

Name: _____

EE354 Spring 2016 Lab 1: Introduction to Lab Equipment

In this lab, you will be refreshed on how MATLAB and the lab hardware can be used to view both the time-domain and frequency-domain version of signals.

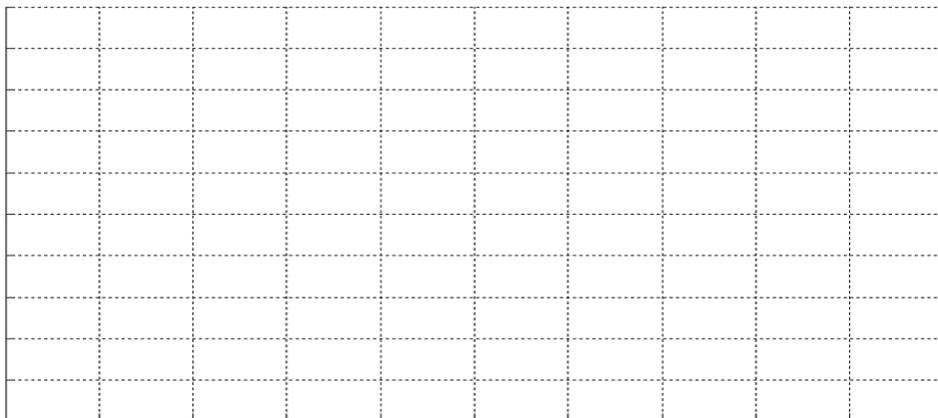
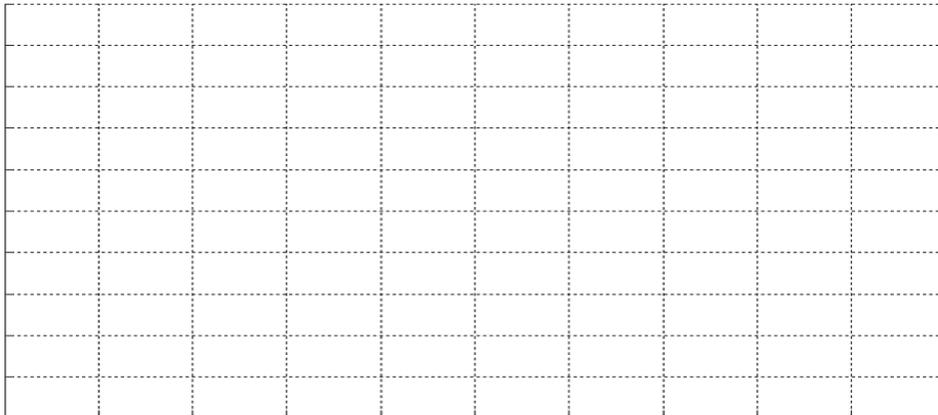
Part 1: Pre-Lab

Before beginning any lab experiment, it is imperative to predict the results we should observe from the hardware. These predictions give us a basis for determining: (a) whether the hardware is operating properly, and (b) how closely our experimental results match our theoretical predictions.

In this experiment, we will be working with a 1.1 MHz sinusoid with a DC offset. Mathematically, our signal is described by the following equation:

$$s_1(t) = 0.5 \cos(2\pi(1.1 \text{ MHz})t) + 0.25 \text{ V}$$

In Table 1 (on page 6), record the theoretical values for the waveform's minimum, maximum, DC offset, amplitude, peak-to-peak amplitude, period, and frequency. In the space below, sketch the time-domain and magnitude frequency spectrum of this signal (remember, graphs have axes, labels, and titles).



Due at the beginning of the lab period prior to embarking on the lab.

Instructor Verification / Score: _____

Part 2: Simulation

First, we're going to create and display a 1.1 MHz sinusoid using the following MATLAB commands.

Note: These values are not "magic numbers"; they have been chosen for specific reasons, many of which we will cover over the semester. For now, go ahead and use them as they are "known good" values.

<pre>» fs = 12e6; » N = 16384; » Ts = 1/fs; » t = 0:Ts:N*Ts; » f0 = 1.1e6; » A = 0.5; » s1 = A.*cos(2*pi*f0*t) + 0.25;</pre>	<pre>% "fs" is sampling frequency. % "N" is the number of samples of the sinusoid % "Ts" is time between samples. % "t" is the time vector; more on this later % "f0" is the frequency (in Hz) of the sinusoid % "A" is the amplitude of the sinusoid, 0.5 Volts % "s1" is our signal vector (a 1.1 MHz sinusoid)</pre>
--	---

Plot the sinusoidal signal as a function of time. When plotting the signal, the following plot options/commands may prove useful:

<pre>» plot(t,s1,'k-', 'linewidth',2) » axis([xmin xmax ymin ymax]) » xlabel('label (units)') » ylabel('label (units)') » title('Title')</pre>	<pre>% These plot options produce lines of different % colors/styles, as well as a thicker line that's % easier to see. % Sets the amount of the waveform that's visible % in the figure window so that you don't get a % big blue (or black, in this case) blob. % I suggest setting xmax to 10 µsec or less. % All figures should have meaningful axes labels % and titles, lest you incur the wrath of the % EE354 instructor.</pre>
--	---

Printout a copy of your plot and turn it in with the completed lab.

