DSP First
Lab 01: Introduction to LabVIEW

Pre-Lab and Warm-Up: You should read at least the Pre-Lab and Warm-up sections of this lab assignment and go over all exercises in the Pre-Lab section before coming to class.

Verification: The *Instructor Verification* should be signed off **during class time**. One of the laboratory instructors must verify the appropriate steps by signing on the **Instructor Verification** line. When you have completed a step that requires verification, simply demonstrate the step to the instructor.

Lab Report: It is only necessary to turn in a report on Section 3 with graphs and explanations. You are asked to **label** the axes of your plots and include a title for every plot. In order to keep track of plots, include your plot *inlined* within your report. If you are unsure about what is expected, ask.

1 Pre-Lab

In this first week, the Pre-Lab will be short and very easy. Make sure that you read through the information in Sections 1-4 below and do everything in Section 1 prior to coming to lab. Bring some **headphones** so you can listen to some signals.

1.1 Overview

LabVIEW will be used extensively in all the labs. The primary goal of this lab is to familiarize your with using LabVIEW. Here are three specific goals for this lab:

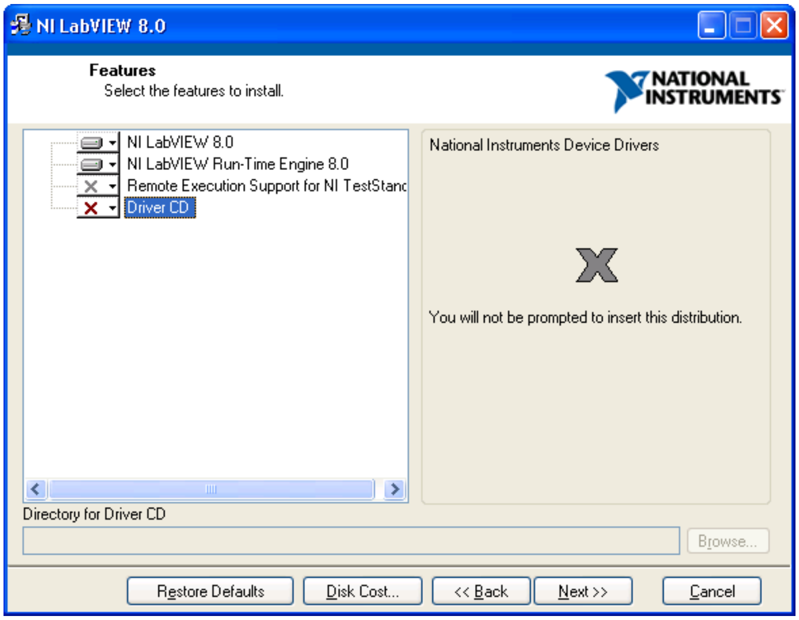
- Learn basic LabVIEW commands and syntax, including the *help* system.
- Learn to write and edit your own VIs (virtual instruments) in LabVIEW, and run them.
- Learn a little about advanced programming techniques for LabVIEW[1].

1.2 Install LabVIEW

Here's how to get LabVIEW installed. Warning, allow at least 30 minutes to install LabVIEW.

- Go to **\\tibia\public\Course Software\National Instruments\LabVIEW 8.0**
- Run `setup.exe.`
- When it asks for a license number, use the one given on Angel**.** This number is for this class only. Please don't give it to others. Doing so will violate our license agreement and prevent Rose-Hulman from getting favorable licenses from NI in the future.
- Select the default answers until you get to **Features**. Select the X next to the Driver CD as shown below.

[1] Read Section I (pages 1-19) of the *Introduction to LabVIEW – Six Hour Course* (on Angel) for more details.



- Continue selecting the defaults until it is all installed.
- Point your web browser to www.rose-hulman.edu/DSPFirst.
- Login with the username and password given in class.
- Click on **Getting Started**. (Near the top of the left column.)
- Scroll to the bottom and follow the instructions for installing the DSP First toolkit.

