

ME220 Lab #1

Introduction to LabView Environment and Signals in the Time and Frequency Domain

Lab Report Due: 4/15/08, before 5:00 pm, outside Terman 538

Before you start:

Form a Lab Team. For later labs, you'll need a lab kit, and we need to know who the teams are in order to build enough lab kits. Your lab team should be 2 or 3 people, and you should be able to arrange schedules so that you're able to work together. Most lab exercises will take 3-5 hours.

We want you to work together in small groups, mainly because that is more efficient use of the lab facilities. Also, you can learn more working in a group than on your own. Hopefully, these labs are more fun in small groups than on your own.

The lab is available for use 24 hours/day, 7 days/week. There are 8 workstations suitable for these exercises in the lab, so there should be plenty of capacity.

1. Go to the lab (Terman 577, Door Combo 52413)
2. Log onto a station, making sure the computer is set to the WIN domain. Username and Password are the same as your regular SUNet ID. All the files you need will be at : U:\220_student. Warning! The startup may be painfully slow. This will be a nice exercise in patience.
3. Create a folder with your team name inside the U:\220_student folder – this will be the location where you save your work during this quarter.

Goals of this Lab :

Many of you have never used LabView for measurements. It is a common tool in experimental research, industrial product development, and manufacturing operations. You'll eventually have many chances to use this tool in research or in a job. In this course, we want to use it to carry out some experiments and to provide some efficient analyses of the resulting data. The goal of this Lab is to get you all up to speed on some of the features of LabView, and to use the LabView environment to learn some useful things about time-domain and frequency-domain analysis of some simple signals.

If you're a LabView "Veteran", there will be some old, familiar material in this exercise, but there are some Features of LabView 7.0 that you will see for the first time and hopefully appreciate.

Acknowledgement

Zachary Nelson of National Instruments orchestrated a donation of the LabView Software and the National Instruments Data Acquisition hardware that we'll be using in the lab this quarter. Zach will visit sometimes during the quarter, so please be nice to him!

Exercise 0 - Open and Run a Virtual Instrument

The goal of this section is to examine the **Signal Generation and Processing** virtual instrument (VI) and run it, change the frequencies and types of the input signals, and notice how the display on the graph changes. You'll get some experience changing the Signal Processing Window and Filter options.

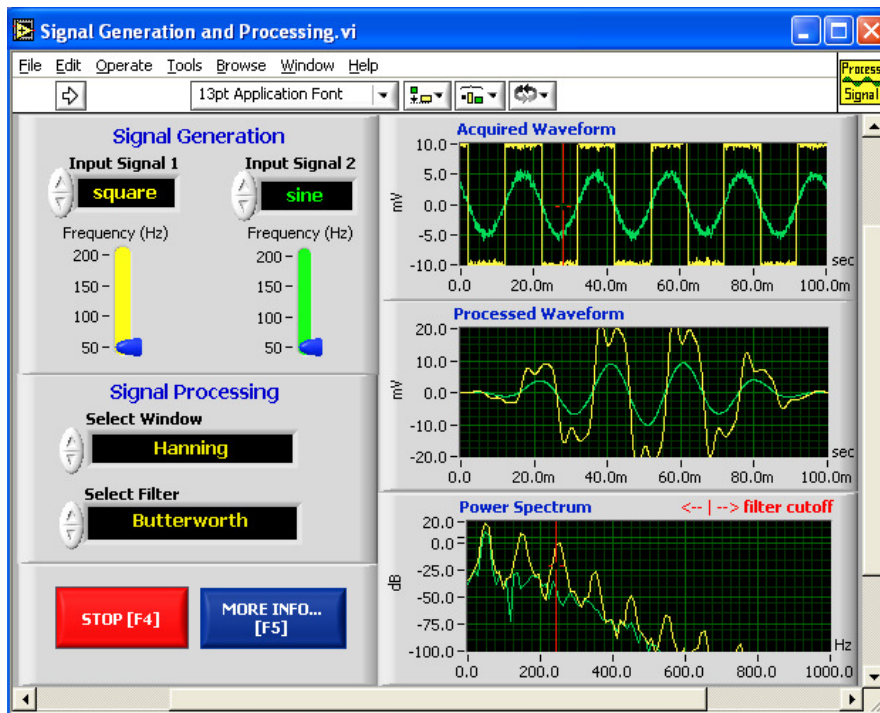
Open U:\220_student\Lab 1 and click on <Signal Generation and Processing.VI>. Ignore any warning messages that appear and close the warnings window. Close the "Controls" window if it appears.