Next, we need to display the spectrum of our signal. Frequency content of sampled signals on a computer are calculated using the Fast Fourier Transform (FFT). In this class we're not worried about the underlying operation of the FFT (that's reserved for EE432), and thus, we're going to use the Matlab function `spec_analysis.m` (available on the course website) to find the frequency spectrum of a signal. For now, use 50Ω as the impedance (we will discuss why later).

<pre>» [freqT,pwrT] = spec_analysis(s1,fs,50); » plot(freqT,pwrT,'k-' 'linewidth',2) » axis([xmin xmax ymin ymax]) » xlabel('label (units)') » ylabel('label (units)') » title('Title')</pre>	<pre>% Computes the Theoretical Spectrum % Plot the theoretical spectrum and set the % axis to a reasonable range. % All figures should have meaningful axes % labels and titles, lest you incur the % wrath of the EE354 instructor.</pre>
---	---

Your output should look like the figure below:

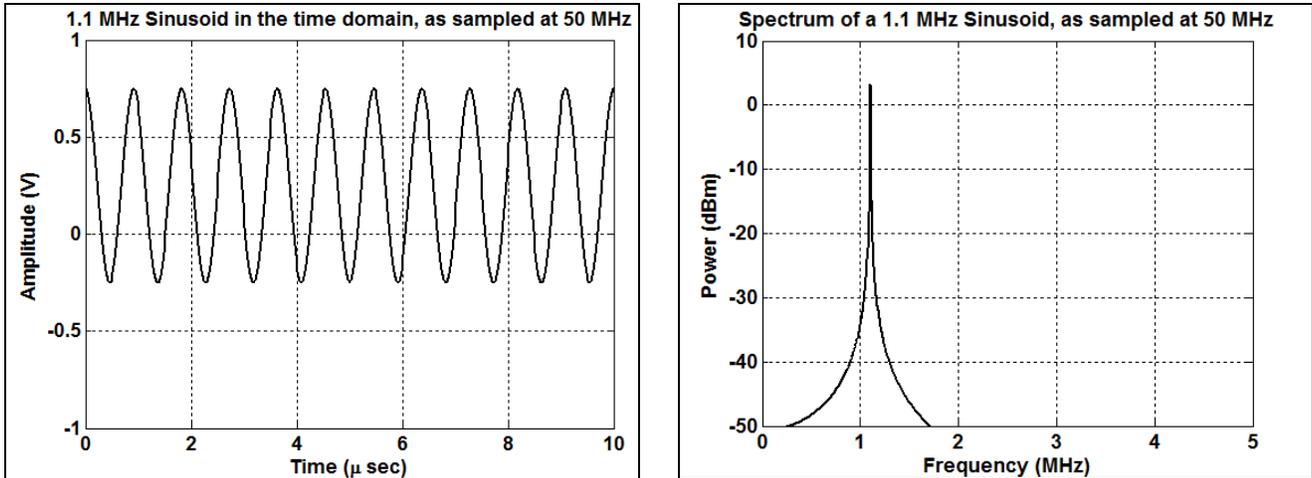


Figure 1: Time Domain and Frequency Spectrum of a single 1.1 MHz sinusoidal tone signal, $f_s = 50$ MHz.

We haven't talked about dB units yet, but don't worry, we will soon! Also recall the time/frequency resolution properties of the Discrete Fourier Transform (higher frequency resolution means a longer sampling time/larger number of samples, NOT a higher sampling frequency). You can observe this yourself by reducing N to 32 points and re-running the code above.

Printout a copy of your plot and turn it in with the completed lab.

Before proceeding, have the instructor check your work.

Instructor Verification: _____

Part 3: Hardware Measurements of the Sinusoidal Signal via the Oscilloscope

The quick reference guide for the LeCroy Oscilloscope is posted on the main EE354 website. Before proceeding, download a copy of the quick reference guide and open it in a new window to use as a reference for operating the oscilloscope.

1. Using one of the Agilent 33220A Function Generators, create a 1.1 MHz sinusoid with a peak voltage of 0.5V. Take a few moments to become familiar with the front panel controls of the function generator. Learn how to select waveshapes, amplitudes, and frequencies using the keypad and the control knob.
 - a. Press the *Sine* button
 - b. Toggle the *Freq* tab until the default frequency is displayed. Enter “1.1” on the keypad, and then press the *MHz* tab.
 - c. To set the amplitude, we have to enter either the RMS value of our signal or the peak-to-peak amplitude of our signal. Toggle the *Ampl* tab until the default amplitude is displayed. Enter “1.0” on the keypad, and then press the *V_{pp}* tab.
 - d. To set the DC offset, toggle the *Offset* tab until the default DC offset is displayed. Enter “0.25” on the keypad, and then press the *V_{DC}* tab.
 - e. Press the *Graph* button and verify that the generator is producing a 1.1 MHz sinusoid with peak-to-peak amplitude of 1.0 Volts, and 0.25 Volt DC offset.
 - f. Press the *Output* button and ensure that it is lighted. Only when the *Output* button is lit will the signal generator produce an output signal.
 - g. Using one of the BNC cables, connect the output of the signal generator to Channel 1 of the LeCroy oscilloscope, as shown in the figure below.