1.3 Getting Started - Open and Run a Virtual Instrument

In this part of the pre-lab you will examine the **DSP First Demo** VI and run it. Change the frequencies and types of the input signals and notice how the display on the graph changes. Here is how to do it:

a) Select **Start»All Programs»National Instruments»LabVIEW 8.0»LabVIEW** to launch LabVIEW. The **LabVIEW** dialog box appears.

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

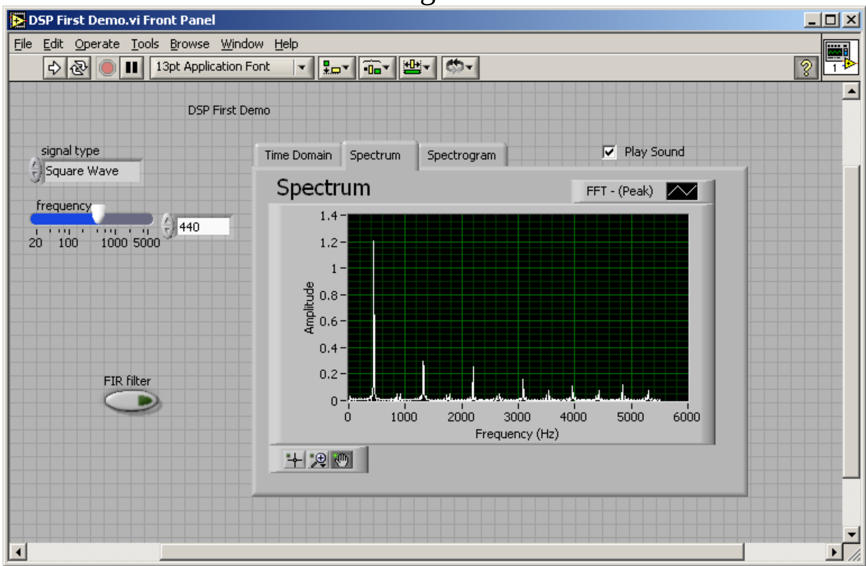
c) On the Search Tab, enter **dsp.** Double-click on **DSPFirst**. Choose **DSP First Demo.vi.** This will open the DSP First Demo VI Front Panel.

Note:	This is how you find all the DSP First demos.

Front Panel

a) Check the box next to **Play Sound**.

b) Click the **Run** button () on the toolbar, shown at left, to run this VI once. You should hear a short tone. This VI determines the result of filtering a generated signal. This example also displays the magnitude spectrum for the generated signal. The resulting signals are displayed in the graphs on the front panel, as shown in the following figure. Try clicking the **Time Domain**, **Spectrum**, and **Spectrogram** tabs to see the various views of the signal.



c) Try changing the **signal type** and **frequency**. You will have to click the **Run** button after each change to see/hear the result.

Block Diagram

a) Select **Window»Show Block 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.

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

c) Close the VI and do not save changes.

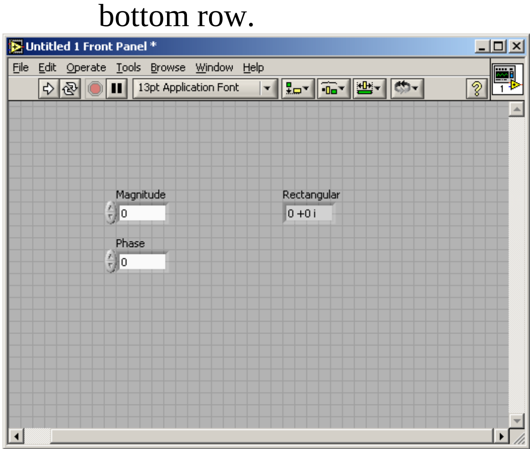
2 Warmup: Building VIs – Display Complex Numbers in Polar and Rectangular

The previous exercise gave you experience with running a VI. The real fun starts when you build your own VIs. The next exercise will walk you through building a VI and modifying it to do interesting things. Complete the following steps to create a VI that takes a complex number in polar form and displays it in both polar and rectangular form.

