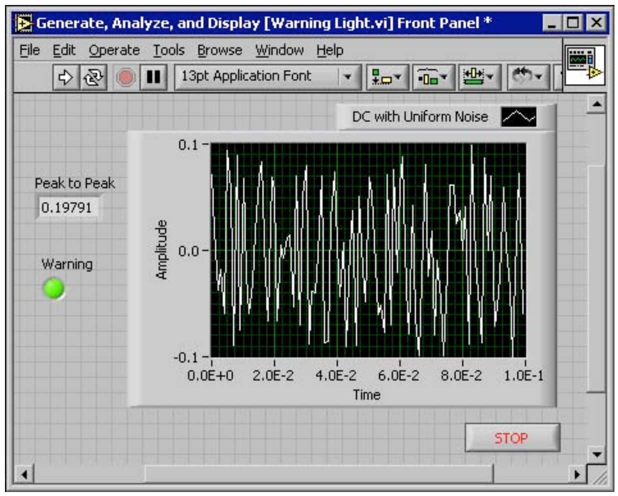


Figure 2-1 Front panel for the Warning Light VI

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.

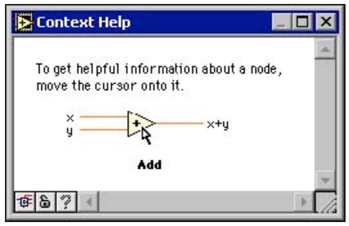
Note You also can access the **New** dialog box by clicking the arrow on the **New** button and selecting **New** from the **New** pull-down menu or by selecting **File>>New** from the front panel or block diagram menu bar.

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

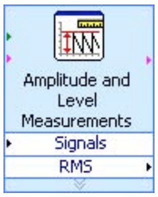
The template VI simulates a signal and analyzes it for its root mean square (RMS) value.

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.



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.

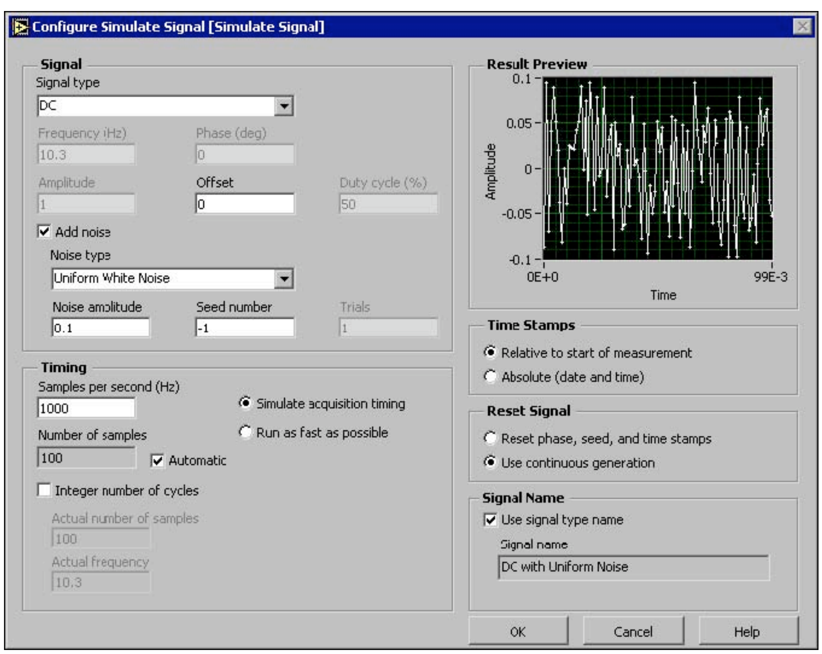


Figure 2-3 Configure Simulate Signal Dialog Box

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.

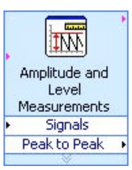
2. Remove the checkmark in the **RMS** checkbox in the **Amplitude Measurements** section.

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.

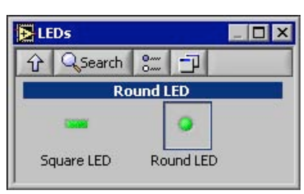


Figure 2-4 LEDs Palette

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

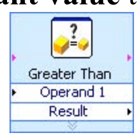
Setting the Warning Level Limit

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.