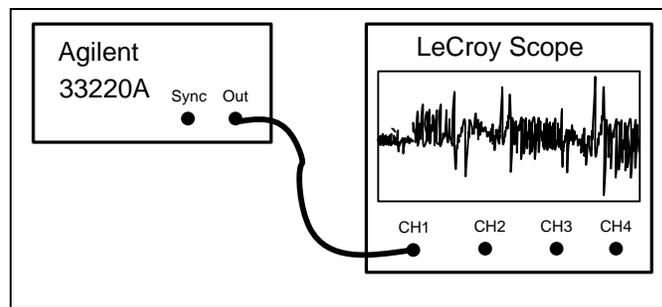


Figure 2: Agilent Signal Generator to LeCroy Scope Connection.

What is the maximum frequency and maximum amplitude sine wave that the function generator can produce?

2. Use the LeCroy scope to display the signal. Learn how to select channels to display, and how to get a good display without using the “Autoscale” button (Autoscale does not do anything you cannot do manually, and there is no guarantee you will get the display settings you need). Most of the information you need can be found in the “Quick Reference Guide” posted on the course website.
 - a. Before beginning a measurement, it is always a good idea to return the instrument to its default configuration. For the LeCroy scope, press *File* → *Recall Setup* → *Recall Default*.
 - b. The default configuration will bring up both Channel 1 and Channel 2. In this case, we do not need to display Channel 2. Select the **C2** Tab at the bottom left-hand corner of the screen, and then un-check the “Trace On” box.
 - c. Next, we need to properly set the input impedance of the scope to match the output impedance of the function generator (for reasons that will become clear in EE372); that is, since the function generator output is $50\ \Omega$, set the scope input to have $50\ \Omega$ impedance vice $1\ \text{M}\Omega$. Select the **CI** Tab at the bottom left-hand corner of the screen, and then under the “Coupling” listbox, select “DC50 Ω ”.
 - d. Next, use the horizontal and vertical control knobs to adjust the display to show several cycles of the sinusoid. Reasonably good values are 200 mV/Div on the vertical scale and 500 ns/div on the horizontal scale.
 - e. Use the **OFFSET** knob to move the waveform into the center of the scope screen. When completed, you should have a display that looks similar to the following:

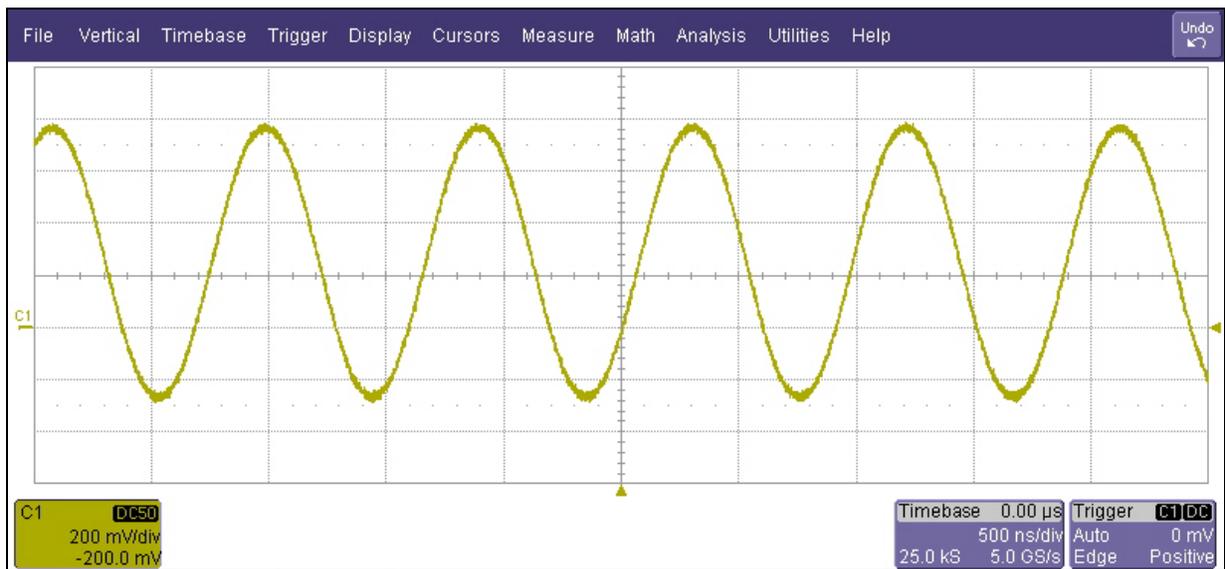


Figure 3: LeCroy Oscilloscope screen when properly set up and configured for this lab.