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

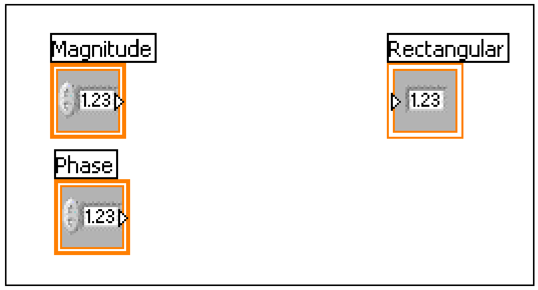
1. Select **File»New VI** to open a new front panel.
2. Create a numeric digital control. You will use this control to enter the value for the sinusoid magnitude.
    a. Right-click an open area on the front panel to display the **Controls** palette. Select the `Numeric Control` on the **Controls»Modern»Numeric** palette.
    b. Move the control to the front panel and click to place the control.
    c. Type `Magnitude` inside the label and click outside the label. If you do not type the name immediately, LabVIEW uses a default label. You can edit a label at any time by double clicking on it.
3. Create a second number control.
    a. Hold the Ctrl key down and click and drag the first control. You now have a second control.
    b. Label the second control `Phase`.
4. Create a numeric digital indicator. You will use this indicator to display the complex value.
    a. Select the `Numeric Indicator` on the **Controls» Modern»Numeric** palette.
    b. Move the indicator to the front panel and click to place the indicator.
    c. Type `Rectangular` inside the label and click outside the label.
    d. Make this indicator complex by right-clicking on the control and selecting **Representation**. Click on the **CDB** block which is in the middle of the bottom row.



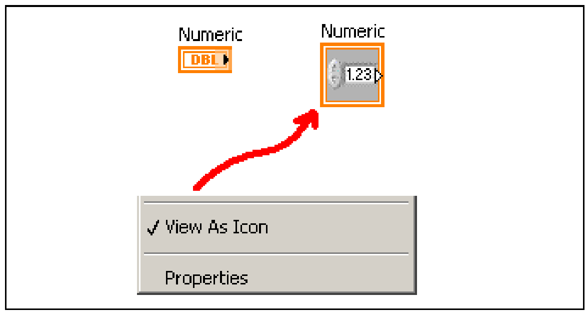
LabVIEW creates corresponding control and indicator terminals on the block diagram. The terminals represent the data type of the control or indicator. For example, a DBL terminal represents a double-precision, floating-point numeric control or indicator.

Block Diagram

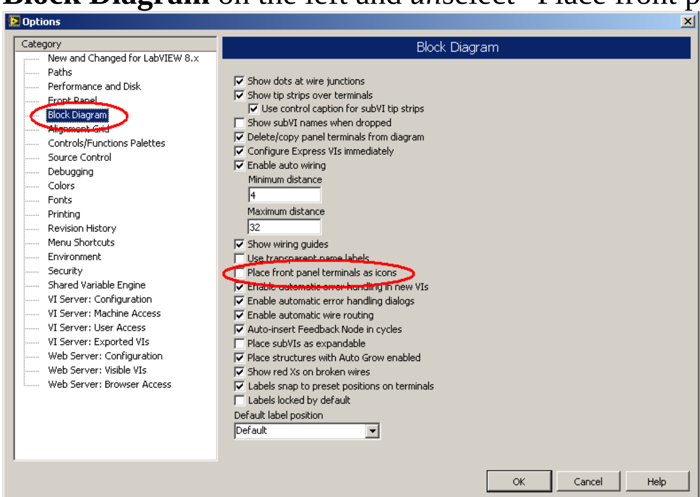
5. Display the block diagram by clicking it or by selecting **Window»Show Block Diagram**.