Front Panel



1. Click the **Run** button on the toolbar, (looks like an Arrow, positioned on the top left corner right under the "edit" pull-down) to run this VI.

This VI generates a pair of time-dependent functions (set by the controls on the upper left), and shows what happens to those signals during the formation of a Fast Fourier Transform of the signals. This example also displays the power spectrum that results from a FFT for the generated signal. The resulting signals are displayed in the graphs on the front panel, as shown in the following figure. Try turning one of the sources off to play around with just a single signal. See comment below for more info.



2. Use the Operating tool, shown at left, to change the parameters of your wave function. For example, you can change the type of input signal, change the frequency of an input signal using the slider, or change the filter used on the signal.

3. Press the **More Info...** button or [F5] to read more about the analysis functions.
4. Press the **Stop** button or [F4] to stop the VI.

Block Diagram

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

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.

6. Select **Window»Show Panel** or press the <Ctrl-E> keys to return to the Front Panel.
7. Close the VI and do not save changes.

End of Exercise

Comments: This VI is a nice demonstration of how the FFT works in real instruments. The incoming waveform is usually measured over some finite time interval, and the beginning and end of the signal over this interval is usually abrupt. The FFT assumes that the signal is periodic, so it is necessary to smooth out the beginning and end. The various “window” functions do this in different ways with different effects on the shape of peaks. This exercise allowed you to explore the relationship between the time-domain view of the signal and the frequency-domain view of the same signal and to investigate the effect of filters on the signal, although don’t worry if you don’t understand the differences between all the filter types. We’ll do more of this exploring in exercise #4.

Exercise 1 – Convert °C to °F

Build a VI

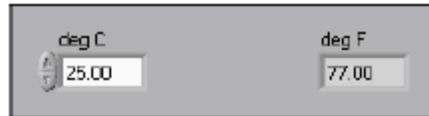


Complete the following steps to create a VI that takes a number representing degrees Celsius and converts it to a number representing degrees Fahrenheit. This should help you gain familiarity with the processes for building your own .vi and getting it to work.

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. Inside the Lab 1 Folder, open Blank.vi. Close the controls window if it starts out open.

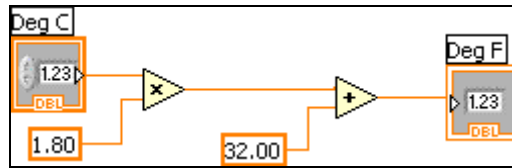


2. Select **Window»Tile Left and Right (or press CTRL+t)** to display the front panel and Block Diagram side by side.
3. Create a numeric digital control by using the following steps. You will use this control to enter the value for degrees Centigrade.
 - a. Select the Num Ctrl on the **Controls»NumCtrls** palette. If the **Controls** palette is not visible, right-click an open area on the front panel to display it (NOTE: right-clicking on the Front Panel and Block Diagram give you different toolbars (Controls and Functions, respectively).
 - 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, which can be found under **Windows»Show Tools Palette**.
4. Create a numeric digital indicator by using the following steps. You will use this indicator to display the value for degrees Fahrenheit.
 - a. Select the Num Ind on the **Controls»NumInds** 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.

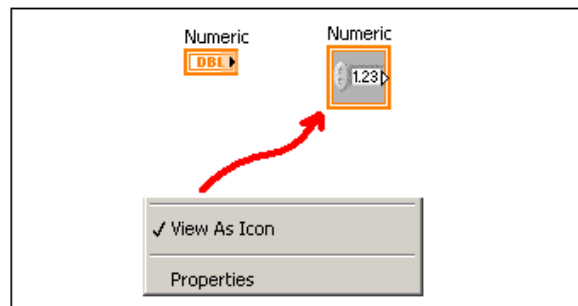
Note: LabVIEW creates corresponding control and indicator terminals on the Block Diagram (which will be on the right side of your screen if you chose to follow step 2 above). The terminals represent the data type of the control or indicator. For example, a DBL terminal represents a double-precision, floating-point numeric control or indicator.

Note: Control terminals have a thicker border than indicator terminals.

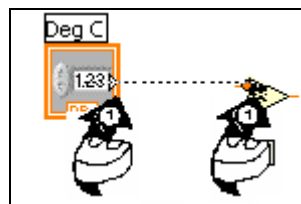
Block Diagram



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**.