A couple of things should be pointed out before we continue. First, the “C1” marker on the left-hand side of the screen indicates the 0V point of the scope. Because our waveform has a 0.25V DC offset, it is not centered on the marker. If you switch the coupling to “AC1M Ω ”, you will notice that the waveform gets larger (a result of the impedance mismatch) and is centered on 0V (due to the scope blocking the DC offset). Note that this is not the correct representation of our waveform! On the right-hand side of the screen, the arrow marker indicates the trigger level for the waveform. If you adjust the **LEVEL** knob, you will adjust where the signal is triggered (it will respond by moving left and right on the screen), and if you adjust the knob to a value too high or too low, the scope will no longer trigger, and the waveform will “dance” on the screen.

Note: Every time you make a measurement using the oscilloscope, you **must** pay attention to how the input is coupled, how the waveform is acquired, how the oscilloscope is triggered, and the sampling rate being used. If you do not, you could end up displaying on the screen a waveform that in no way represents the signal you are trying to measure.

3. Use the LeCroy scope to manually perform some measurements on the signal using the vertical and horizontal cursors.
 - a. Go to the *Cursors* → *Horizontal(Time)* menu to turn on the horizontal cursors. Note that the cursor value will be displayed in the lower right-hand corner and lower left-hand corner of the scope screen.
 - b. Use the **CURSORS** knobs to measure the period and frequency of the sinusoid and record them in the table below.
 - c. Go to the *Cursors* → *Vertical(Amplitude)* menu to turn on the vertical cursors.
 - d. Use the **CURSORS** knobs to measure the peak amplitude, peak-to-peak amplitude, maximum, and minimum values of the sinusoid and record them in Table 1.

4. Use the LeCroy scope to automatically perform some measurements on the signal using the Measure functions.
 - a. Go to the *Measure* → *Measure Setup* menu to bring up the built-in measurement functions.
 - b. The scope will allow you to display up to six different waveform measurements, labeled “P1” through “P6”. At the moment, they should all say “None”
 - c. Select measurement “P1”, which should bring up a menu of the possible operations. Scroll down until you see “Period”, and then select it. Note that the scope will then compute and display the measurement on the screen under the waveform, along with a green check-mark. The green check-mark means that the scope believes the measurement to be good quality and valid data; a red x-mark means that the measurement is of poor quality and possibly invalid.
 - d. Repeat steps b and c and setup P2-P6 to measure: Frequency, Maximum, Minimum, Average, and Peak-to-Peak amplitude. Record the scope automated measured values in Table 1.

5. Use the LeCroy scope to display the frequency spectrum of the signal using the Math functions.
 - a. On the Oscilloscope select *Math*→*Math Setup*. Set the operator to FFT Magnitude, and verify that the FFT is operating on the channel that contains your signal. Then check the “Trace On” box.
 - b. As we have seen before, the FFT computes frequency content from 0 Hz to $f_s/2$ Hz, where f_s is the sample frequency. This means that the left end of the display is set for 0 Hz, and the right end is set for $f_s/2$ Hz. The current sample frequency is shown on lower right corner under “Timebase”, in units of Sa/s (which means samples/sec). Because the sampling frequency is likely 5.0 GHz, your frequency spectrum likely appears as a big orange blob.
 - c. Select Channel 1 by selecting the *CI* tab. Adjust the **HORIZONTAL** timebase knob until the sampling frequency (which is displayed under the “Timebase” block in the lower right-hand corner of the screen) displays a rate of 10 MS/s, or a timebase of 10.0 μ sec/div.
 - d. Next, set the scope to record exactly 1,000 samples of the waveform by selecting *Timebase* → *Max Sample Points* → *1.0 ks*. You may have to readjust the timebase knob to get back to 10 MS/s.
 - e. Use the Horizontal cursors to measure the frequency of your sinusoid and verify that it matches the time-domain measurement. When finished, your screen should look like the following:



Figure 4: LeCroy Oscilloscope screen when properly set up and configured to perform spectrum measurements.

6. Save the LeCroy scope data in a format that can be used by Matlab.
- Note:** You must use an ITSD-Approved External Hard Drive to save your data.
- a. Save the time-domain waveform by going to the *File* → *Save Waveform* menu and making sure the following options are selected:
 - i. Save To: File
 - ii. Source: C1
 - iii. Format: Matlab
 - iv. Autosave: Off
 - b. Give the trace a meaningful title, and press the Save Now button.
 - c. Save the frequency spectrum by going to the *File* → *Save Waveform* menu and making sure the following options are selected:
 - i. Save To: File
 - ii. Source: Math
 - iii. Format: Matlab
 - iv. Autosave: Off
 - d. Give the trace a meaningful title, and press the Save Now button.

Table 1: Theoretical and Measured Characteristics of the 1.1 MHz Sinusoid

Parameter	Theoretical Value	LeCroy Cursor Measurement	LeCroy Automated Measurement	Matlab Measurement
Minimum				
Maximum				
Amplitude				
Peak-to-Peak				
DC Offset				
Period				
Frequency				

How do the two LeCroy measured values compare with the theoretical values? What is the average percent difference between the measured values and the theoretical values?