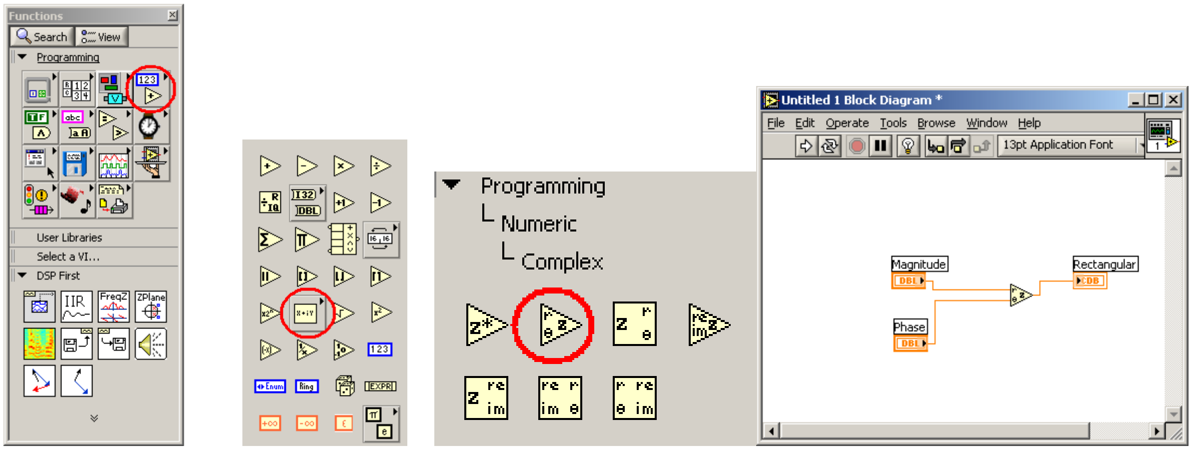
Note: Block Diagram terminals can be viewed as icons or as terminals. To change the way LabVIEW displays these objects right click on a terminal and select **View As Icon**.



You can make this happen automatically by selecting **Tools»Options…** then click on **Block Diagram** on the left and *un*select "Place front panel terminals as icons".

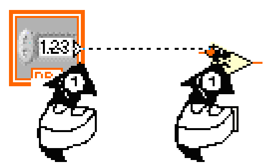


6. You can display the complex blocks by right-clicking an open area on the block diagram. Select the `Polar to Complex` function on the **Programming»Numeric»Complex** palette and place them on the block diagram.



These are the blocks for manipulating complex numbers. You should be able to guess what they do. If not, display the **Context Help** window (**Help»Show Context Help** or <Ctrl-H>) and point to each block.

7. Use the Wiring tool to wire the icons as shown above.
    - To wire from one terminal to another, hover your cursor near the first terminal. It should turn to a spool of wire. When it does, 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.

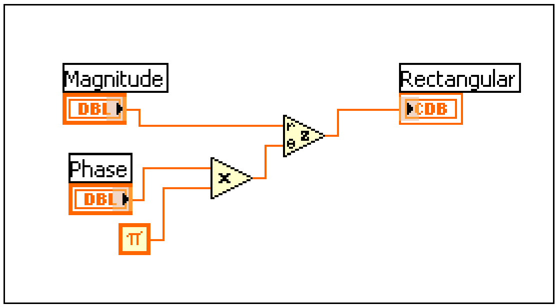


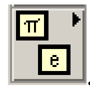
    - You can bend a wire by clicking to tack the wire down and moving the cursor in a perpendicular direction. Press the spacebar to toggle the wire direction.
    - When you move the Wiring tool over a terminal, the terminal area blinks, indicating that clicking will connect the wire to that terminal and 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.
8. Display the front panel by clicking it or by selecting **Window»Show Panel (**or <Ctrl-E>.
9. Save the VI because you will use this VI later in the course.
    a. Select **File»Save**.
    b. Navigate to your class directory.

Note Save all the VIs you edit in this course in your class directory.