5. Bring up the toolbar for the Block Diagram (remember that right click!). Choose **Arith/Compare»Numeric**. Place one of the **Multiply** functions and one of the **Add** functions on the Block Diagram.
6. Select the numeric constant on the **Arith/Compare»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.
7. 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 (**Window»Show Tools Palette**) tool to enter the values.
8. Use the Wiring tool (looks like a spool on the Tools Palate) to wire the icons as shown in the previous Block Diagram. Instructions for using the Wiring tool are given below.
 - a. 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.



- b. 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.
 - c. 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.
 - d. 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.
 - e. To cancel a wire you started, press the <Esc> key, right-click, or click the source terminal. Also, when you're done with the wiring tool (or any other of the tools) you can press space bar to stop using it.
9. Display the front panel by clicking it or by selecting **Window»Show Panel**.
10. Save the VI because you will use this VI later in the course.
- a. Select **File»Save as**.
 - b. Navigate to the folder you created for your group's work and save there.
11. Enter a number in the digital control and run the VI by following the steps below.
- a. Use the Operating tool (looks like a hand) or the Labeling tool (looks like an "A") to double-click the digital control and type a new number.
 - b. Click the **Run** button (same arrow as in Exercise 0) to run the VI.
 - c. Try several different numbers and run the VI again.
12. Save changes. Select **File»Close** to close the Convert C to F VI.

End of Exercise

Exercise 2 – Create a SubVI

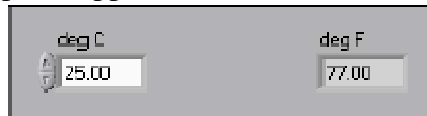
This exercise will give you more experience in building operational .vi files

Front Panel

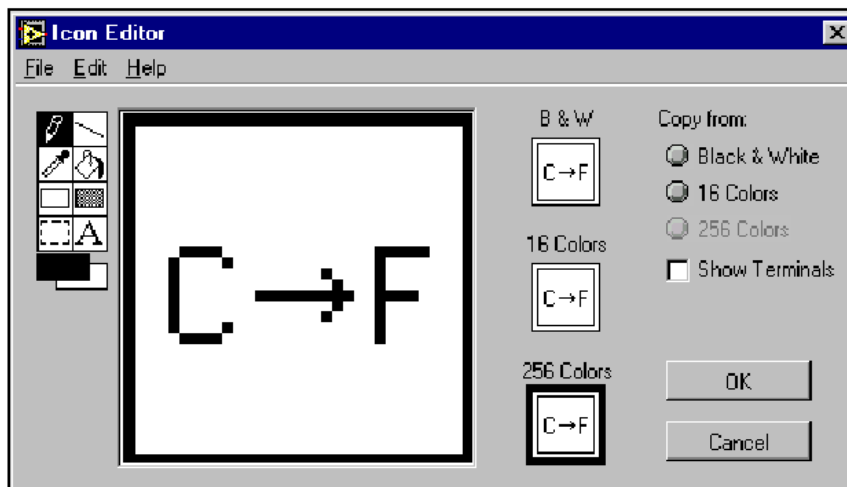
1. Select **File>Open** and navigate to your group's folder to open the Convert C to F VI created in the last exercise.

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 menu bar 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 using the following steps.



- 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 your new 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 by doing the following.
 - 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 (indicating that it is a floating point number), 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.