Part 4: Comparison of the Theoretical and Measured Sinusoidal Signals

To load the saved oscilloscope data into Matlab (for further analysis or plotting), first copy the wave data from the external hard drive into Matlab's current working directory.

Next, Issue the following commands (I **highly** recommend writing a function that does this, as you will be performing the following operations repeatedly throughout the course):

First, start with your time-domain sinusoid.

```
» load('filename.dat'); % Loads waveform data, replace "filename.dat" with
» % the name of your time-domain file.
» time = filename(:,1); % The time axis (in sec) is stored in the first col.
» amp = filename(:,2); % Amplitude info (in V) is stored in the second col.
```

Next, plot the waveform and adjust the axes (see the sections above for instructions) to show a few cycles of the sinusoid. Use the Matlab commands `min`, `max`, and `mean`, to fill in the time-domain measurement values for Table 1 (we'll get to the frequency and period measurements in a moment).

Once you are satisfied that your captured signal was successfully loaded into Matlab, generate a single plot that shows both the theoretical signal as well as the scope captured signal on the same plot. Use a different identifying feature (whether line style, line width, or line color) to differentiate the two signals. You can type "help plot" if you need to refresh your memory. Add a legend to your plot to clearly identify the two signals (type "help legend" if you need to refresh your memory). Don't forget to add meaningful axes labels and titles. When finished, your plot should look similar to the following (note that phase offsets will prevent the two signals from exactly lining up).

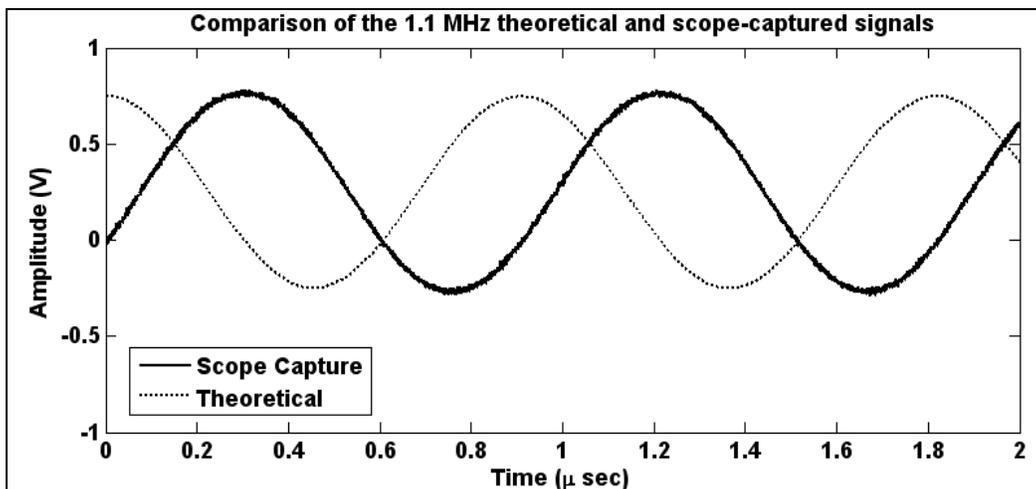


Figure 5: Example comparison of the Theoretical and LeCroy Oscilloscope Captured Sinusoids

Printout a copy of your plot and turn it in with the completed lab.

How closely does the time-domain sinusoid captured from the oscilloscope match the theoretical sinusoid? What, if anything, causes the differences?

Next, we'll observe the frequency spectrum as captured by the scope and compare it with the theoretical spectrum. Use the following Matlab commands to load the FFT capture into your Matlab workspace.

```
» load('filename.dat'); % Loads waveform data, replace "filename.dat" with
» % the name of your frequency-domain file.
» freq = filename(:,1); % The frequency axis (in Hz) is stored in the first col.
» pwr = filename(:,2); % Amplitude info (in dB!) is stored in the second col.
```

Next, plot the waveform and adjust the axes (see the sections above for instructions) to show the frequency spectrum of the signal. Once you are satisfied that your signal was successfully loaded into Matlab, use the following series of commands to plot both the theoretical spectrum and scope-captured spectrum on the same figure.

```
» figure % Sets up the figure
» subplot(2,1,1) % freqT/pwrT are Theoretical Spectrum
» plot(freqT,pwrT,'k-', 'linewidth', 2) % Plot the theoretical spectrum and set the
» axis([X X X X]) % axis to a reasonable range.
» subplot(2,1,2) % Plot the measured spectrum and set the
» plot(freq,pwr,'k-', 'linewidth', 2) % axis to a reasonable range.
»
```

Use the Matlab command `max` and/or the Figure Data Cursor  (the button on the Matlab figure) to determine the frequency and period of the captured spectrum, and record these values in Table 1. When finished, your figure should look something like the one below:

Figure 6: Comparison of the theoretical and LeCroy Oscilloscope captured spectra of the sinusoidal signal.

Printout a copy of your plot and turn it in with the completed lab.