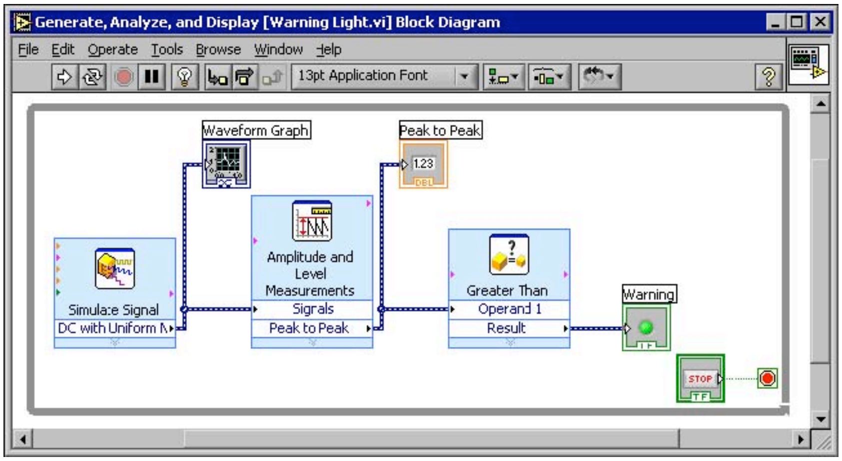
Notice how a **Peak to Peak** terminal, shown at left, appears on the block diagram. If the **Peak to Peak** terminal appears to be on top of the wires of the Express VIs, move the Express VIs and **Peak to Peak** terminal around to create more space. For example, move the **Peak to Peak** terminal into 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. Makesure 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.



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.
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** dialogbox 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

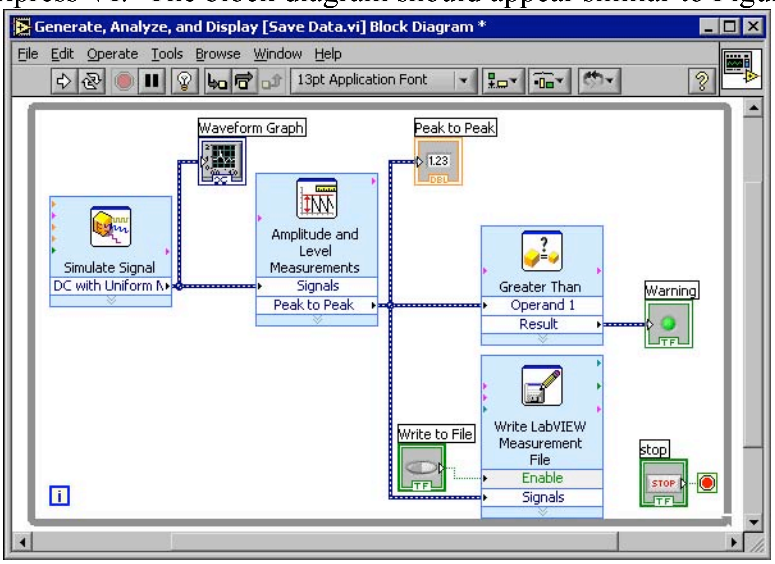
Complete the following steps to build a VI that logs data to a file when the user clicks a button on the front panel.

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.



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.

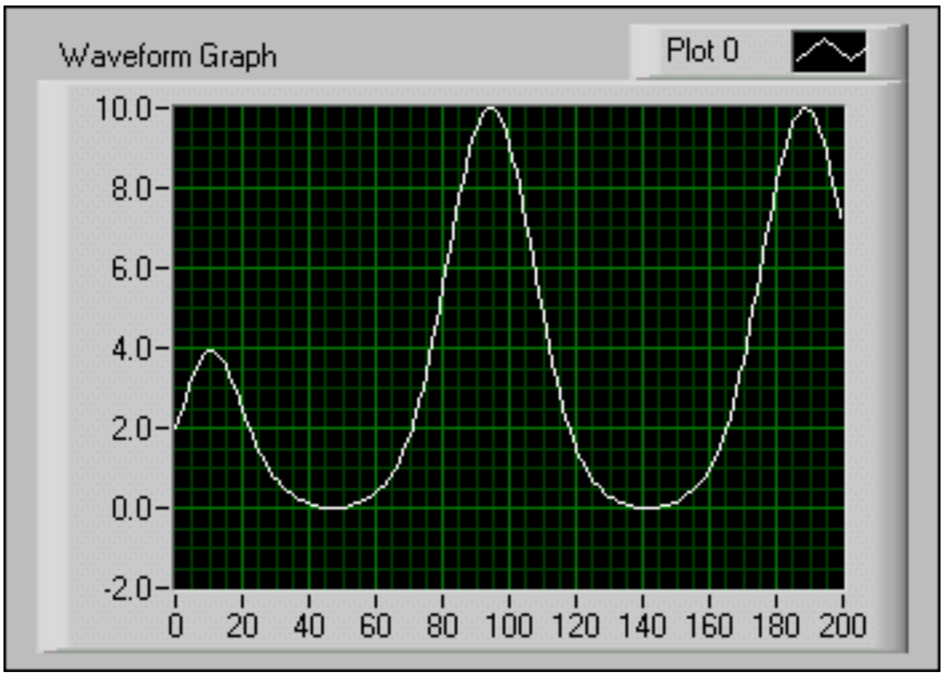
    Notice how the Selected Samples.lvm file differs from the test.lvm file. test.lvm recorded all the data generated by the Save Data VI, whereas Selected Samples.lvm only recorded the data when you pressed the **Write to File** button.