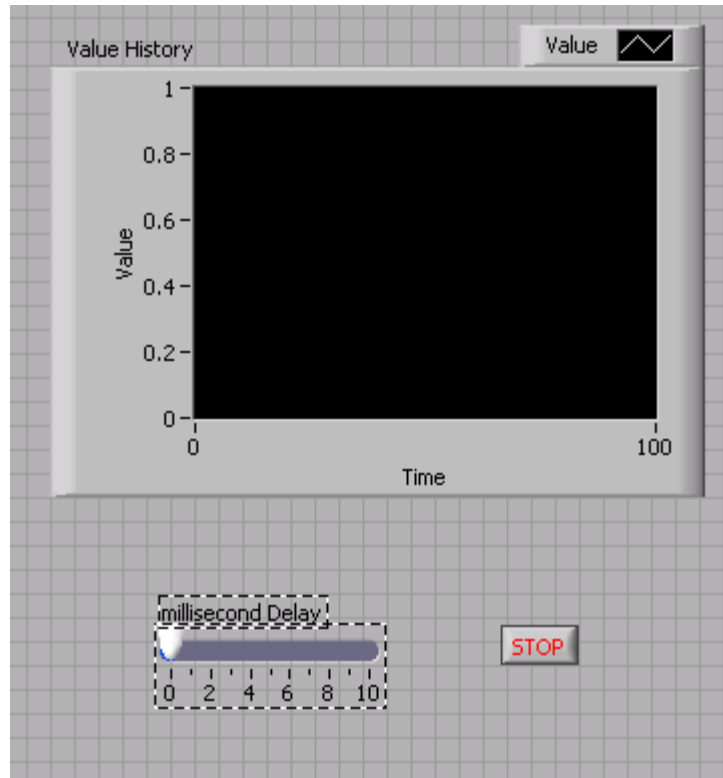
End of Exercise

Exercise 3 – Using Loops

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

Front Panel

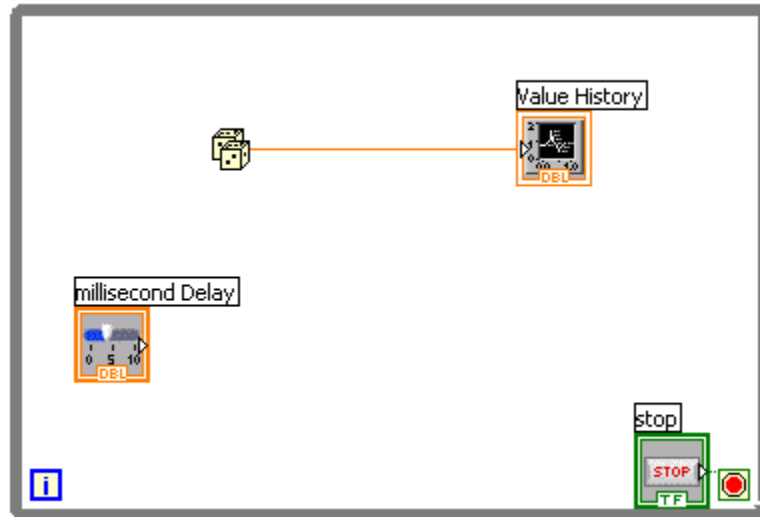
1. Open a new VI.
2. Build the following front panel with the following instructions.



- 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.
- c. 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.
- d. Type `Value History` inside the label and click outside the label.
- e. 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.
- f. 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.
- g. Change `-10.0` in the y-axis to `0`.
- h. Label the y-axis `Value` and the x-axis `Time (sec)`.

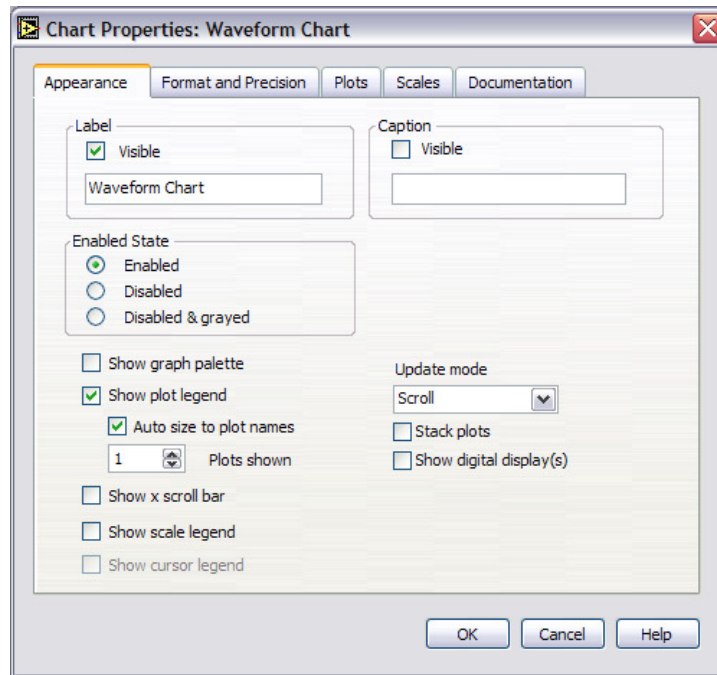
Block Diagram

3. Select **Window»Show Diagram** to display the Block Diagram (or press **CTRL+t**).
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 components of the Block Diagram. A stop button will appear after you draw the loop.
 - 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. The random number generator generates numbers between 0 and 1, and is a nice simulation of a noisy signal for this exercise.
 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.