How closely does the frequency spectrum of the sinusoid captured from the oscilloscope match the theoretical spectrum? What, if anything, causes the differences?

Part 5: Hardware Measurements of the Sinusoidal Signal via the Spectrum Analyzer

The quick reference guide for the Anritsu Spectrum Analyzer is posted on the main EE354 website. Before proceeding, download a copy of the quick reference guide and open it in a new window to use as a reference for operating the oscilloscope.

Disconnect the Agilent 33220A Function Generator from the Oscilloscope and connect it to the input of one of the Anritsu MS2711D spectrum analyzers, as shown in the figure below. Also, connect the Anritsu to the PC using the RS-232 cable at the lab bench.

NOTE: The very first thing that will likely happen is that the spectrum analyzer will produce an **“OVER RANGE! Increase Attenuation”** warning on the screen. We shall deal with that momentarily.

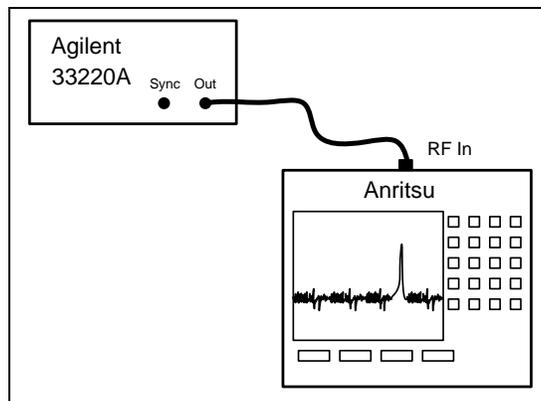


Figure 7: Agilent Signal Generator to Anritsu Spectrum Analyzer Connection.

For reference, Figure 2-2 from the quick reference guide is provided in Figure 8.

1. Adjust the spectrum analyzer to get rid of the Over Range Warning
 - a. Press the *AMPLITUDE* function hard key, select the *Ref Level* soft key, enter “10” on the numeric keypad, and then press *ENTER*. At this point the “*OVERRANGE*” warning should go away.
2. Configure the spectrum analyzer to display the spectrum of our sinusoid. The spectrum analyzer will display a 3.0 GHz range of frequencies, which is too large to view the 1.5 MHz sinusoid, so we need to adjust the frequency range. You may also notice an
 - a. Power on the spectrum analyzer and return it to the default configuration by pressing *Recall Setup* (which is keypad hard key “6”), use the up/down arrow keys to select “0 --- <Default>”, and then press *Enter*.
 - b. Press the *FREQ/SPAN* function hard key, then select the *Start* soft key, enter “0.75” on the numeric keypad, and then press the *MHz* soft key.
 - c. Select the *Stop* tab, enter “1.25” on the numeric keypad, and then press the *MHz* soft key.
 - d. At this point, something approximating the spectrum of the sinusoid should appear on the screen.