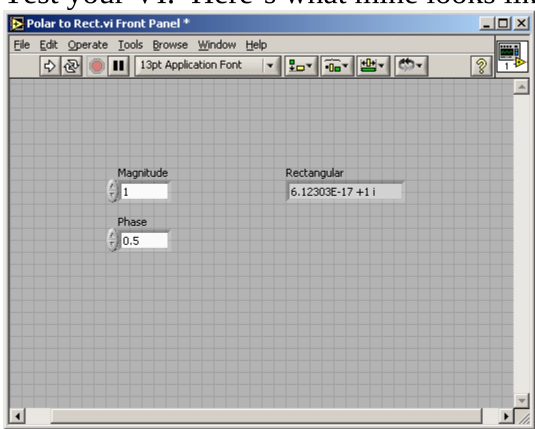
    c. Type `Polar to Rect` in the dialog box.
    d. Click the **Save** button.
10. Enter a number in the digital control and run the VI. The result is too big for the box, so stretch the Rectangular box so you can see both the real and imaginary parts.
    a. Double-click the digital control and type a new number.
    b. Click the **Run** button to run the VI.
    c. Try several different numbers and run the VI again.

11. It would be nice to enter the phase in fractions of π. Modify your block diagram to look like:



Hint: You will find the π block in the palette that looks like .

12. Test your VI. Here's what mine looks like:

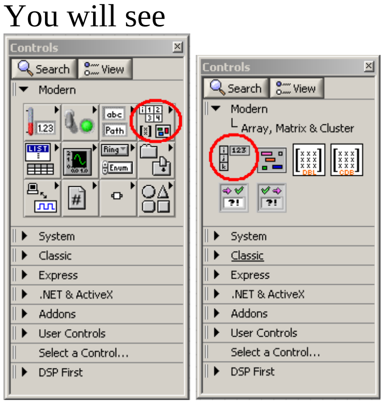


13. Select **File»Close** to close the VI.

2.1 Arrays of Complex Numbers

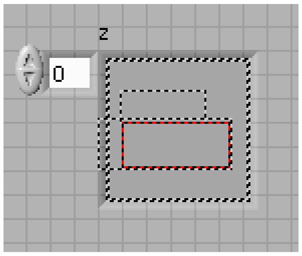
Throughout this class we will be doing a lot with complex numbers, so let's see how LabVIEW handles arrays of them.

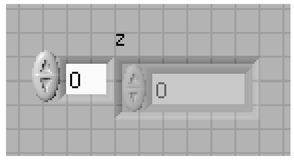
1. Select **File»New VI** to open a new front panel.
2. Go to the front panel and create an array control by right-clicking on the background. You will see

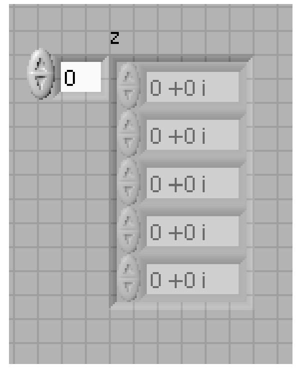


Select **Array, Matrix & Cluster**. You will see the second palette above, select **Array** and place it on the front panel. Change the name to z. Note: In LabVIEW an array is a one dimensional list of elements, while a matrix is a two dimensional list.

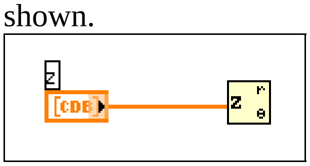
3. Now you need to say what the array will hold. Do this by selecting a Numeric Control (like you did before) and dragging it *into* the array.