11. Save and close this VI.

➢ **Exercise 9: [2]**

Summary: In this exercise, the objective is to use the Formula Node in a VI. Complete the following steps to build a VI that uses the Formula Node to perform a complex mathematical operation and graphs the results.
Problem 1
1.a) Front Panel

1. Open a blank VI and build the front panel shown in figure 1.
   Figure 1



1.b) Block Diagram

1. Build the block diagram shown in figure 2.

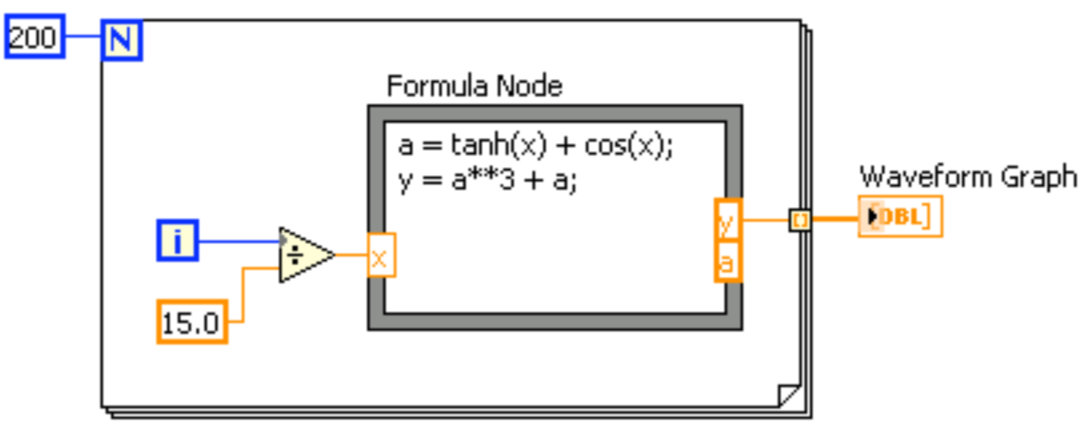


Figure 2

1. Place the **Formula Node**, located on the **Functions>>All Functions>>Structures** palette, on the block diagram.

2. Create the x input terminal by right-clicking the left border and selecting Add Input from the shortcut menu. Type x in the box that appears.

3. Create the y and a output terminals by right-clicking the right border and selecting Add Output from the shortcut menu. Enter y and a, respectively, in the boxes that appear. You must create output terminals for temporary variables like a.

Note: When you create an input or output terminal, you must use a variable name that exactly matches the one in the equation. Variable names are case sensitive.

4. Type the following equations in the Formula Node, where ** is the exponentiation operator. Refer to the LabVIEW Help for more information about syntax for the Formula Node.

$$a = Tanh(x) + Cos(x)$$
$$y = a^3 + a$$

5. Complete the block diagram as shown in figure 2.

2. Save the VI as Formula Node Exercise.vi.

1.c) Run the VI

1. Display the front panel and run the VI. The graph displays the plot of the equation $y = f(x)^3 + f(x)$ where $f(x) = Tanh(x) + Cos(x)$. During each iteration, the VI divides the iteration terminal value by 15.0. The quotient is wired to the Formula Node, which calculates the function value. The VI plots the array as a graph.

2. Close the VI.

- **Exercise 10: Stop Watch ***

Make a program that starts counting time when you click on "run." It should have a switch that you can click on to stop the program, at which time the total elapsed time in displayed in seconds. You will need while loops and sequence structure with sequence locals for this program.

- **Exercise 11: Arrays**

Build a VI that generates a 2D array of three rows by 10 columns that contains random numbers. After generating the array, index each row, and plot ezaach row on its own graph. The front panel should contain three graphs.

REFERENCES

1. http://zone.ni.com/devzone/learningcenter.nsf/webmain/60c2782788a811c986256cd50001a0a6
2. http://cnx.org/content/m12224/latest/
3. LabView Basics I: Introduction Course manual, National Instruments