The National Instruments LabVIEW™ software is an example of a visual programming application. This software uses graphic objects and methods, rather than text-based source code, to build data acquisition and instrument control applications called virtual instruments, or VIs. In this activity you will use simulated signal sources and measurement tools to build a VI, but LabVIEW can interface with voltage supplies, frequency generators, oscilloscopes, and other equipment using GPIB, Ethernet, USB, and other communication standards.

To begin, start the LabVIEW software and click the Launch LabVIEW button on the screen that appears. The program displays a start-up window similar to that in Figure LV–1.

Click on Blank VI under New in the left-hand pane. LabVIEW will create a workspace for a new VI that contains four basic elements: a front panel workspace with an associated Controls palette, and a block diagram workspace with an associated Functions palette. You can drag and reposition these workspaces and palettes so that all are visible in the LabVIEW window. If the Controls palette is not visible, click the Controls Palette item in the front panel View menu. If the Functions palette is not visible, click the Functions Palette item in the block diagram View menu. Figure LV–2 shows the default front panel workspace, Controls palette, block diagram workspace, and Functions palette when you start a blank VI.
The front panel workspace is the area in which you create the graphical user interface (GUI) for the VI using such graphic objects as button, knobs, and displays, from the Controls palette. The GUI is the part of the VI through which you interact with the instrument, much like the front panel of real-world devices. The block diagram workspace is the area in which you build the functional portion of the VI by selecting objects from the Functions palette and wiring them together. The functional part of the VI responds to the operation of the front panel controls and generates the information for the front panel displays. LabVIEW creates the code for the VI as you place and connect the graphic objects in the workspaces.

For the activities in this section, you will build and operate a circuit that allows you to configure and display the outputs of two simulated sine wave sources as an X-Y trace. This is called a Lissajous (pronounced LEE-sah-jo) figure, which shows the relative amplitude, phase, and frequency of two sine waves.

Activity 1: Placing the Workspace Objects

For this activity you will build the circuit GUI in the front panel workspace and the functional system in the block diagram workspace.

1. Right-click in the Controls palette and move the cursor over the Graph Indicators icon to open the Graph Indicators subpalette. Alternatively, you can double-click an icon to open its associated subpalette.

2. Click on the XY Graph icon to select the Express XY Graph object and move the cursor into the Front Panel workspace. An outline of the object will appear as you move the cursor inside the workspace.

3. Move the cursor to position the XY Graph object inside the workspace and click to place the display. Note that when you do so, an XY Graph object also appears in the Block Diagram workspace.
4. Right-click in the **Controls** palette and move the cursor over the **Num Ctrl** icon to open the **Numeric Controls** subpalette.

5. Use the **Num Ctrl** icon to place two **Numeric Control** objects to the left of the **XY Graph** object in the **Front Panel** workspace.

6. Select the **Block Diagram** workspace to activate the **Functions** palette.

7. Right-click in the **Functions** palette and move the cursor over the **Input** icon to open the **Input** subpalette.

8. Click on the **Simulate Sig** icon to select the **Simulate Signal** object and move the cursor into the **Block Diagram** workspace. An outline of the object will appear as you move the cursor inside the workspace.

9. Move the cursor to position the **Simulate Signal** object to between the **Numeric Display** and **XY Graph** objects in the workspace and click to place the object. Note that the Front Panel does not show a corresponding object for the **Simulate Signal** object.

10. Place a second **Simulate Signal** object in the workspace below the first **Simulate Signal** object. You can do this either by using the **Input** subpalette or by selecting the first Simulate Signal object and using the **Copy** (Ctrl+C) and **Paste** (Ctrl+V) commands in the **Edit** menu to create a second **Simulate Signal** object.

11. Click on or window around objects in the **Front Panel** and **Control Block** workspaces to arrange the objects as shown in Figure LV–3. You can use the mouse to drag the select objects, or use the arrow keys (←, ↑, →, or ↓) to position them.

![FIGURE LV–3](image)

Lissajous VI with Placed Objects

**Activity 2: Wiring the Workspace Objects**

For this activity you will connect the **Block Diagram** objects together so that you can configure the amplitude and phase of the **Simulate Signal2** object and display the X-Y trace on the **XY Graph** display.