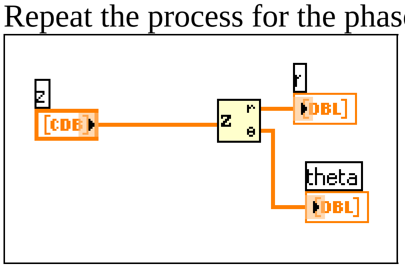




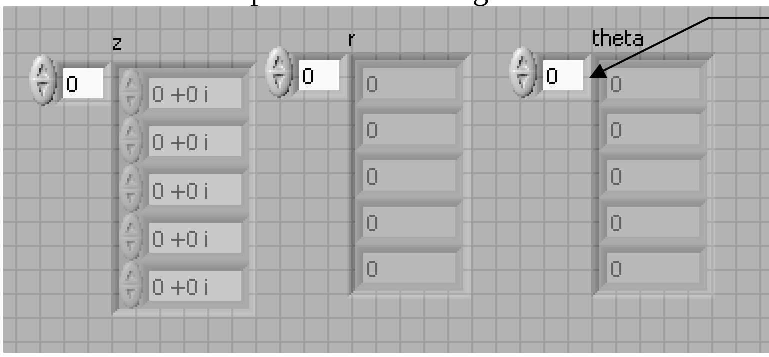
   Release the control and it will look like the middle picture above.
4. Change the data type to Complex Double (CDB) by right-clicking on the control and selecting **Representation**. Select **CDB** from the bottom middle. You just made this control handle complex double values.
5. Click on the bottom of the control and drag it down a bit. You will now see something like the right-most picture above. This lets you see and input several values at once.
6. Switch to the **block diagram**. <Ctrl-E> is a fast way to do this.
7. Your goal is to display the real, imaginary, absolute value, and phase of z using these Complex blocks. I'll start you with the first one and you should be able to figure out the rest. Place the **Complex to Polar** block (middle one) on your block diagram and wire as shown.



8. Right-click on **r** terminal of the **Complex to Polar** block and select **Create»Indicator**. This is a quick way to making a new indicator.
9. The indicator is labeled **r** by default. You may wish to change it to **magnitude**.
10. Repeat the process for the phase output as shown below.

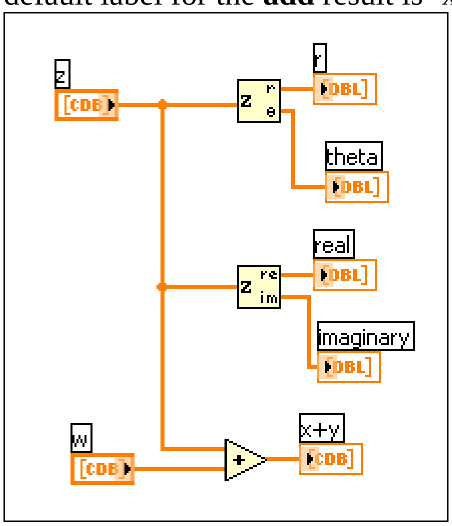


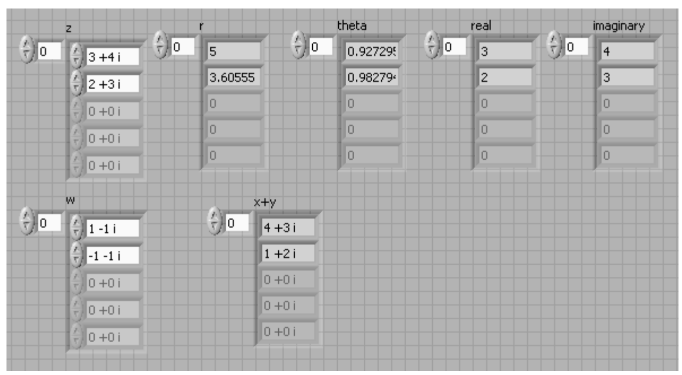
11. Switch to the front panel and rearrange like:



I find that clicking in between the textboxes is a good place to "grab" the entire control for repositioning.

12. Enter a complex number in `z` (try 3+j4, which has an easy magnitude)[2] and click **run**. Does the correct answer appear? Add other values to `z` and try again.
13. Place the **Complex to Re/Im** block on the block diagram and attach indicators. Click **run** and verify that they are working.
14. Go to the block diagram view and add another complex variable input by holding the Ctrl key down and clicking and dragging the `z` block. Label this new block `w`.
15. Get an **add** block[3] and add `z` to `w`. Attach an indicator. Verify that it works. Note the default label for the **add** result is 'x+y'. You may wish to change it to 'z+w'.