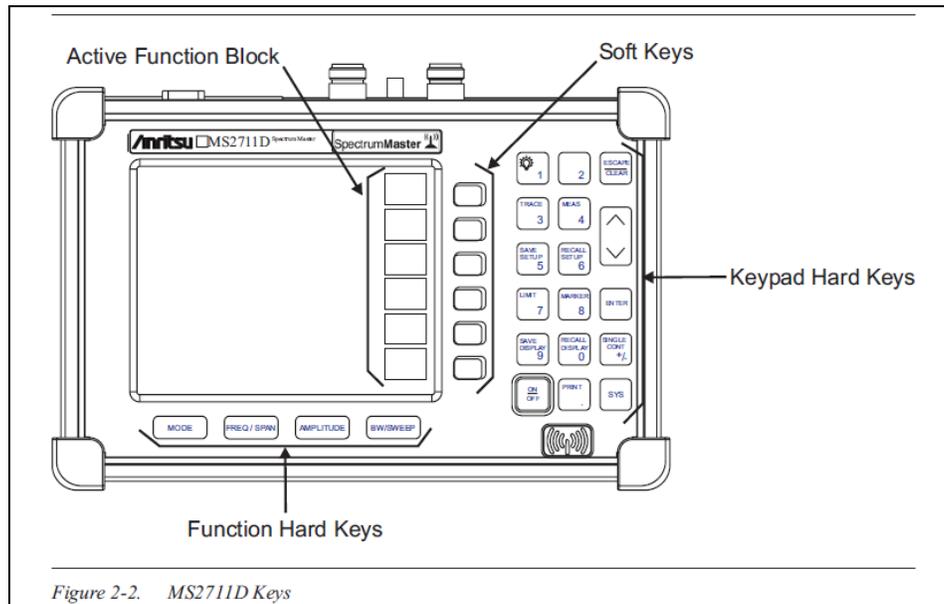


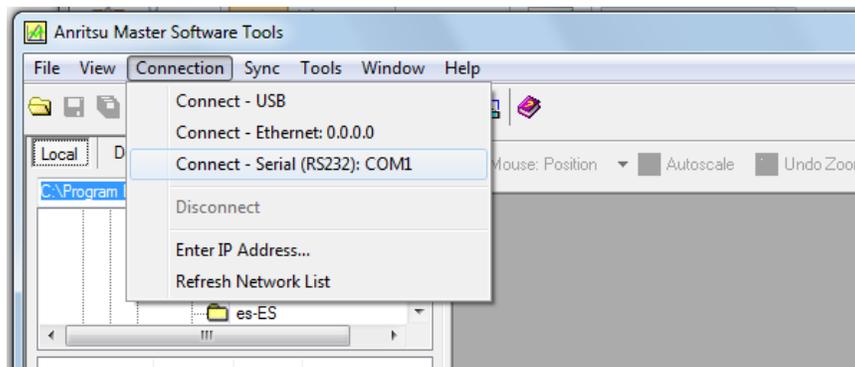
Figure 8: The MS2711D Instrument Overview

Note: The Frequency Start, Stop, and Reference Level (Ref Level) values control how the signal spectrum is displayed on the screen, much like the horizontal and vertical time scales control how the time-domain signal is displayed on the oscilloscope. The Reference Level represents the maximum signal *power level* (in dBm, which we will learn about shortly), at the *very top* of the screen. If you adjust the Reference Level, you should be able to move the spectrum up and down on the screen (try it by selecting “20” or “-20” and see what happens). The Frequency Start and Stop control how much of the spectrum is shown on the screen. Setting these too wide will obscure very important features of your signal, setting these too narrow will cause a complex spectrum to look like a single sinusoid. Try setting the stop frequency to 100 MHz, then 1.0 GHz and observe what happens to the displayed signal spectrum.

3. Use the spectrum analyzer markers to measure and verify the frequency of the sinusoidal signal. The Anritsu has four (4) independent markers which can be turned on or off individually.
 - a. Press the *Marker* (which is keypad hard key “8”) button, select the *M1* soft key, and toggle the *On/Off* soft key until the marker is turned on.
 - b. The marker will likely appear as a red line to the left- or right-hand side of the screen. Use the up/down arrow keys to move the marker until it is centered on the peak of the sinusoid. You may find it useful to adjust the frequency range of the spectrum analyzer to zoom in on the sinusoid. The marker frequency and amplitude are displayed in the lower left-hand corner of the Anritsu screen (the “message area”) under the “M1” label.
 - c. Verify that you have the peak value by pressing the *Marker to Peak* soft key.
 - d. Record the amplitude and frequency of the Marker in below.

Frequency: _____ **Power Level:** _____

4. Now, transfer the spectrum analyzer plot to the PC using the following steps.
 - a. On the PC, select **Start → Anritsu → Master Software Tools**
 - i. Sometimes, the Anritsu software does not install to the Start Menu. If this is the case, then navigate to C:\Program Files (x86)\Anritsu\Master Software Tools and double-click on the “MST” application.
 - b. On the Menu Bar, select **Connection → Connect Serial (RS-232): COM1** as shown below
 - c. On the Menu Bar, select **Sync → Capture Current Measurement**.



- d. After the program established communication with the spectrum analyzer, your screen should display a replica of what’s currently displayed on the spectrum analyzer.
- e. Note that Marker 1 is currently displayed on the screen. You can use the mouse to move the marker around, and you can add any of the M2-M6 markers if you need to make additional measurements. Your completed spectrum should look like the one shown below.

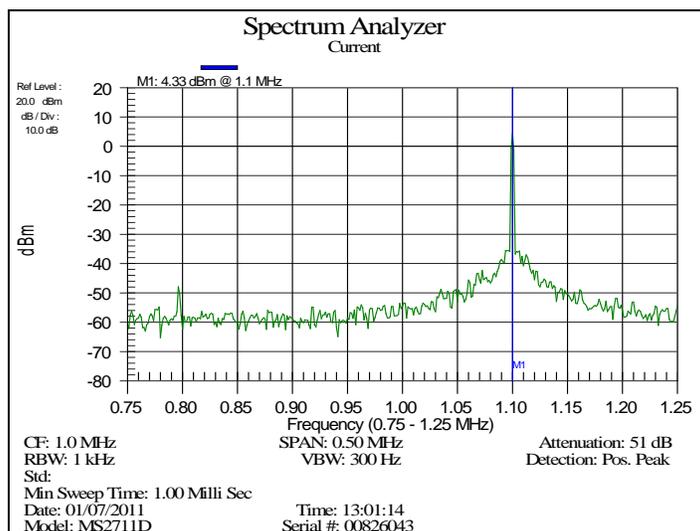


Figure 9: The Sinusoidal Signal Spectrum as recorded by the MS2711D Spectrum Analyzer.

5. We can also export spectrum analyzer data to Matlab in order to make nice, neat, crisp plots just like we did with the Oscilloscope. Go to **File → Export CSV**, give the trace a meaningful title, and press the **Save** button.