1. Each of the **Simulate Signal** objects has six connect points along its left edge. Move the cursor over the **Simulate Signal2** connect points to display the associated control (**Offset**, **Frequency**, **Amplitude**, **Phase**, **Error In**, and **Reset**) for the connect points. Note that as you do so, the cursor changes to a spool of wire, indicating that you can connect a wire to the connect point. Click on the **Amplitude** connect point to connect a wire to the **Amplitude** control of the **Simulate Signal2** object.
2. Each of the Numeric Control objects have a connect point in the middle of its right edge. Move the spool to the connect point of the top Numeric Control object and click to complete the wire connection.

3. Connect the Phase connect point of the Simulate Signal2 object to the connect point of the bottom Numeric Control object.

4. If necessary, clean up the wiring by moving the wire segments so that they do not overlap. To do so, hover the cursor over a segment until the cursor becomes an arrow, and then click on and drag the segment to another location. Alternatively, you can click on the segment to select it and then use the arrow keys to move it.

5. Save your work as “Lissajous.vi” by selecting Save All (Ctrl+Shift+S) from the File menu of either workspace. Your VI should now look similar to Figure LV–4.

Activity 3: Configuring the VI Objects

To make the VI more user-friendly, you can configure object properties so that the representations and behaviors of the objects are consistent with their purpose. As an example, the x- and y-axes of the XY Graph display in the Front Panel workspace are labeled “Time” and “Amplitude”, respectively, even though both represent amplitude. For this activity you will customize the properties of the Numeric Control and XY Graph objects.

1. Right-click the top Numeric Control object in the Block Diagram workspace to open the context menu. Click on Properties to open the Numeric Properties window.

2. Select the Appearance tab and change “Numeric” to “Amplitude” in the Label section. Note that the label for the Front Panel object also changes.

3. Select the Data Entry tab and uncheck the Use Default Limits box. Change the Minimum value from “-Inf” to “0”, the Maximum value from “+Inf” to “5.0”, and the Increment value from “0.0000” to “0.1”. This configures the control so that its value changes from 0 to 5 in increments of 0.1.

4. Select the Display Format tab and change the Digits value from “6” to “2”. This limits the displayed value to two significant digits.

5. Open the Numeric Properties window for the bottom Numeric Control object in the block diagram workspace.
6. Select the Appearance tab and change “Numeric 2” to “Phase” in the Label section.

7. Select the Data Entry tab and uncheck the Use Default Limits box. Change the Minimum value from “−Inf” to “−180”, the Maximum value from “+Inf” to “180”, and the Increment value from “0.0000” to “5”.

8. Select the Display Format tab and change the Digits value from “6” to “3”.

9. Right-click the XY Graph object (NOT the Build XY Graph object) in the Block Diagram workspace to open the context menu. Click on Properties to open the Graph Properties window.

10. Select the Appearance tab and change “XY Graph” to “Lissajous Trace” in the Label section. Note that the label for the Front Panel object also changes.

11. Select the Scales tab and verify that the top drop-down list shows “Time (Y-Axis)”.  


13. Right-click the top Simulate Signal object in the Block Diagram workspace to open the context menu. Click on Properties to open the Configure Simulate Signal window. Alternatively, you can double-click the Simulate Signal object.

14. Change the Frequency value from “10.1” to “10”.

15. Use Save All in the File menu of either workspace to save your work. Your configured VI should now look similar to Figure LV–5.

**FIGURE LV–5**  
VI with Configured Objects

**Activity 4: Running the VI Application**

You are now ready to run the VI application. Click the Run Continuous button in either workspace. The trace on the Lissajous Trace display should be a horizontal line like that in Figure LV–6, because the amplitude of the Simulate Signal object connected to the Y-input is always 0.

Click the control arrows to the left of the Amplitude display (or click in the display and entering a positive value) to change the amplitude of the Y-input. As the amplitude of the Y-input increases, the slope of the trace becomes steeper. If the amplitude of the Y-input equals 1.0 (the same as the amplitude of the X-input) the slope is 45°.

Set the Amplitude value to “1” and change the Phase value, which can vary from a minimum value of −180° to a maximum value of 180°. Determine what phase values correspond to the Lissajous traces in Figure LV–7.
Click the **Abort Execution** button to stop the program. Use the **Configure Simulate Signal** window to change the frequency of the Y-input Simulate Signal object to 20 Hz so that the frequency of the Y-input signal is exactly twice that of the X-input signal. Set the **Front Panel** workspace **Amplitude** value to 1.0 and the **Phase** value to 0 and run the program.

Which of the Lissajous traces in Figure LV–8 do you see?

How do you explain the shape of the trace?

What would you expect to see if the frequency of the Y-input signal was 30 Hz (exactly three times that of the X-input signal)? Change the Y-input frequency to 30 Hz and verify your prediction.