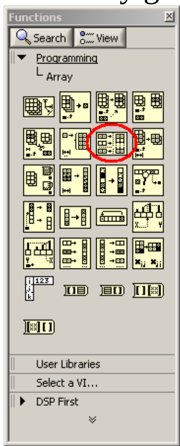


16. You can combine the arrays `z` and `w` into one array.

[2] You'll have to type `3+4i`.

[3] Reminder: right-click on the block diagram to get a palette of blocks. Click on **Numeric** and select the **Add** block. If you aren't sure where a block is, right-click on the background and then click on **Search**. Type the name of the block for which you are looking.

a. Do this by going to **Programming»Array** and selecting **Build Array**.



b. Click on the bottom of the **Build Array** icon and stretch it until it has two inputs.
c. Right-click on **Build Array** and select **Concatenate Inputs.**
d. Attach `z` and `w` to the inputs.
e. Create an indicator for the output. Hint: Right-click, create.
f. Run the VI. Does it work as expected?
g. This also works for combining scalars into an array.

17. Save this as `Complex Arrays`.

You seen that it is rather easy to build up entire arrays of complex numbers and manipulate them. Next you'll see how to build some signals.

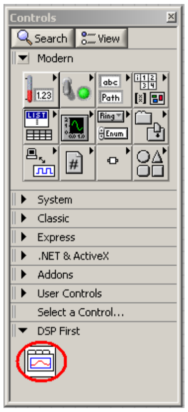
2.2 Generating Signals

In this section we'll see that it's easy to generate waveforms in LabVIEW. A waveform is a datatype that contains an array of sample values and information about the sampling rate.

1. Start with a new VI by clicking **File:New VI**.
2. Select the **block diagram** <Ctrl-E>.
3. Right-click and then click the pushpin in the upper left. Click on **DSP First**. You will see:

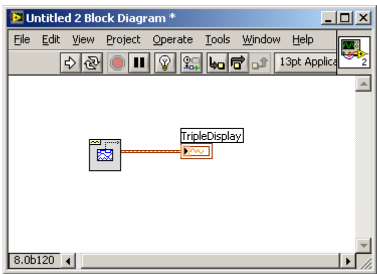
These are the VIs for this class.

4. Select the **DSP First Function Generator** (upper left) and place it on the block diagram.
5. Go to the Front Panel <Ctrl-E>, right-click, choose DSPFirst, then select and place the **TripleDisplay**.



This will place a VI on the block diagram. If the **TripleDisplay** block has the connector on the right side rather than the left side, right-click on it and select **Change to Indicator**.

6. Attach the two as shown.



7. Click the run button.
8. Use the context help[4] to learn what the Amplitude, Frequency, Phase, and Sampling Info inputs on the **DSP First Function Generator** default to. What are the units on the Phase?
9. Attach controls to Amplitude, Frequency, and Phase inputs. Hint: right-click on the input and select **Create**.
10. Try different values.

The **DSP First Function Generator** generates cosines by default. You can generate other waveforms with it by attaching a control to the **signal type** connector. There are many other sinusoidal generators in LabVIEW, however they generate sines. Since Electrical Engineers use cosines, stick with our generator.

2.3 LabVIEW Sound

The exercises in the section involve sound signals, so you should bring headphones to the lab for listening.

1. Generate a tone (i.e. a sinusoid) in LabVIEW and listen to it. The previous section showed how to generate a sinusoid. For this part, the frequency of your sinusoid tone should be 2000 Hz and its duration should be 1 second. Use a sampling rate equal to 11025 samples/sec. To set the sampling rate and duration, right-click on the **sampling info** terminal on the **DSP First Function Generator** (bottom left) and select **Create»Constant**. The top number in the box is the sampling rate and the bottom is the number of samples. Change these to match above.

[4] If the small context help window isn't appearing, type <Ctrl-H> to make it appear.

2. Listen to the tone by clicking the **Play Sound** check box on the **TripleDisplay**. Click run. Does the tone sound correct?
3. The Time Domain display is not very informative at this scale. See if you can zoom in to see something more sinusoidal. Click the magnifying glass at the bottom of the graph for zoom options.
4. Try different amplitudes and frequencies. To make this easier, change the input constants to controls. Hint: right-click the constant and select **Create»Control**. Delete the constant and attach the control. Can the frequency be too high or too low?