6. Note that a CSV file (which stands for “Comma Separated Values” is readable by Microsoft Excel. Open the trace in Excel, and you will notice that it contains a significant amount of header information. For the purposes of plotting, the data we’re interested in is the sequence of (*freq / power*) measurements. Scroll down until you determine the row/column where the data begins. In the example below, the data starts at **B129**. The spectrum analyzer always saves 400 points, so the range will be **B129 – B529** (for the power measurements) and **C129 – C529** (for the corresponding frequency points).

	A1		fx	<Anri
	A	B	C	D
127	# Begin SPA Data			
128	<APP_DATA>			
129	P_0	-87	924.499968	MHZ
130	P_1	-87.012	924.502468	MHZ
131	P_2	-86.952	924.504968	MHZ
132	P_3	-87.16	924.507468	MHZ
133	P_4	-86.7	924.509968	MHZ
134	P_5	-86.672	924.512468	MHZ

7. We can now use the Matlab command `dlmread` to load the data in Matlab for plotting. Use the following commands (as modified to fit your data) to load the spectrum analyzer data into Matlab.
- Note:** Even though the Anritsu only outputs 400 data points, the way in which the file is formatted causes Matlab to load zeros on every other row. So we have to do a little manipulation in order to extract out the values we’re interested in.
 - Once you’ve successfully loaded it into Matlab, generate a plot and compare it to what you saw on the Anritsu Master Software Tools suite.

```

>> x = dlmread('filename.csv', ',', 'B129..C929'); % Loads waveform data, replace
>> % "filename.csv" with the name of
>> % your spectrum analyzer file.
>> %
>> % Replace the range 'B129..C929'
>> % with the range specific to your
>> % file (note range is doubled)
>>
>> freq = x(1:2:800,2); % Frequency in MHz
>> % (data in every other row!)
>> pwr = x(1:2:800,1); % Power in dBm
>> % (data in every other row!)

```

Printout a copy of your Matlab spectrum and turn it in with the completed lab.

8. Now, we're going to explore one final feature of the spectrum analyzer: how the Span and Resolution bandwidth impacts the way the spectrum is displayed on the screen. We haven't (yet) described AM Modulation, but at this point, all you need to know is that the signal you're about to create will have a spectrum that contains three distinct peaks, one at 1.1 MHz, one at 1.101 MHz, and one at 1.099 MHz.
 - a. On the Agilent Function Generator, press the *Mod* button, toggle the *Type* tab until it reads "AM", toggle the *Source* tab until it reads "Internal", then press the AM Depth tab and set it to 75%.
 - b. Next, select the *AM Freq* tab and set it to 1.0 kHz, and toggle the *Shape* tab until it reads "Sine".
 - c. You will probably not notice any meaningful change in the spectrum analyzer's display, because the span is set too large to actually see the waveform. We need to zoom in on the signal in order to differentiate the peaks.
 - d. On the spectrum analyzer, first, press the *FREQ/SPAN* hard key, select the *Center* soft key, enter "1.1" on the numeric keypad, and then press the *MHz* soft key.
 - e. Select the *Span* soft key, enter "10" on the numeric keypad, and then press the *kHz* soft key.
 - f. At this point, you will likely see all three peaks on the screen, with the 1.1 MHz in the center of the screen, and each of the other peaks one horizontal division to the right or left. What we have done is center the spectrum analyzer around our frequency of interest, and set the screen to show a total span of 10 kHz (i.e., 1.0 kHz per horizontal division). Change the span and observe how you can zoom in or out on your signal of interest.
 - g. The last feature of the spectrum analyzer that will be explored in this lab is the Resolution Bandwidth. The spectrum analyzer operates by sweeping across the frequency range set on the screen and measuring the power level at a given point in that range. The spectrum analyzer, however, does not have infinitely fine resolution, and its ability to show distinct components is set by the Resolution bandwidth. Signals that fall within the Resolution bandwidth will be displayed as a single frequency, potentially giving the viewer inaccurate results.
 - h. Press the *BW/Sweep* hard key, followed by the *Bandwidth* soft key, then the *RBW Manual* soft key. Use the Up/Down arrow keys to change the value to 1.0 kHz, and then press *ENTER*.
 - i. On the screen, you'll notice that your waveform has been resolve as a single, large blob of spectrum (you may see some "lumpiness" that looks like the two side signals). This is because the Resolution Bandwidth is set too large to effectively show the two side signals.
 - j. Using the procedure above, change the Resolution Bandwidth several different values and observe the effect on your signal. When finished, reset the Resolution Bandwidth to "Auto" (*BW/Sweep* → *Bandwidth* → *RBW Auto*).
 - k. Use the markers to measure the frequency and power levels associated with the three signals on the screen and record the values below.

Frequency	Power Level

Printout a copy of your spectrum and turn it in with the completed lab.

Items to Turn In

Completed Lab

Matlab Code

Plot of the Matlab generated time-domain version of a single sinusoid

Plot of the Matlab generated frequency spectrum of a single sinusoid

Plot of the comparison between the simulated and scope-captured time-domain signal.

Plot of the comparison between the simulated and scope-captured spectrum.

Printout of the Anritsu captured spectrum of the single sinusoid

Printout of the Anritsu captured spectrum of the AM signal.