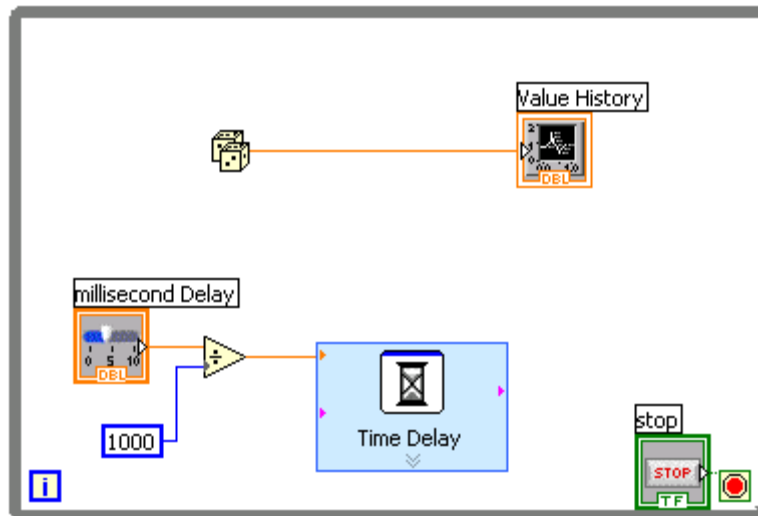
- a. Right-click the chart and select **Properties** from the shortcut menu. The following dialog box appears.
- b. Click the **Scale** tab and select different styles for the y-axis. You also can select different mapping modes, grid options, scaling factors, and formats and precisions. Notice that these 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.



13. 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) as the default value, but we will override this default with a separate input (next step).
14. 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.
15. Save the VI, because you will use this VI later in the course.
16. Run the VI.
17. Try different values for the millisecond delay and run the VI again. Notice how this effects the speed of the number generation and display.
18. Close the VI.

End of Exercise

Exercise 4 – Building Your Own Source and Analyzer

Complete the following steps to build a VI that generates waveforms, filters the waveforms, and analyzes the resulting waveform in the time domain and the frequency domain. This relies on steps you've already carried out.

Block Diagram

1. Open a new, Blank VI
2. On the Block Diagram, select **Functions » Input » Sim Signal**. This creates a variable frequency signal source – something like a virtual function generator. Change settings to Square Wave, 5 Hz, Amplitude = 1, Add Noise On, (uniform noise with amplitude of 0.4). In the timing box, set samples per second to 5000, uncheck the “Automatic” box, and set the Number of Samples to 5,000. If you want to edit the properties later, you can double-click on the icon.
3. Select **Functions » Analysis » Filter**. This creates a variable signal filter that can be used to change the signal from the function generator. Double click on the icon and set it for low-pass, 70 Hz, First-Order Butterworth. The different filter types represent variations in the design of a multiple-order filter that feature different cutoff shapes, different phase response, and different amounts of attenuation near the corner frequency of the filter.
4. Use the wire tool to connect the output of the function generator to the input of the filter.
5. Create a *graph* (not a chart) indicator (from the front panel menu) at the output of the function generator so that you can see the waveform on the front panel.
6. Turn the vi on and look at the waveform as you change the settings for the source.
7. Create a new graph indicator at the output of the filter, so that you can look at the filtered signal.
8. Turn the vi on again, and change the filter settings to see the effect on the waveform. See what happens if the filter frequency is set close to and then below the frequency of your signal. Be sure to look at squarewaves and triangle waves to see the effect of the lowpass filter on the filtered waveform.
9. Select **Functions » Analysis » Spectral** to create a Frequency Analysis of the signal. This function computes the Fourier Transform of the input signal and displays the result. This result appears as a Power Spectral Density as a function of Frequency, and allows easy determination of the frequency distribution of the signal.
10. Use the wire tool to connect the output of the filter to the input of the frequency analyzer.
11. Create a new graph indicator at the output(FFT-(RMS))of the frequency analyzer.

Now, you've built a signal source, a filter, and a signal analyzer, and hooked them all up together. This system can be used to explore the relationship between the input signal and the filtered signal in the time domain and the frequency domain. Carry out the following exercises and report the results in a Lab Report.