Instructor Verification (separate page)

3 Lab Exercise: Manipulating Sinusoids with LabVIEW

Now you're on your own. **Include a short summary of this Section with plots in your Lab report.** Build a LabVIEW VI to do steps (a) through (d) below. Include a printout of both the front panel and the block diagram with your report.

(a) Set the **sampling info** so that the **Function Generator** will produce approximately two cycles of the 4000 Hz sinusoids defined in the next part, part (b). Set the sampling rate so that you have at least 25 samples per period of the sinusoidal wave.

(b) Generate two 4000 Hz sinusoids with arbitrary amplitude and time-shift.

$x_1(t) = A_1 \cos(2\pi(4000)(t - t_{m_1}))$ $x_2(t) = A_2 \cos(2\pi(4000)(t - t_{m2}))$

Select the value of the amplitudes and time-shifts as follows: Set A_1 equal to your age and set $A_2 = 1.2A_1$. For the time-shifts, set $t_{m_1} = (37.2/M)T$ and $t_{m_2} = -(41.3/D)T$ where D and M are the day and month of your birthday and T is the period. Make a plot of both signals over the range of $0 \le t \le 2T$. For your final printed output in part (d) below, place three **TripleDisplay**s one above the other on the front panel.

(c) Create a third sinusoid as the sum: $x_3(t) = x_1(t) + x_2(t)$. In LabVIEW this amounts to summing the vectors that hold the values of each sinusoid. Make a plot of x_3 over the same range of time as used in the plots of part (b).

(d) Before printing the three plots, put a title on each subplot, and include your name in one of the titles. Then right-click on each plot and select **Data Operations»Export Simplified Image…**[5]. Put the plot on your clipboard and then paste into your word processor.

3.1 Theoretical Calculations

Remember that the phase of a sinusoid can be calculated after measuring the time location of a positive peak,[6] if we know the frequency.

(a) Make measurements of the "time-location of a positive peak" and the amplitude from the plots of x_1 and x_2, and write those values for A_i and t_{m_i} directly on the plots.

Then calculate (by hand) the phases of the two signals, $x_1(t)$ and $x_2(t)$, by converting each time-shift t_{m_i} to phase. Write the calculated phases ϕ_i directly on the plots.

Note: when doing computations, express phase angles in radians, not degrees!

[5] If **Export Simplified Image** doesn't appear, press **Alt-PrintScrn** and paste into your word processor.

[6] Usually we say time-delay or time-shift instead of the "time location of a positive peak."

(b) Measure the amplitude A_3 and time-shift t_{m_3} directly from the plot and then calculate the phase ϕ_3 by hand. Write these values directly on the plot to show how the amplitude and time-shift were measured, and how the phase was calculated.

(c) Now use the phasor addition theorem. Carry out a phasor addition of complex amplitudes for $x_1(t)$ and $x_2(t)$ to determine the complex amplitude for $x_3(t)$. Use the complex amplitude for $x_3(t)$ to verify that your previous calculations of A_3 and ϕ_3 were correct.

4 LabVIEW things to remember for future labs:

- Where to find DSP First demos
- How to turn on and use Context Help
- Using right-click to find palettes
- How to search for VIs (blocks)
- Using right-click to attach constants, controls, and indicators
- Where to find the complex operators
- How to create array controls
- How to declare the 'type' of an array
- How to change the data representation of a numeric input
- How to generate a cosine waveform
- How to change the sampling rate and number of samples generated
- How to use a **TripleDisplay**.
- How to copy a plot to be pasted into a word processor.
- … What else?

Lab 01
INSTRUCTOR VERIFICATION SHEET
Turn this page in to your grading TA.

Name: ______________________________ Date of Lab: ________________________

Part 2.2 Generating Signals
Part 2.3 LabVIEW Sound

Verified: _________________________ Date/Time: ____________________