A) Measure the frequency content of a square wave. Set the source to 25 Hz Square wave with no noise. Set the filter frequency to 1000 Hz (this should set things so that the filter does not change the waveform at all). Look at the output of the spectral analysis. You will probably need to adjust the plot (click on the plot and open the "properties" window. Adjust the frequency range to 0-250 Hz, and change other settings as needed to allow you to see the response.) You should see a series of peaks in the spectral measurement. What frequencies appear here? Can you explain why these frequencies appear? What is the amplitude of the first 3 peaks?

(Measure by right-clicking on the plot and scrolling to "visible items" and "cursor legend", and then right click on the legend (which may be hidden off screen) and scroll to "properties" and select "cursors" Add cursor and set "allow dragging" and "snap to point". With these settings, you should be able to drag the cursor laterally across the plot and read the values from the display. This allows measurement of the peak heights)

B) Now, change the cutoff frequency of the filter to 100 Hz. Run the vi, and describe the differences in the shape of the waveform at the output of the filter. Measure the peak heights in the frequency plot and compare with the results from before. Explain what is happening.

C) Now, change the filter cutoff frequency to 30 Hz. Change the filter from Butterworth to the other choices, and change the order from 1 to other numbers up to 6. Look at the output of the filter and at the frequency analysis and explain what is happening. For a 4th order butterworth at 30 Hz, what is the height of the first 3 peaks in the frequency plot.

D) Change the filter settings back to 100 Hz cutoff and first-order Butterworth. Change from Squarewave to Sinewave in the frequency source. Describe what happens to the frequency domain plot and explain the differences.

E) Change the source to Sawtooth. What has changed in the frequency plot? There should be some new peaks. Can you explain why these new peaks appear?

F) Now, we're going to explore the relationship between the formation of the function and the ability to analyze the function. In the Simulate Signal Window, The Samples per second and number of samples were both set to 5000. This creates a function (sawtooth in this case) that is digitally generated by changing the "signal" 5000 times per second to follow the 25 Hz waveform, and displays an entire second of data in the window. If we change the Number of samples, the

result is that the amount of data in the result will change. Change to 500 samples and see. Now, close the simulate signal window and look at the result – the Waveform in the input and output windows has fewer oscillations, and the FFT has many fewer points. What is going on here?

Without going into too many details about the FFT, there are a few simple rules to remember. When taking data for a FFT, there are 2 real-time numbers that matter: the interval between samples and the total interval of the scan. In the FFT output, the total range of frequencies that can come out is 1/2 of the sampling rate. (to see this, right click on the fft plot, select properties>>scales and set the x-axis to autoscale. This will force the plot to display the entire range of available data. In this case, the data will go from 0-2500 Hz)

The other simple number is that the interval between data points in the FFT (in Hz) will be the reciprocal of the total interval of the scan. With the present settings (5000 samples per second, 500 total samples), the total interval is 1/10 of a second, and the interval in the FFT is 10 Hz. Since we're interested in seeing the peaks with more resolution than this, we need to increase the total sweep time of the plot. This can be done by reducing the sample rate from 5000 samples/sec to something lower (like 500), or by increasing the total number of samples from 500 to something higher (like 5000). Try these both and describe the difference.

Throughout this quarter, we'll be doing time-domain and frequency-domain analysis of signals. As you've seen, both views are interesting and valuable for determining what is going on with a signal. In the time domain, you can see what it looks like, how big it is, and roughly what the signal/noise ratio is. In the frequency domain, you can see how the signal and noise are distributed, what frequency components are included with the signal, and what things can be accomplished by filtering. To be able to make all this meaningful, you need to know a little about the Fourier Transform, and the point of this exercise is to explore this for the first time (for some of you), and to refresh the details (for some of you). We'll definitely be returning to these kinds of measurements, so be sure to remember that there are a lot of things to set properly for the time and frequency domain displays to be useful.

End of Exercise

Note – in all these exercises, we're exploring how a waveform is constructed of collections of sinewaves at various harmonics of the fundamental frequency. By looking at the frequency domain, we're able to measure the frequencies and amplitudes of those added components that are required to build the squarewaves or other waveforms. As filter settings are adjusted, you're attenuating or eliminating some of the added components that are necessary to construct the waveforms, and the resulting distorted waveforms reflect this removal of needed components.

During this quarter, we're going to be discussing signals and noise, and one important method for separating them (reducing the noise and increasing the signal) is by filtering and by looking at the results in the time domain and the frequency domain. We're going to rely a lot on the frequency domain to understand what the noise and signals look like and to help improve the signal/noise ratio. So, these first exercises are important for beginning to build some expertise about frequency domain measurements.